Shape optimisation of an Automotive Rim using FEA and mesh morphing

Hasnain Habib

Supervisor: Marco Biancolini

Engineering Sciences

University of Rome Tor Vergata.

Introduction

The work has an objective to study the feasibilty of a component made of unconventional materials.

The idea of this type of a product came from it's importance in the commercial market.

To carry out the project we used Finite Element Mehod,using Ansys Mechanical.

Standards Used

Certification TÜV

The TÜV Rheinland Group is a world leader in the field of assessment and evaluation of products and managment systems.TÜV Rheinland checks and provide certification services in the field of safety and product quality for almost all categories of goods: toys,equipment for information technology and heavy industrial machinery.

Following is a brief description of tests required in order to obtain the approval from TÜV. Such tests are purely experimental and should be perfomed directly on the Rim on a test bench.At the end of each test The rim should not develop any ruptures,cracks,loosening of bolts.

There are four different types of tests including three Fatigue with alternating symmetrical loads,and one of impact.

The tests are classified as follows:

Try Alternating Bending

The purpose of the test is to simulate the lateral forces acting on the wheel when the vehicle travels a curve,which in turn establishes a bending moment on the rim,with a longitudinal axis relative to the wheel.The standard provides for the determination of a maximum bending moment calculated according the following relationship:

$M_{max} = f. Fr.(\mu.r+e) = 2844Nm$

f = 2 is a factor of safety;

Fr= 650 kg is the static load of wheel admitted;

μ =0.9 is coefficient of friction tire / asphalt;

r = 0.240 m is the outer radius of the largest tire provided;and

e = 0.007 m is the offset of the rim;

Try Rolling

This test is conducted in such a way as to simulate the stress rotating due to the weight of the vehicle when it travels the straight line.The vertical weight directed downward is determined by the following relationship:

$F_p = f.F_r$

f = Safety Factor

F_r = Static load admitted

Try alternating torque

The simulated torques during acceleration and braking.The rim is then subjected to time alternating symmetrical for a number of cycles.The moment agent is calculated as follows:

$M_t = f.F_r.r$

r = outer radius of the tire

Side Impact Test

The only non-fatigue test,to verify the impact resistance on the edge of the rim.

Rim Models

Model of Aluminum rim

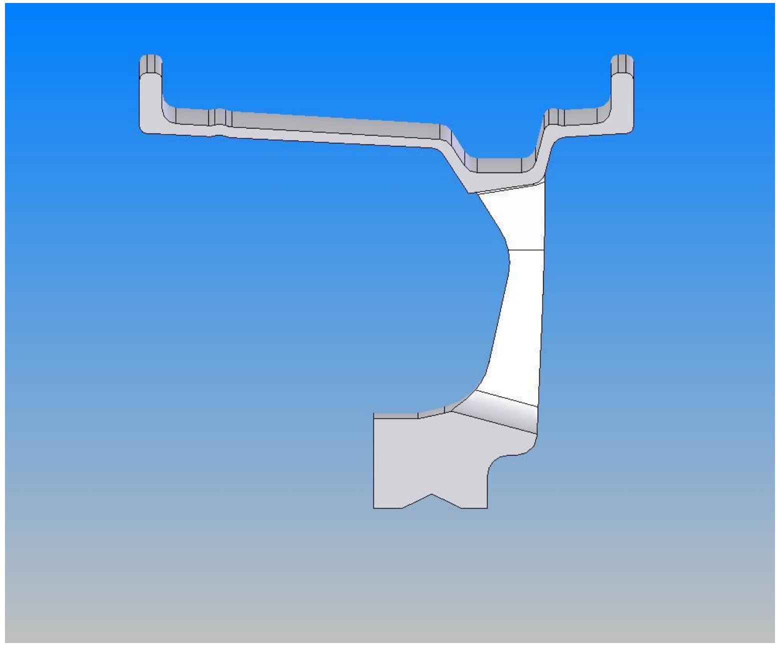
Below is the description of the modulation techniques for the generation of the aluminum rim.

The rim is constructed of aluminum alloy AlSi7Mg with a size of 6x13",are fastened with a central nut(hole diameter 58mm) and an offset of 7mm

For the purpose of having a reliable reference,the 3D model was generated from a scratch rim through software SOLIDWORKS.

The component is a solid axisymmetric with 12 breeds/races and for this reason it was convenient to use symmetry,and generate only a single race with the intent of mirroring it to get a full circle.

The modeling techniques used for the generation of single breed /race was that of LOFT, which allows to define the solid from two or more straight surfaces, and a guide line that allows greater control of the profile.



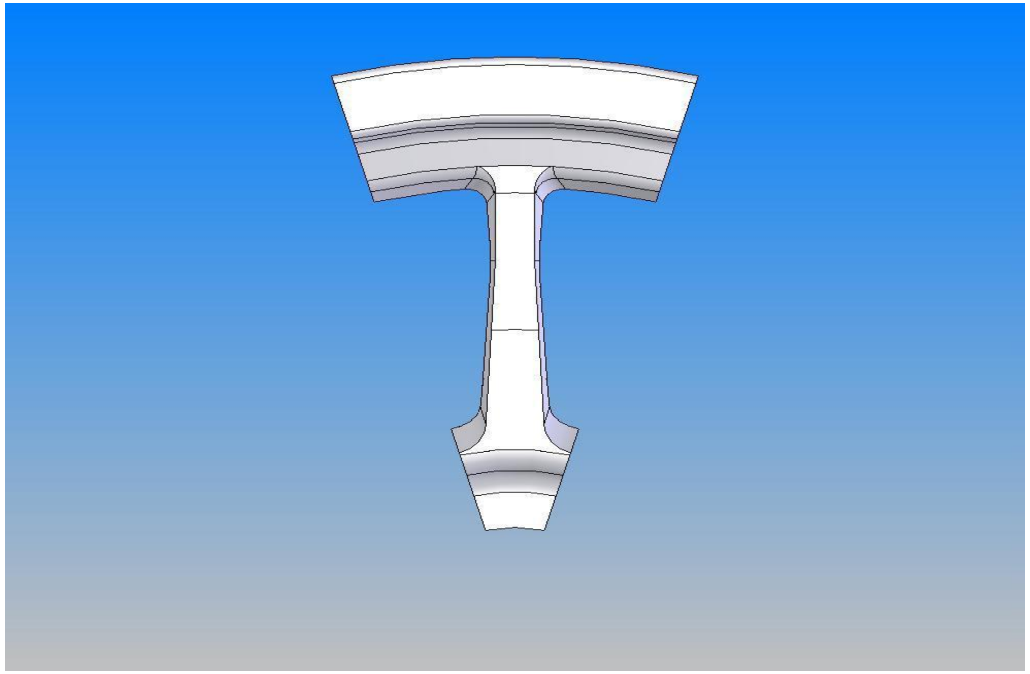


Fig 1. Detail of a single race rim(aluminum)

With the portion of the rim on the single race we are able to perform the radial repetition to generate the full circle (see Fig 2)

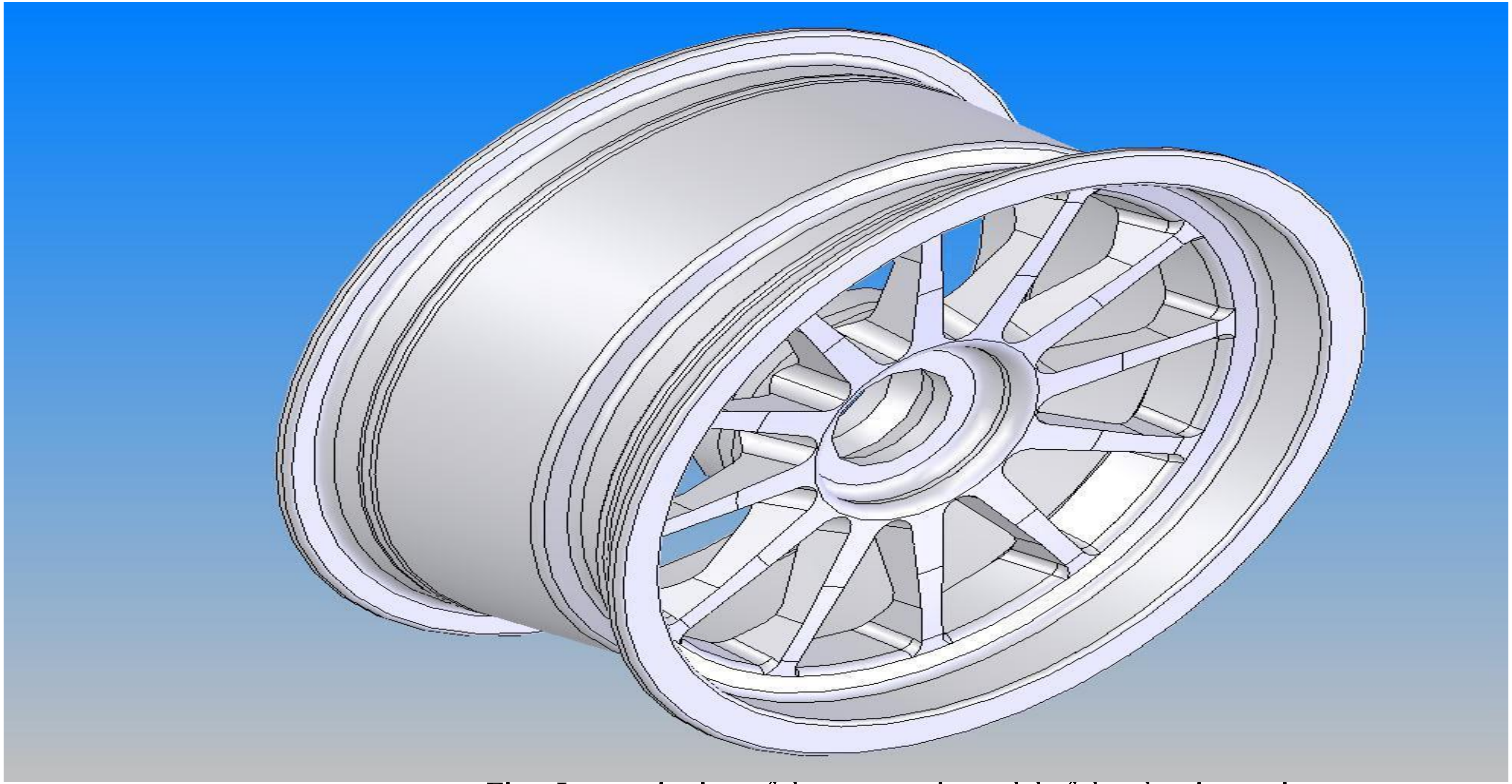


Fig 2 Isometric view of the geometric model of the aluminum rim.

The next step was to import the model into the software Ansys Mechanical. The latter fact allows to manipulate the geometric model to prepare it and turn it into mathematical model finite elements. The most delicate operation of the creation of the model is the phase of meshing, in fact at this stage is associated with the geometric model, the mathematical model. A grid (or mesh) generated in a correct way allows the subsequent analysis, otherwise not possible.

The import is often a critical operation, a weak point of these systems. For the present work and due to the fact that the model is imported solid type, there have been no major problems. The ideal would be to generate the surfaces directly in Ansys, but the rigidity of the internal geometric modeler makes even this road prohibitive for the generation of complex models as an automotive wheel rim.

Importing a model often requires a scaling factor that allows the program to set the correct proportions, in this case we have chosen a scale of 1: 1, as already generated the correct size.

Before the model is meshed we must define the material and the properties of the mesh. In Ansys this is particularly simple and is accomplished through data entry form very begining.

At this point, the grid can be generated, using the tetrahedral solid elements, size 7 mm, with the material and the property just defined. The automatic mesh generation has provided a result suitable for our purpose.

In fig. 3 is shown an isometric view of the mesh of aluminum rim.

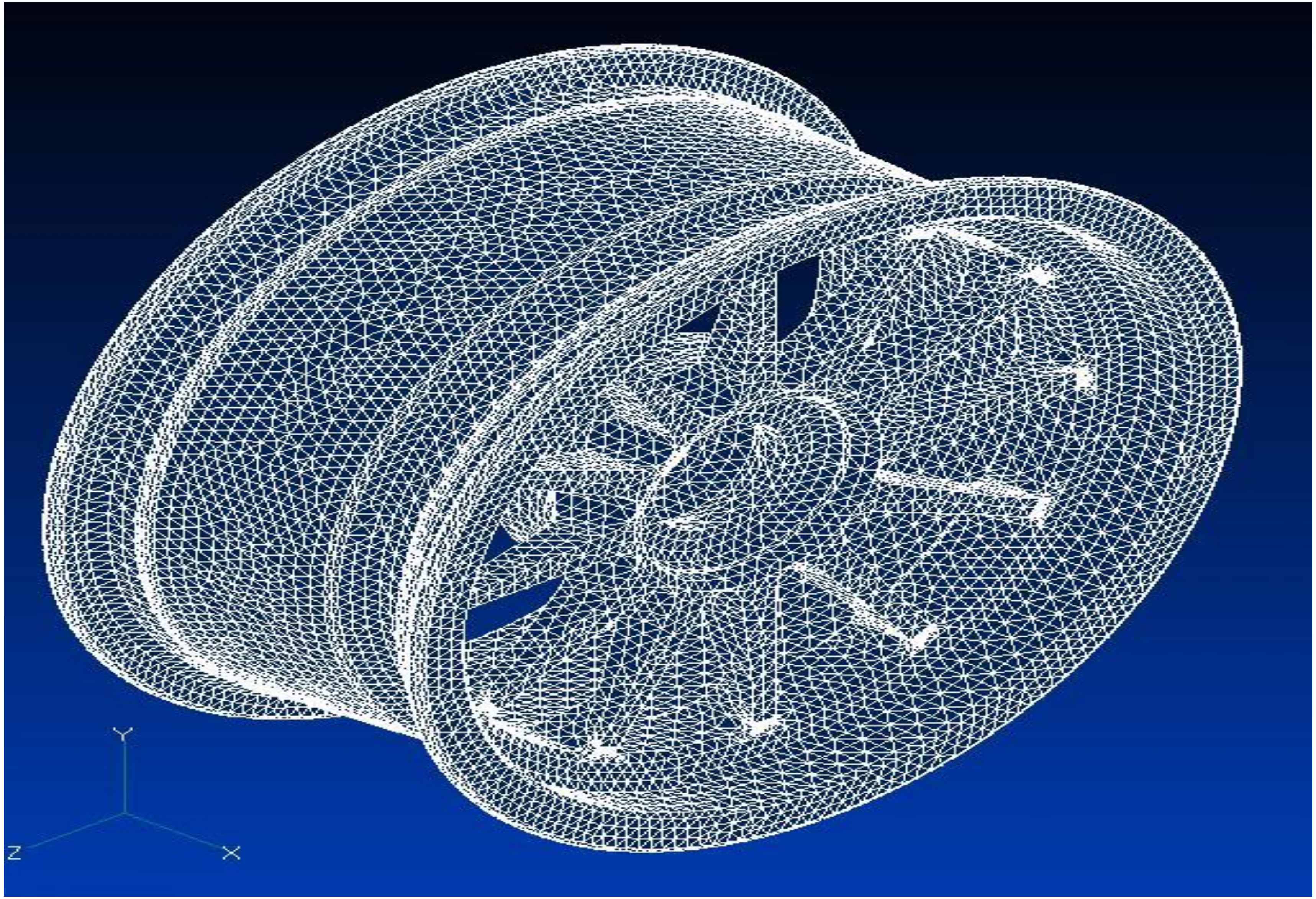
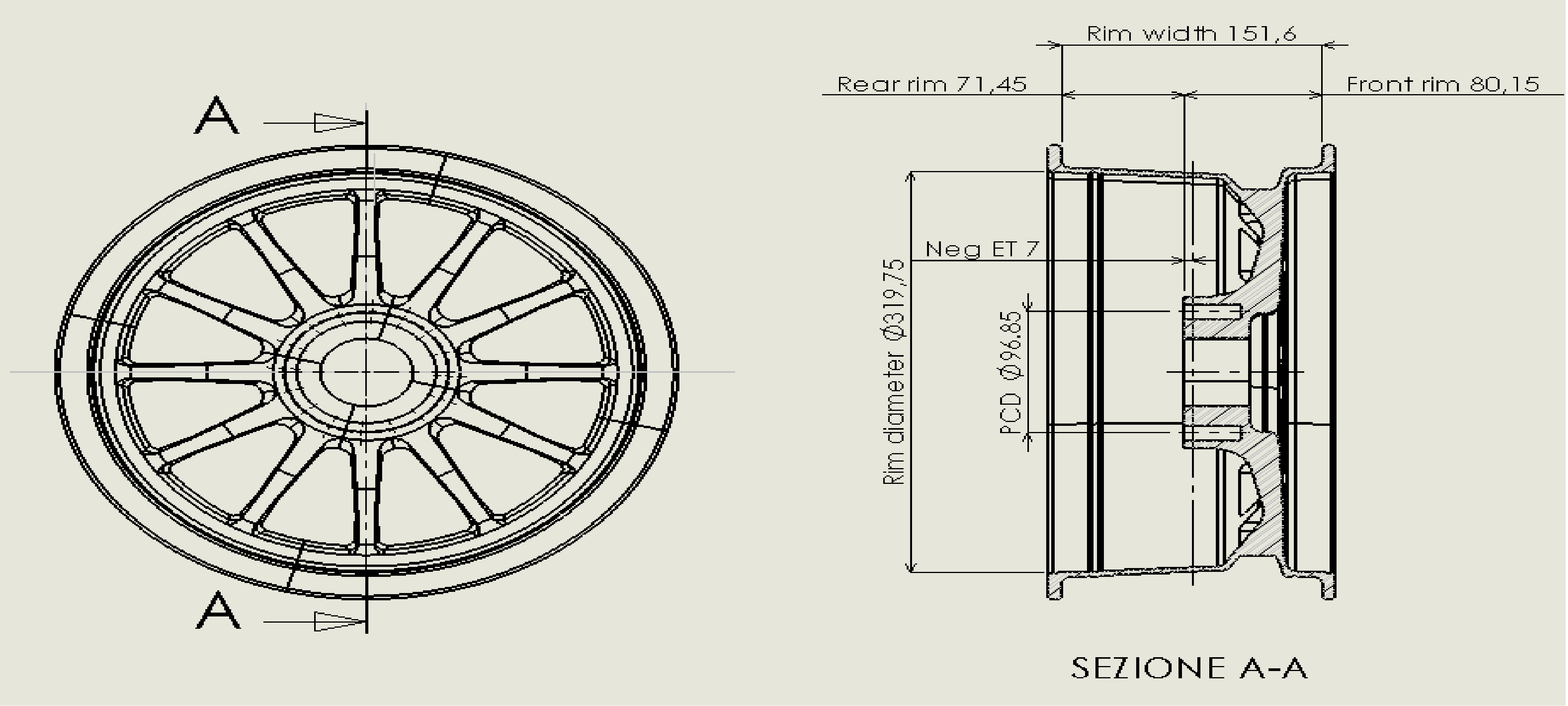


Fig 3 Isometric View of the model of the fem aluminum rim

Model of Composite rim

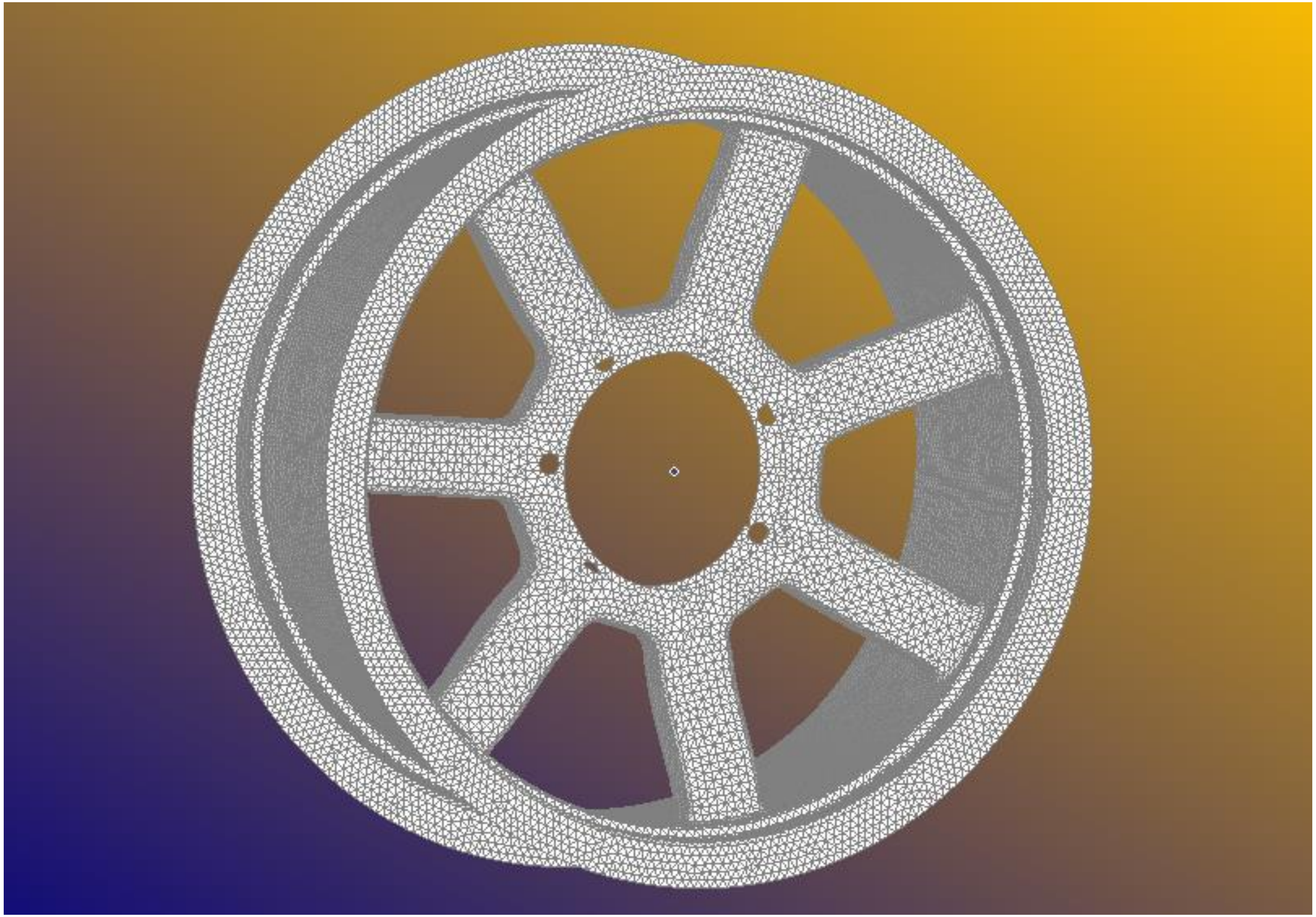
The geometric model was generated purely as a starting point for the construction of a grid computing. In this context it is minimized (where possible) the use of functions such as "loft" that would inevitably segment surfaces creating problems in the next phase of "meshing". During modeling, there is a tendency therefore towards a logic of "patching" decreasing the total number of surfaces. Only in this way it was possible to create a geometry simply exportable. The modeling technique is based on the axial-symmetry of the object under examination. It has been possible to represent a single breed of the rim and then use a function of circular mirror. Starting then from the geometric model has used a function of thickening. The purely graphical representation has seen the use of the thickening function in such a way as to provide a consistency to the rim that would enable an adequate "rendering" of the surfaces.



The characteristic size of the circle will be: 13 x 6J (-7) 16-97; that is, a 13-inch circle with a channel width of 6 inches, a J-shaped flange with negative offset in mm, 16 holes of attack whose centers lie on a hypothetical circle of diameter 97mm.

The FEM model has been realized starting from the geometric generated above. Having to evaluate the mechanical behavior of a component made of composite material has been produced with the grid elements of the type "rolled" to the discretization of the surfaces and with rigid elements in the later stage of application of the loads. You used the element laminated as this is the type of element responsible for the simulation of the behavior of composite materials. Having assumed the material as ortotropic 2D it was necessary to also define the orientation of the fibers in different laminae.

This was done by assuming a priori to know the trend of the stress state in the rim subject to various sets of load.



Morphing techniques for Optimization

Morphing an existing simulation mesh according
to updated geometric parameters in the underlying computer aided
design model is a crucial technique within fully automatic
design optimization. By avoiding costly automatic or
even manual meshing, it enables the automatic and parallel
generation and evaluation of new design variations, e.g.,
through finite element or computational fluid dynamics simulations.

Moreover, the same unified code can morph tetrahedral, hexahedral,
or arbitrary polyhedral meshes.

The adaptation of an existing simulation mesh is accomplished by using a mesh morphing or mesh warping technique: Given an initial CAD surface G and a volumetric mesh M of that geometry, a shape variation G to G' is generated by changing the geometric embedding of G while keeping its topology fixed. The mesh morphing technique then adapts the mesh M such that the updated version M' conforms to the updated boundary surface G'. Analogously to the geometric changes in the CAD model, only the geometric embedding of M (i.e., its node positions) is updated in this process, while the mesh topology (i.e., its connectivity) stays fixed.

Mesh morphing techniques aim at preserving the element
quality as much as possible, thereby allowing for as
large as possible geometric changes before inevitably requiring
some remeshing due to element inversion.

Courier Model

The Courier model is the most complex model in our comparison.
In contrast to previous examples, the hexahedral mesh
of this model is an unstructured one.

Unfortunately, due to a mismatch between CAD geometry and initial tetrahedral
mesh, we could not apply our surface morphing method for
the tetrahedral Courier model.