Hochschule Osnabrück
University of Applied Sciences

Fakultät
Ingenieurwissenschaften und Informatik

Master Thesis

About the theme

FEM and Modal Analysis of a Car Body

Author: Iosu Azcona - Alberto Blanque
iosuazcona@msn.com
blanqueaguilera@gmail.com

Professor: Prof. Dr. Bahlmann
Prof. Dr. Schmidt

Date: 01.07.2016

I Aknowledgments

First of all, we would like to thank to the Hochschule of Osnabrück for giving us the opportunity of coming here and let us having one of the most important experiences in our life.

Hereto, we would like to have an important words to our home university, "la Universidad Pública de Navarra", for all the years that we spent there and for all the professors that helped us to achieve this moment.

It gives us great pleasure in acknowledging the support and help of Professor Mr. Bahlmann, because from the first day he showed his predisposition to help us with everything. Moreover, we would like to thank for his continuously supervision and support during our Master thesis time.

This thesis would not have been possible unless the contribution of Mr. Kaeller. He was always ready to help us with everything besides he was not our professor and we found him inspirational.

At the end, we owe our deepest gratitude to our families and friends that along this period, they always have been showing their support and trust in what we were doing, and without them it would have not been possible.

II Abstract

Our Master Thesis consists on getting a proper validation from a convertible car body model in FEM to approach later to the same real car body. To do so, we were given the FEM model by Volkswagen with all its components perfectly modeled.

After understood how the software used to do this project worked, we started the comparison of our results with the results of our colleague Mr. Krampe, who was testing the real car. We had to change many different parameters and also the configuration of our car model to obtain some similar results.

Moreover, we did a brief introduction to the modal analysis and its relation with FEM. It was the theoretical part that allowed us to understand better the results obtained.

In dieser Masterarbeit werden FEM Simulationen am Fahrzeugrahmen eines Cabrios durchgeführt, um diese später an das echte Fahrzeug anzunähern. Um dies zu erreichen, bekamen wir ein FEM-Modell von Volkswagen bei dem alle Komponenten bereits optimal modelliert waren.

Nach einer Einarbeitung in die für diese Arbeit verwendete Software wurden die Ergebnisse mit denen des realen Cabrios, ermittelt von Herrn Krampe, verglichen. Es mussten viele Parameter und die Ausstattung des simulierten Modells angepasst werden um einige ähnliche Ergebnisse zu erzielen.

Zusätzlich wurde eine kurze Einführung in die Modalanalyse und ihre Beziehung zur FEM untersucht. Durch diese theoretischen Teil konnten die erhaltenen Ergebnisse besser verstanden werden.

III Contents

IV List of Figures

V List of Tables

1 Introduction

1.1 Introduction to FEM

1.1.1 A Brief History of Computer Aided Engineering

We would like to start our Master thesis with just a brief history in the software that we are going to use during our project, Finite Element Method (since now we refer as FEM).

The history of engineering is, in essence, the history of human culture. The organization and quality of life have been categorized as much by the tools and technologies of the age as any other factor. All of these different ages attest to the importance of technology, or engineering, to the story of who we are today, with the amazing development of FEM.

To fully appreciate the complexity and capabilities of the tools that we have nowadays available, one must try to understand this story and acknowledge its impact on their daily work.

If we ask who invented Finite Elements, there is no just a singular answer to that question. Only one individual did not create FEM, as we could see in the next lines. We are going to start our research chronologically, since the first steps that were made by different people to end in the latest developments of FEM.

We begin in the late 1800s, when Lord Rayleigh developed a method for predicting the first natural frequency of simple structures. It assumed a deformed shape for a structure and then quantified this shape by minimizing the distributed energy in the structure. After this, Walter Ritz expanded this into a new method, for predicting the stress and displacement behavior of structures. The choice of assumed shape was critical to the accuracy of the results and boundary conditions had to be satisfied as well. Unfortunately, as we could imagine, the method proved to be too difficult for complex shapes because the number of possible shapes increased exponentially as a complexity increased. However, because the number of possible shapes increased exponentially as complexity increased, this predictive method was also critical in the development of the future of FEM algorithms.

If we jump to the 1940s, numerical methods had been developed to predict behavior of more general structures. Most of them were frame and truss based and utilized energy methods from Alberto Castigliano (we all have used the Castigliano's theorem for determining the displacements of a linear-static system based on the partial derivatives of the energy) and William Rowan Hamilton. In addition, in 1943 Richard Courant proposed breaking a continuous system into triangular segments. In this time, it was the birth of digital computing too.

In the 1950s, analog computers were developed to process more complex structural problems. As the increasing number of more powerful computers was going on, at the same time analytical methods were advanced to include matrix based solutions of frame and truss structures. In this decade, we have to stop and analyze the importance of the Boeing Company and we have to underline the important role of one of the members of this company, M.J. Turner. He generalized and perfected the Direct Stiffness method, and forcefully got Boeing to commit resources to it while other aerospace companies were mired in the force method. In addition to Turner, we can find another important contributors as for example R.J. Melosh, who recognized the Rayleigh-Ritz link and systematized the variational derivation of stiffness elements. There were more pioneers that had an important contribution, but the main point is that all of them were in the aerospace industry at least during part of their careers. And that is not a coincidence; just because FEM is the confluence of three ingredients, one of which is digital computation, and only large industrial companies were able to afford mainframe computers during the 1950s. The benefit to the growing aerospace business was clear and most major manufacturers in this industry were developing in-house programs for structural analysis on computers. The basic concepts of FEM were born, although the process was still time consuming and limited.

In the next decade, Dr. Ray Clough coined for the first time the term "finite element". He published an article called "Stiffness and Deflection Analysis of Complex Structures". A finite element differed from previous 2D methods for computer simulation by replacing combinations of 1D elements with a single entity that could model 3D strain. The 1960s saw the true beginning of commercial FEM as digital computers replaced analog ones with the capability of thousands of operations per second. Academicians also drove this, because they were the responsible for the "technology transfer" from the aerospace industry to a wider range of engineering applications during the 1950s and 1960s.

In the early 1960s, a small analog computer manufacturer and consultant for the aerospace industry in southern California was awarded a contract from NASA to develop a general purpose FEM code. This company is nowadays so popular, "The MacNeal-Schwendler Corporation" (MSC), ensure the growth of commercial FEM by developing what is now known as NASTRAN, just the software that we are going to use for our Master thesis.

This original code had a limit of 68000 degrees of freedom, which was believed to be larger than anyone would ever need. Thanks to that contract, MSC continued the development of its own version, that was called "MSC/NASTRAN", while the original "NASTRAN" was available to the public and formed the basis of dozens of the FEM packages available today, as ANSYS, MARCMENTAT and SAP. Linear static and limited dynamic analyses were available to engineers who could justify the expense of buying the time from a computer center running.

In 1967 was published the first book on the FEM published by the Dr. O. C. Zienkiewicz. To this day, it remains a standard reference text for the basis of the FEM.

By the 1970s, minicomputers became more readily available and were more powerful than earlier mainframes. The power and availability FEM software matched the growth of the computer industry. While shared computing centers were still the norm, frequent users were migrating to in-house software, either internally developed or leased from the commercial FEM vendors. Although most analysis was linear, nonlinear solvers were developed and made available.

With the commercialization of FEM taking off, many analysts were no longer aware of the conceptualization errors that can arise when performing simulations with the FEM. For example, combining 1D and 3D theories in the same mathematical model. This practice led to unknown uncertainties, and became known as finite element modeling. Moreover, the speed of computers was improved to 10000 and even 100000 operations per second. Computer-aided design, or commonly called "CAD", was introduced later in this decade, helping the development of this movement.

If the 1960s marked the birth of commercial FEM, the 1980s represented its coming of age. Computing centers were becoming a thing of the past as workstations and PCs on the engineer's desktop began to dominate the market. Another key advancement was the use of FEM and CAD on the same workstation with developing geometry standards such as files IGES and DXF. Standards permitted limited geometry transfer between the systems. These workstations were capable of over 1 million operations per second and took advantage of optimized algorithms for working with the math behind FEM. Also, it is remarkable that in 1981, Drs. Babuska, Szabo and Katz performed the mathematical proof of p-convergence. This method is a numerical method for solving partial differential equations, that later would be very used in the FEM world.

Moreover, in the 1980s, CAD progressed from a 2D drafting tool to a 3D surfacing, and then to a 3D solid modeling system. The developments in graphics processing left their mark on FEM as graphical "pre" and "post" processors became available and engineers could examine colored stress contours instead of poring over tabular output on green-bar fanfold. Thanks to this advancement, design engineers began to seriously consider incorporating FEM into the general product design process. The link to CAD was the catalyst for this natural next step.

As the 1990s draw to a close, the PC platform has become a major force in high-end analysis. The technology has become so accessible that it is actually being hidden inside CAD packages. It is not uncommon for a product engineering company to have no specialists performing non-linear, vibration analysis, computational fluid dynamics (CFD), and multi-physics simulation. In 1991, Drs. Barna Szabo and Ivo Babuska, published "Finite Element Analysis". This text focused on the implementation and application of the FEM with a focus on solution verification. This text would set a new standard for rigorous implementation of the p-version of the FEM.

If we continue checking relevant events related with FEM, we have that again the aerospace industry has an important role, just because in 2008, NASA released a standard for the development of model and simulations. In 2011, again Drs. Barna Szabo and Ivo Babuska published a follow-up to their 1991 text "Finite Element Analysis" with "Introduction to Finite Element". This text was published as a teaching companion, as well as a learning aid, for educators, students and professionals.

1.1.2 Analytical Problem Solving Process

The merits of planning have been stressed in many sources for many types of problem solving techniques. Planning is also important for solving problems in FEM. The thought process for solving a problem using FEM is similar to that for solving a problem using lab tests, free body diagrams, classical equations, or spread sheets. Understanding the "big picture" as well as specific data objectives will bring the selection of tools, the problem solving approach, and the interpretation of the results into clear perspective. Moreover, setting proper expectations and being technically prepared for the meaning of any and all data are critical to the difference between "pretty pictures" and solid engineering with advanced tools. Many would be analysts plunge right into FEM without first becoming comfortable building free body diagrams, navigating simple beam calculations, and investigating the meaning of von Mises stress and why it is a poor indicator of failure in many engineering problems.

1.1.2.1 Process Most engineering problems are solved using the following four steps. They will not always be explicitly documented but they will be considered at some point in the process. Short-changing any of them is the quickest path to failure. The steps are:

1. Establish a clearly defined goal: the goal of the analysis should generally include two important decisions. First of all, it is important to predict an exact solution to a problem or maybe sometimes it is sufficient to show trends. It will depend on the steps taken to solve the problem and the precision of the inputs ride on the answer to this question. The second important decision involves developing an analysis

for overall displacements versus localized stress data. If stresses are not deemed important, many details can be left off the model and your options for idealizations to improve efficiency are expanded.

2. Compile and qualify the inputs.
3. Solve the problem with the most appropriate means.
4. Verify and document the results.

1.1.2.2 Predictive Engineering versus Failure Verification. The most efficient and cost effective point to begin the structural analysis of a component or a system is at the conception stage of design. Letting the results of the analysis drive the choice of materials, features, wall thickness, and so forth is called predictive engineering. Using this methodology, data are generated when they can have the greatest impact on the cost and quality of the design at the lowest cost of change and schedule.

Failure verification represents the most rigid usage of structural analysis. Correlating FEM data to an existing condition requires a careful examination of the boundary conditions, material properties, geometry, operating environment, and the actual failed part. Assumptions and approximations must be minimized. Prior to undertaking an analysis, evaluate whether failure is consistent across several parts or is isolated to one sample. An isolated instance might suggest that a defect in geometry or material, and/or an unexpected loading scenario was responsible. An analysis that does not take these factors into account may not provide useful data.

If analysis is to be used early in the design process, start on simplified geometry. Complex solutions can be more easily approximated as simple ones at this stage. Refine your analyses as the part definition and behavior become more refined.

So in our case, we are located in the failure verification, because we have our model totally defined, and our intention is to get a validation model of some parameters that have been tested and we would like to approach them. For that reason, we have to be careful with the boundary conditions of our car, and try to guess how we could modify to be nearer to the real parameters.

1.1.2.3 Input and Output Data; What Level of Uncertainty does it Introduce? After setting up the boundary conditions of our model, the next step in solving an engineering problem is compiling all inputs required by the chosen in solution technique or tool. We have to realize the importance of this input data, because at the end, we are representing a real model that it will suffer some external conditions, so this input data should be the more accuracy to the real problem as we could.

A hand calculation might only require a cross-sectional areas and moments of inertia whereas a CFD analysis might require temperature dependent viscosity, density, and specific heat properties of several interacting fluids. In addition to filling in the blanks on a data form or in a closed form equation, care must be taken to qualify the data being used.

In the same way as with the input data, we have to know what to deal with the output data. The need for a specific data is often intuitive and obvious. If strength is a consideration, solve for stress and deflection. If cooling is an issue, solve for temperatures. However, there are cases that for example, the stress is localized in a specific area, or the cooling solution steady state or are transients to be considered. In conclusion, it exists some specific questions that will drive the choice of tools and dictate the level of detail or skill required to use these tools. At the end, we have to know what we exactly have to ask for our program to get our interested solution for our problems.

Related with the level of uncertainty that we can get from the input data, we know that a material or geometric parameter is rarely consistent for all produced components, and loading measured in a test apparatus may differ dramatically from actual field usage. An engineer's ability to evaluate results with such an incomplete data set separates an analyst from a skilled button pusher.

Engineers are charged with making decisions based on incomplete data sets every day. Assumptions regarding material uniformity, assembly variability, user inconsistency, and general unpredictability need to be weighted, qualified, and documented. In the end, however, a decision must be made. This is the role of the design analyst, who must take this process a step further. In addition to the uncertainty mentioned previously, the engineers must use an idealized computer model with clearly unrealistic boundary conditions to represent a physical phenomenon, which could rarely be repeated in testing. The key to success in light of these seemingly uncontrollable circumstances is a scientific qualification of the uncertainties involved.

1.1.3 The Role of the Finite Element Method in Engineering Analysis.

In the last point, we have been talking about the influence of the solving a problem analytically, and we personally believe that is also useful to where the Finite Element Method fits in with other methods of engineering analysis. Engineering can be broadly divided into two categories: classical methods and numerical methods. We can look more clearly in the Figure 1:

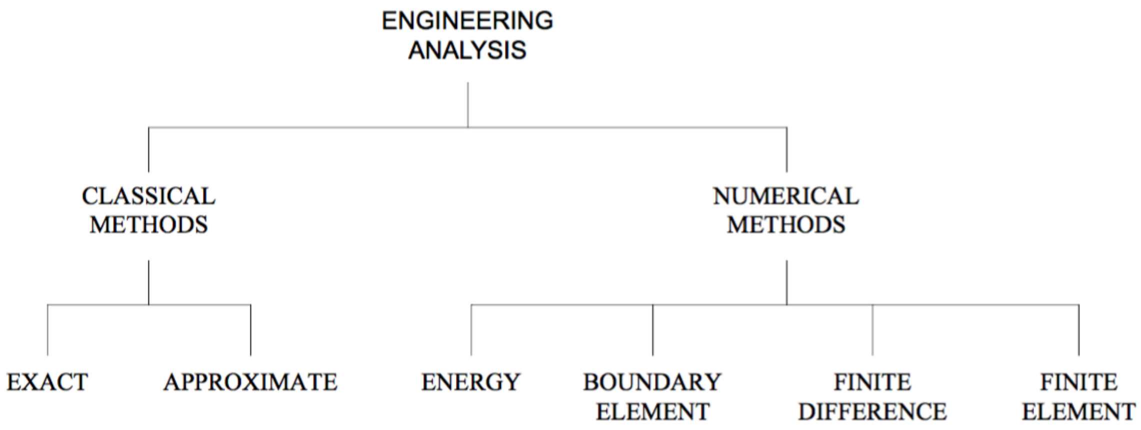

Figure 1: Engineering analysis

Classical methods attempt to solve field problems directly by forming governing differential equations based on fundamental principles of physics. Exact solutions are possible only for the simplest cases of geometry, loading, and boundary conditions. A somewhat wider variety of classical problems can be solved using approximate solutions to the governing differential equations. These solutions take the form of series expansions that are truncated after a reasonable degree of convergence.

In the other side, we have numerical methods that address a broad range of problems. The energy method seeks to minimize an expression for the potential energy of a structure over its entire domain. This approach works extremely well for certain problems, but it is not broadly applicable. The boundary element method approximates functions satisfying the governing differential equations, but not the boundary conditions. Problem size is reduced because elements represent only the boundary of the domain. However, the application of this method relies on knowing the fundamental solution to the governing equations, which can be difficult to obtain. So, the finite difference method replaces governing differential equations and boundary conditions with corresponding algebraic equations. This permits the representation of somewhat irregular problems, but complex geometry, boundary conditions, or loads become difficult to handle.

At the end we arrive to the Finite Element Method. FEM offers virtually unlimited problem generality by permitting the use of elements of various regular shapes. These elements can be combined to approximate any irregular boundary. In similar fashion, loads and constraints of any type can be applied. Problem generality comes at the expense of insight, it means, a finite element solution is essentially a stack of numbers that applies only to the particular problem posed by the finite element model. Changing any significant aspect of the model generally requires a complete reanalysis of the problem. Analysts consider this a small price to pay, however, since the Finite Element Method is often the only possible method of analysis. The Finite Element Method is applicable to all cases of field problems, including structural analysis, heat transfer, fluid flow, and electromagnetic. We can imagine how important is to engineers to know this method to solve a lot of different kind of problems.

1.1.4 The Finite Element Process.

We could say that generally, Finite Element Analysis seeks to approximate the behavior of an arbitrarily shaped structure under general loading and constraint conditions with an assembly of discrete finite elements. Finite elements have regular, or we it is better to say "nearly regular", geometric shapes and known solutions. The behavior of the structure is obtained by analyzing the collective behavior of the elements.

This process is illustrated in the Figure 2:

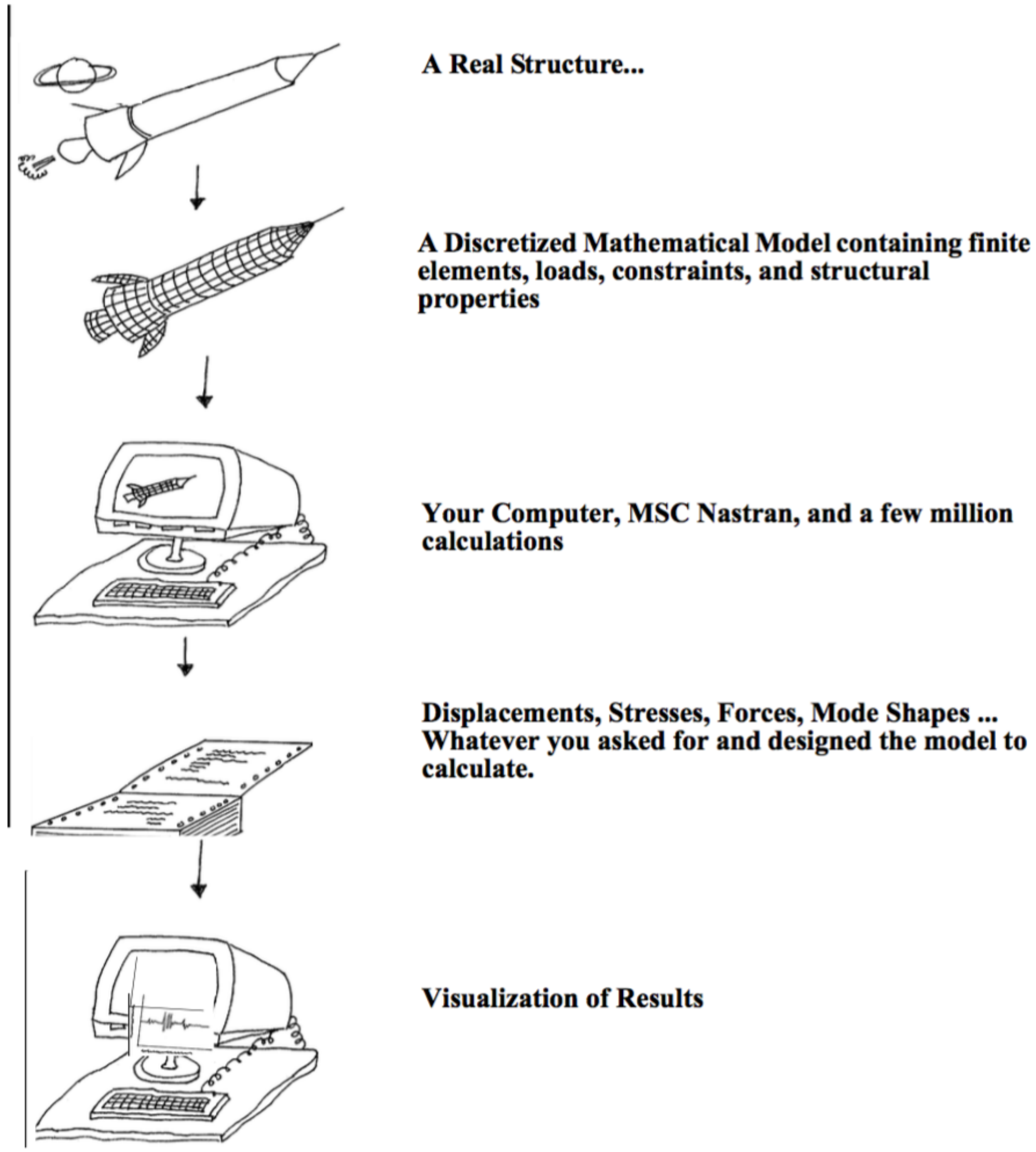

Figure 2: Finite Element Process

1.1.5 Some Common Misconceptions about FEM.

Those who are not intimate with FEM will certainly harbor some misconceptions about the ease of use or the degree of accuracy entailed. It is interesting to note that nonusers or casual users with similar levels of exposure can have diametrically opposed viewpoints on these issues. Typically, most preconceptions are based on some defining experience with the technology. Some common incidents which have a profound impact on one's opinion might be unsupported use of a difficult pre-processor, inconsistent or event disastrous correlation to test or field data, a bad experience with a consultant who was not much more qualified than the nonuser, or a successful project that seemed effortless and nearly pushbutton.

Continuing with these misconceptions, we would like to mention some of the most common ones:

- Meshing is everything: it is often believed that if a part can be meshed, the battle is over. The exact opposite is often closer to the truth. With the efficiency and quality of today's automeshers, developing a solid Auto-mesh of clean CAD geometry is probably the easiest step in the process. Ensuring that the final mesh has been converged on the desired behaviors is a more difficult proposition. It is also remarkable that meshing is only one input to the problem. New users must resist in some cases the temptation to utilize mindless auto-meshing.

- FEM replaces testing: it is unwise to assume that all answers kicked out of a FEM program are accurate, given all the assumptions and uncertainties which must accompany any study. Only through correlation of test models and actual prototypes can the methods and assumptions used in FEM be qualified. After qualification, it may be decided that the analysis results for similar studies in the future are reliable and some developmental testing may be eliminated. Another point o view could be that a physical test can only examine a small number of quantities and typically ends when a catastrophic failure occurs. In most cases, the observer has no way to evaluate the nearness to failure of other portions of the system. If we have a well-developed analytical model, it will show the relative quality of all parts of the structure. So, a test can provide valuable data about the validity of boundary conditions. However, when applied correctly, it can be said that a solid predictive engineering program can reduce testing in the design as the confidence in simulation results, as we would like to do in this project.

- Finite Element Analysis is Easy: the surest route to failure in FEM is to underestimate the complexity of the technology. Without a healthy respect for the variability of the physical systems being modeled and the sensitivity of output to multiple interrelated input quantities, it could be that an analyst may jump to conclusions too quickly and make design decisions based on a CAD part extremely easy. Consequently, obtaining answers is literally a few button pushes away from the completed geometry. Knowing what those answers mean and how to adjust the model to improve, or even evaluate the accuracy, requires a deeper understanding of the working of the technology and the meaning of the various parameters involved.

- Finite Element Analysis is Hard: it is said that when you work with FEM, 20% is creativity, and the 80% rest is hard work and patience. Analysts become successful through the exercise of patience and thoroughness, not necessarily exceptional intelligence or ability. It cannot be stressed enough that the technology, with reliable software, will provide correct answers to questions defined by mesh, properties, and boundary conditions. It has been said that there are no wrong answers in FEM analysis, only wrong questions, so if users can learn how to pose the questions correctly, we will take good answers.

- Learning the Interface Equals Learning FEM: it is thought that the best candidates to use FEM software are those candidates who are most proficient at CAD, because as they are speedsters on the keyboard, they are expected to learns the interface quickly and make the most out of the FEM investment. But it is not always like that. Most experienced analyst would agree that the first model should be performed slowly and repeated with minor variations of inputs to better understand the sensitivities and anomalies of the technology. The sooner all interested parties up and down the management chain understand that getting some kind of results is easy, while getting the right result requires the analytical problem solving of an experienced engineer, the sooner the qualifications for choosing the best users will be adjusted. Learning the interface or the pre-processor is relatively simple compared to the subtle nuances of assembly interaction or the ability to ascertain the meaning of local stress concentrations. For that reason, we would like to highlight that at the beginning of our project, we spend a long time with the tutorials of the different software, trying to learn the new interfaces, as trying to know how we could change the parameters to get good results, starting always with simple examples and then continuing with the next steps of analyze.

To sum up with this point, we would like to say that along our project, we tried to improve our skills in this field of the engineering, and we also tried to improve our analyst roles as future engineers.

1.1.6 Working with Existing Geometry.

There are sometimes that we need to analyze some geometries that were not developed to be analyzed using FEM in mind. In our case, we have to analyze a geometry given by Volkswagen, so we have to try to guess how we could try to manage it to get good results using FEM. Otherwise, there are some techniques for identifying and correcting problems and provides tips for preparing geometry, planned for FEM or not, to create a good mesh.

These guidelines apply to geometry setup in general. They should be considered regardless of whether or nor the part is constructed for eventual analysis.

- Symmetry: when symmetry is allowed, it should be clearly used. Symmetric FEM modeling will always result in faster runs and may result in more accurate or accurate looking results.
- Boundary condition adjustments: the term ?patch? is used to describe a logical break in a surface or curve for more precise placement of boundary conditions. Patches are also used to provide more control over the mesh, possibly near a high stress gradient where transition from very small elements is required. In general, however, applying patches in the native CAD systems is the easiest. But in our case, as we have the geometry given, we have to work with ANSA (pre-processor) and try to reach the best conditions with the objective of applying the best boundary conditions.
- Feature suppression versus sub modeling: feature suppression is always one of the most temptresses ways to work with FEM. Choosing the features to remove or suppress requires engineering judgment. It is not good practice to assume that all small features or fillets mat be suppressed. Feature suppression will be differentiated from sub modeling in that sub modeling involves the isolation of certain portions of the geometry that can be assumed decoupled from the rest of the part. The assumption of decoupling warrants further discussion. Depending on the model, it may not be easy to determine if decoupling is possible, and if so, where to break the model.
- Mid-plane extraction: there are components that as we easily can see, the overall size of that part is so large in comparison to its wall thickness that a mid-plane model will behave nearly the same as a mesh on the outer surface. As we learnt in one tutorial with ANSA, some pre-processors allow for mid-plane extraction to facilitate shell model construction. However, another commonly used technique is to simply use the inner or outer surfaces of a thin-walled solid as the shell model surface.

- Knowing when to bail: knowing when to stop can be as valuable as knowing when to start when it comes to geometry preparation and simplification. While more applicable to design analysts, even specialists have time constraints on their projects. It is not advisable to spend valuable hours or days cleaning up the geometry when starting form scratch or running with a slightly larger mesh will tale less time overall and will allow you to reach the end faster.

1.2 Pre-processor, Processor and Post-processor.

In this project we have used three software to do the analysis of our car. There are another kind of software that include this three steps in just only one software, but in our case, as we have been studying the body of a car, we had more possibilities using ANSA in the first step and then processing with NASTRAN, to be able to read the results at the end with µETA. It is necessary to say that the same company, called µETA, develops both ANSA and µETA.

1.2.1 ANSA.

ANSA is an advanced multidisciplinary CAE pre-processing tool that provides all the necessary functionality for full-model build up, from CAD data to ready-to-run solver input file, in a single integrated environment. ANSA is the user's preference due to its wide range of features and tools that meet their needs. The list of productive and versatile features is long and the alternative tasks and processes to be completed using them are countless.

ANSA is widely used in the automotive industry. ANSA allows us to read different kind of CAD data, manipulated and healed by the powerful built-in geometry engine. A wide range of geometry healing functions, including those for the generation of neutral fibers, deliver geometry descriptions ready to be meshed.

ANSA also maintains the association between CAD geometry and the FEM mesh, so this means that FEM meshes are better representations of their geometric "parents". Also, we use ANSA because it is easy to maintain and update any changes in the geometry by simply reworking the updated area instead of recreating the FEM from scratch, as in another programs. For this reason, we are convinced that ANSA is the best option to make our project, as we have to change many parameters to get good possible results.

Moreover, we have to underline the most important features about ANSA are the possibilities that offers to mesh and the assembly. The integrated Batch Meshing tool leads to controllable and effortless optimal results, for both shell and volume meshing. Following the versatile mesh area idealization, geometry can be meshed according to modeling requirements by cutting edge surface and volume meshing wrapping algorithms.

At the same time, ANSA is powered with fully comprehensive parts and welding management tools, accommodates parts assembly, with alternative node-dependent or independent connections types, appropriate to various disciplines. Interfaces to numerous connections data file formats allow the completion of a single stage assembly. So this module is really interesting for our project, because the assembly of a car is quite complex. For that reason, ANSA is the perfect tool for us.

1.2.2 MSC NASTRAN.

MSC NASTRAN is a general-purpose finite element analysis computer program. MSC NASTRAN addresses a wide range of engineering problem-solving requirements as compared to specialty programs, which concentrate on particular types of analysis. This program is written in FORTRAN and contains over one million lines of code. This is important in order to NASTRAN is composed of a large number of building blocks called modules. And a module is a collection of FORTRAN subroutines designed to perform a specific task-processing model geometry, assembling matrices, applying constraints, solving matrix problems, calculating output quantities, conversing with the database, printing the solution, and so on.

The main structure of MSC NASTRAN to create a finite element model is composed by:

- Coordinate Systems.
- Model Geometry.
- Finite Elements.
- Loads.
- Boundary Conditions.
- Material Properties.

As we are using ANSA as pre-processor, these steps are configured before, but it is good to know and try to understand the steps following by NASTRAN to make future changes in our model.

The input file that we make with ANSA is a text with a filename and a ".DAT" extension. This input file contains a complete description of the finite element model, including:

- The type of analysis to be performed.

- The model's geometry.
- A collection of finite elements.
- Loads.
- Constraints (Boundary conditions).
- Requests for the type of output quantities to be calculated.

The input file can be created with a text editor or finite element pre-processor. In our case, as we said before, ANSA gives us directly this text file. To execute MSC NASTRAN, the user types a system command followed by the name of the input file.

The structure of the input file is shown in the Figure 3:

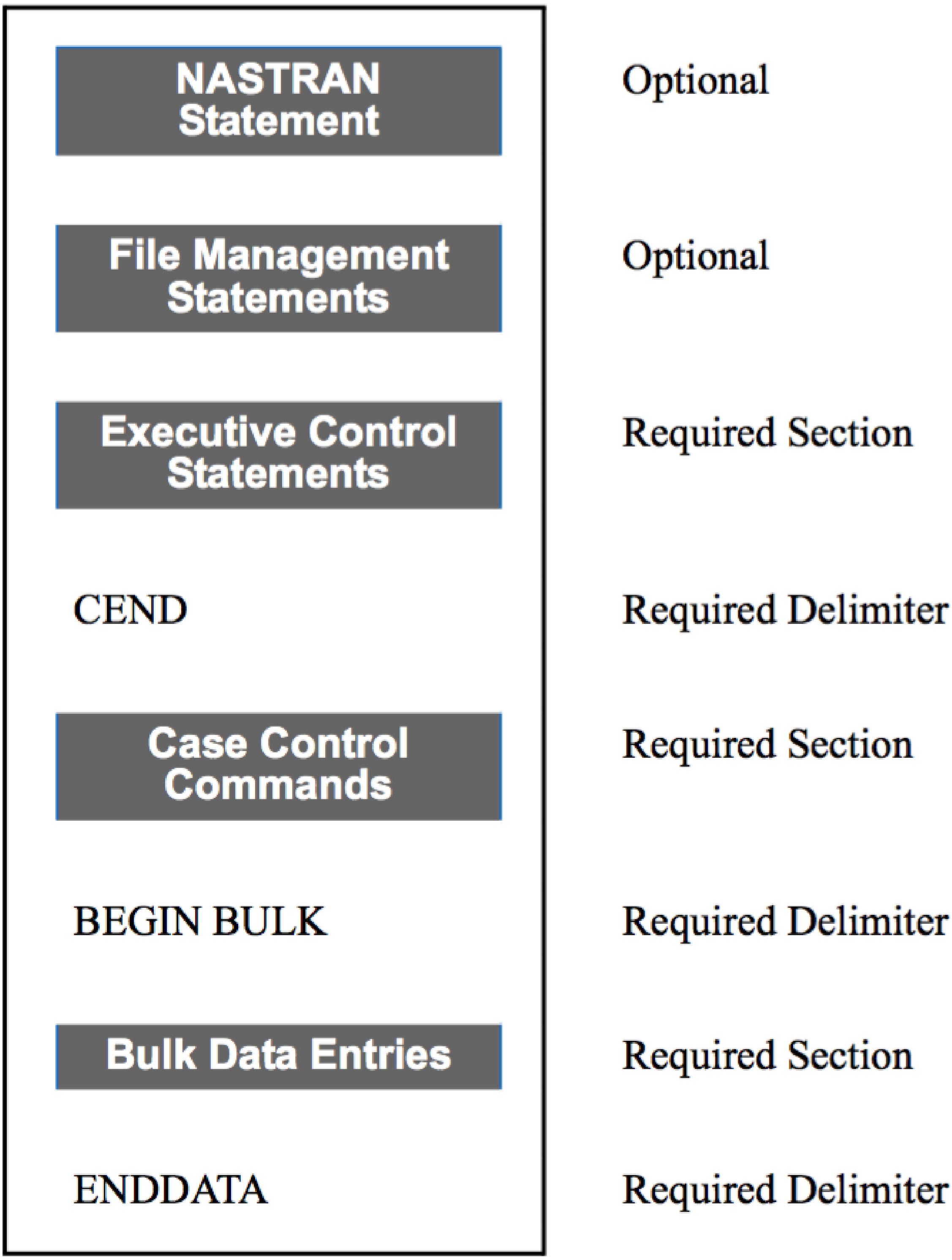

Figure 3: Structure Input File

Executive Control Section, Case Control Section and Bulk Data Section are always required. However, NASTRAN Statement and File Management Section are optional.

In the ECS, the primary function is to specify the type of analysis solution to be performed. The CCS is used to specify and control the type of analysis output required. And BDS contains everything required to describe the finite element model-geometry, coordinate systems, finite elements, element properties, loads, boundary conditions, and material properties. In most analyses this section constitutes the vast majority of the total MSC NASTRAN input file.

To sum up, we would like to clarify the using of Pre- and Post processors. Developing a finite element by hand is a time consuming, tedious, and error-prone activity. Making sense of a large stack of finite element computer output is also a considerable challenge. A finite element pre- and post processor is a graphics-based software package primarily designed to aid in the development of a finite element model and to aid in the display and interpretation of analysis results. In addition, pre-processing software helps the analyst modify the original model if results show that changes and subsequent reanalysis are required.

As we can imagine, in our case, ANSA and µETA are integrated with the analysis software. The role of pre- and post processors in the finite element analysis is shown in the Figure 4:

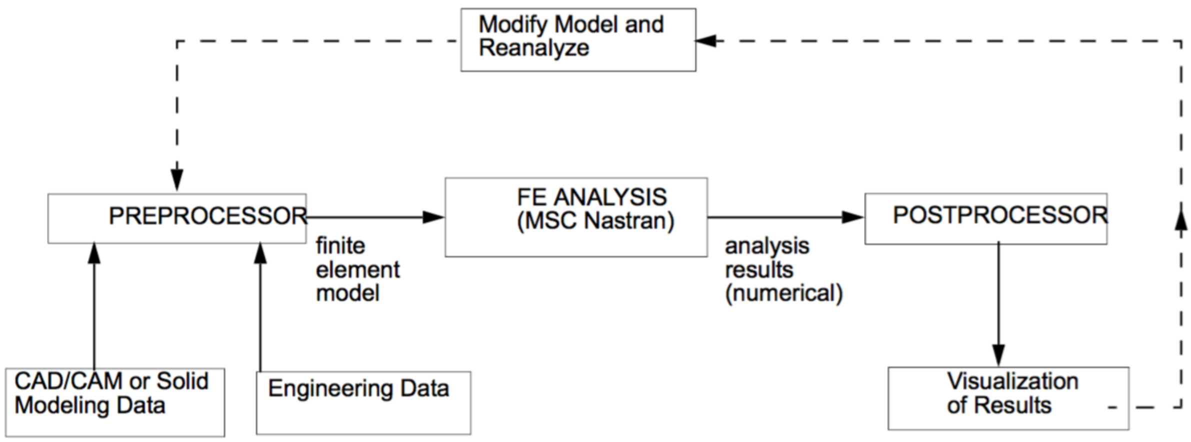

Figure 4: Role of Post Processor

1.2.3 µETA.

µETA is a thriving multi-purpose post-processor meeting diverging needs form various CAE disciplines. It owes its success to its impressive performance, innovative features and capabilities of interaction between animations, plots, videos, reports and other objects. So it is perfect to get the results of our car body and analyze them.

We can get a great benefit from this software thanks to the top quality graphics, the high performance and the efficient handling of large, multiple models and data, form a productive working environment.

1.2.4 Why ANSA and µETA.

The combination of both software is extensively deployed in many industries, including the automotive, motorsports, aerospace, defense, energy, maritime, offshore...The combined and fully integrated product suite is the preferred choice of industry worldwide as it seeks exceptional performance to impact its product development processes.

The main features that benchmarks have highlighted form these programs are:

- Best geometry clean up
- Faster and correct mid-surface generation.
- CAD data reconstruction capabilities.
- Fast and high quality shell and volume meshing, of complex geometries.
- Batch meshing with controllable and predictable results.
- Morphing capabilities.
- Robust FEA and CFD modeling for a large number of solvers.
- Highest level of support for large number of solvers, including NASTRAN, and Abaqus.
- Overall performance of large models post-processing.
- Level of integration of software features.
- Excellent learning curve and ease of use.

1.3 Failures Modes and Dynamic Analysis.

The first step in results interpretation is to review the goals set forth at the beginning of the study. These should tell us where to look and what to look for. As it is normal, we should look for some evidences of failure or assurance that failure is unlikely. So, now we are going to make a simple review about the typical failure modes, because it is necessary to know, but on our project, we will be more focus in the dynamic analysis that we will explain later.

1.3.1 Typical Failure Modes.

As our work is going to be more thoughtful to the dynamic analysis, we would like only to mention the more common types of mechanical failure, as we consider that maybe in our car body, we could find some problems related with this theme:

- Fracture.
- Yielding.
- Insufficient stiffness.
- Buckling.
- Fatigue.
- Creep.

In most engineering problems, two or more of these failure modes may be possible given the operating conditions of the system. It is important to review the data for all occurrences of any potential failure.

1.4 Dynamic Analysis.

Dynamic analyses as applies to FEM involve as we know, loads and corresponding response states that vary with time. Strictly speaking, such analyses should be referred to as vibration and time response analyses, because large displacement, completely rigid body motion is not the realm of FEM.

In this moment, we should have clear that vibration and response analyses can be divided into the following three related categories:

- Modal or natural analysis: this category involves the free vibration of the dynamic system. This analysis characterizes the system in the absence of external loading and serves to define its dynamic properties.

- Frequency response analysis and Transient response analysis: these two categories are known as ”forced response”, because they involve systems under externally applied loading functions, which can be either frequency or time dependent.

1.4.1 Fundamentals of Dynamic Analysis.

We believe that we should start this theme having a look in the main differences between static and dynamic analysis. Two basic aspects of dynamic analysis differ from static analysis. First, dynamic loads are applied as a function of time or frequency. Second, this time or frequency-varying load application induces time or frequency-varying response, as displacements, velocities, accelerations, forces and stresses. These time or frequency-varying characteristics make dynamic analysis more complicated and more realistic than static analysis.

1.4.1.1 Dynamic Analysis Process. Before conducting a dynamic analysis, it is important to define the goal of the analysis prior to the formulation of the finite element model.

It is possible to observe in the Figure 5, a way of making a dynamic analysis:

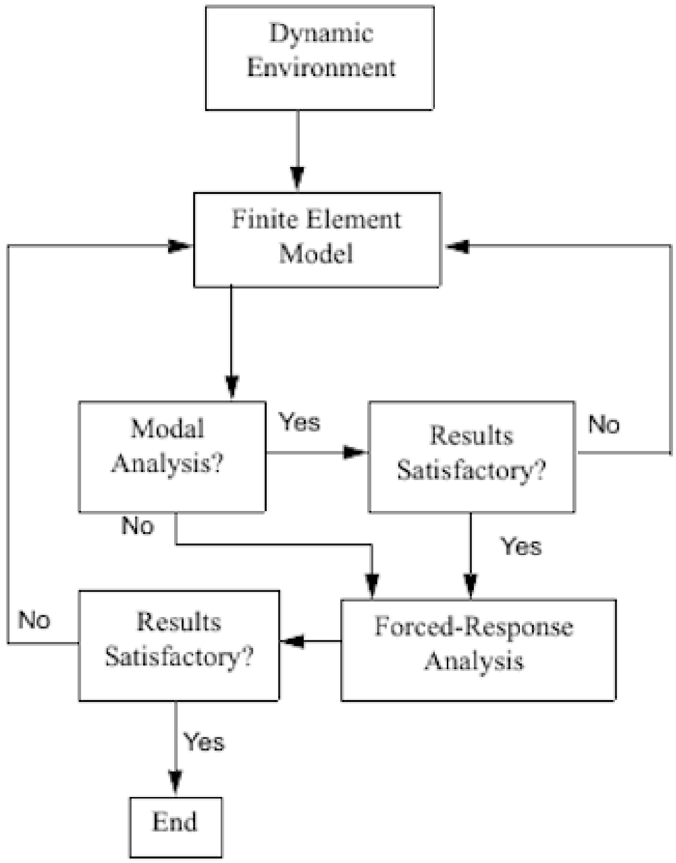

Figure 5: Dynamic Analysis Process

In our research, we must evaluate the finite element model in terms of the type of dynamic loading that will be applied to our car body. In our scheme, the dynamic load is called as the dynamic environment. The dynamic environment governs the solution approach, it means, normal modes, transient response, frequency response, etc...that we will explain later.

This environment also indicates the dominant behavior that must be included in the analysis, taking care of the contact, large displacements. At the end, proper assessment of the dynamic environment leads to the creation of a more refined finite element model and more meaningful results.

The first steps in performing a dynamic analysis are summarized as follows:

1. Define the dynamic environment (loading).
2. Formulate the proper finite element model.

3. Select and apply the appropriate analysis approach or approaches to determine the behavior of the structure.

4. Evaluate the results.

1.4.1.2 Dynamic Analysis types. With our software MSC NASTRAN, we are able to make different kind of dynamic analysis. These types are:

- Real eigenvalue analysis.
- Linear frequency response analysis: steady-state response of linear structures to loads that vary as a function of frequency.
- Linear transient response analysis: response of linear structures to loads that vary as a function of time.

We are going to describe carefully each of these types and try to understand which one corresponds to our project.

1.4.1.2.1 Real Eigenvalue Analysis. The usual first step in performing a dynamic analysis is determining the natural frequencies and mode shapes of the structure with damping neglected. These results characterize the basic dynamic behavior of the structure and are an indication of how the structure will respond to dynamic loading.

The natural frequencies of a structure are the frequencies at which the structure naturally tends to vibrate if it is subjected to a disturbance.

The deformed shape of the structure at a specific natural frequency of vibration is termed its normal mode of vibration. Each mode shape is associated with a specific natural frequency.

Natural frequencies and mode shapes are functions of the structural properties and boundary conditions. For example, a cantilever beam has a set of natural frequencies and associated mode shapes. If the structural properties of the beam change, the natural frequency change, but the mode shapes may not necessarily change. So, if we change the elastic modulus of this beam, the natural frequencies change but the mode shapes remain the same. Otherwise, if we change the boundary conditions, then the natural frequencies and mode shapes would both change.

There are many reasons to compute the natural frequencies and mode shapes of a structure. One reason is to assess the dynamic interaction between a component and its supporting structure. If the component that we are studying is going to be installed in a determined place, first of all, we have to be sure which is its natural frequency of the component, just in case, it could be that it would be similar to the natural frequency of the place that it is going to be located. If these frequencies are closed, the operation of our component may lead to structural damage or failure.

Decisions regarding subsequent dynamic analyses like transient response or frequency response can be based on the results of a natural frequency analysis. Moreover, normal modes analysis can often provide an indication by paying attention to how the frequencies shift and if they now align with frequencies to be avoided.

In summary, there are many reasons to compute the natural frequencies and mode shapes of a structure. All of these reasons are based on the fact that real eigenvalue analysis is the basis for many types of dynamic response analyses. Therefore, an overall understanding of normal modes analysis as well as knowledge of the natural frequencies and mode shapes for your particular structure is important for all types of dynamic analysis.

1.4.1.2.2 Frequency Response Analysis. Frequency response analysis is a method used to compute structural response to steady-state oscillatory excitation. A frequency response analysis requires that all applied loads in a model vary sinusoidally at the same frequency. Examples of oscillatory excitation include rotating machinery, unbalanced tires and helicopter blades. In frequency response analysis the excitation is explicitly defined in the frequency domain. All of the applied forces are known at each forcing frequency. These forces can be in the form of applied forces and/or enforced motions, as displacements, velocities, or accelerations. The response may be in terms of stress, displacement, velocity and/or acceleration.

As we can appreciate in the Figure 6, the shift in response is called a phase shift because the peak loading and peak response no longer occur at the same time. This quantity as well as the ratio of input to output magnitudes, or amplitude ratio, is constant for a given node at each input frequency, and is a function of the system damping. Theoretically, if there were no damping, the response would track the input exactly.

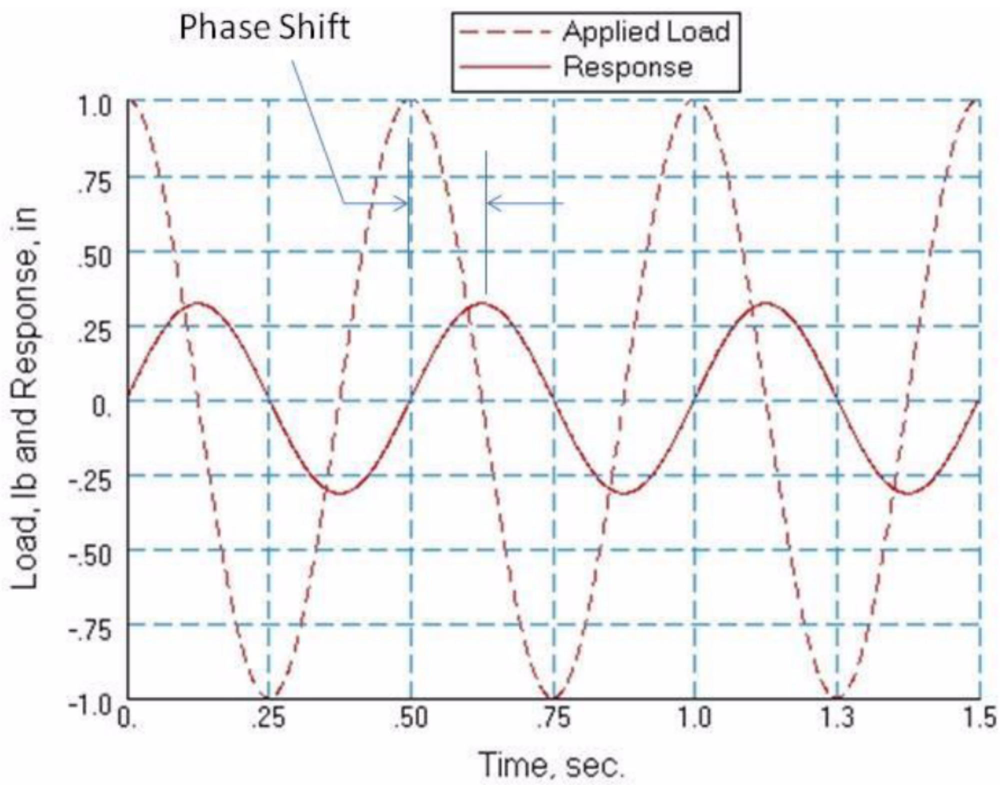

Figure 6: Phase Shift

Moreover, the time between each peak of the input signal is called its period, T, which is the reciprocal of the input frequency, f. Note that the period/frequency of the response is equal to that of the input.

The important results obtained from a frequency response analysis usually include the displacements, velocities, and accelerations of grid points as well as the forces and stresses of elements. The computed responses are complex numbers defined as magnitude and phase (with respect to the applied force) or as real and imaginary components, which are vector components of the response in the real/imaginary plane. These quantities are graphically presented in the next Figure 7:

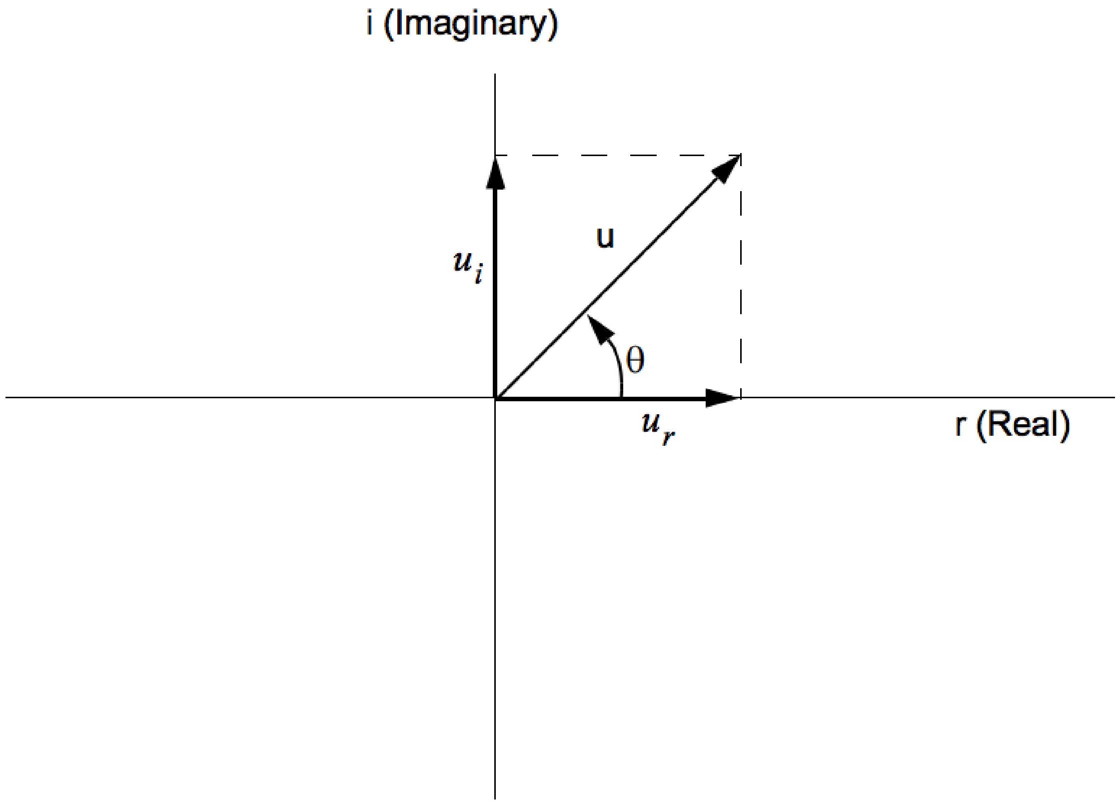

Figure 7: Complex Plane

As we are interested in our research to make a frequency response analysis, we should know that NASTRAN offers two different numerical methods. The direct method, SOL 108, solves the coupled equations of motion in terms of forcing frequency. The modal method, SOL 111, utilizes the mode shapes of the structure to reduce and uncouple the equations of motion, so the solution for a particular forcing frequency is obtained through the summation of the individual modal responses. However, the choice of the method depends on the problem that we would like to solve.

The methods used for generating dynamic loads in NASTRAN are very different from hose used for static loads. As always, the input to a frequency response analysis is dependent on problems goals, but in its basic form, when a system is excited at only one or a few distinct frequencies, the load is applied as in a static analysis and the solution frequencies are specified. However, dynamic loads generally vary with time or frequency. They may also be applied with different phases or time lags to different portions of the structure.

For structures that can experience many excitation frequencies, it is common to input a load versus frequency plot that essentially sweeps each applied load through a range of frequencies scaled according to a defined function.

At this point, we should talk about frequency response results. These results can be calculated in real-imaginary format and in magnitude-phase format. The second one is the most common and practical for understanding the response of the system. Magnitude data available as values of maximum displacement, stress, velocity, or acceleration for each frequency over an entire output cycle. A plot of these data would have frequency values on the x axis and output magnitude on the y axis. Phase data are presented as the value of the phase shift angle for each input frequency.

While these data are excellent for evaluating peak responses and identifying operating frequencies to avoid, they provide little insight on the distribution of the vibrational energy and ways of improving peak responses if these operating frequencies cannot be avoided.

To better understand the total response of the system, the magnitude-phase data must be exported to incremental results at a sequence of input frequency angles, for a given frequency. These output sets can grow in size very quickly. However, when animated in sequence, it becomes clear how the part or system actually responds to the input. So, we could examine our car body to determine how the energy is being dissipated through it. At the end, we get a simple way to implement corrections to better absorb the energy or distribute it to improve performance.

In frequency response analysis, we have two types analysis that it will depend on the many factors that we will comment later:

- Direct frequency response analysis: in this type of analysis, structural response is computed at discrete excitation frequencies by solving a set of coupled matrix equations using complex algebra.
- Modal frequency response analysis: modal frequency response analysis is an alternate approach to computing the frequency response of a structure. This method uses the mode shapes of the structure to reduce the size, uncouple the equations of motion and make the numerical solution more efficient. Since the mode shapes are typically computed as part of the characterization of the structure, modal frequency response is a natural extension of a normal modes analysis.

Some general guidelines can be used when selecting modal frequency response analysis versus direct frequency response analysis, as we can appreciate in the next Figure 8:

	Modal	Direct
Small Model		X
Large Model	X	
Few Excitation Frequencies		X
Many Excitation Frequencies	X	
High Frequency Excitation		X
Nonmodal Damping		X
Higher Accuracy		X

Figure 8: Modal versus Direct Response

1.4.1.2.3 Transient Response Analysis. Transient Response Analysis is the most general method for computing forced dynamic response. The purpose of a transient response analysis is to compute the behavior of a structure subjected to time-varying excitation. The transient excitation is explicitly defined in the time domain. All of the forces applied to the structure are known at each instant in time. Forces can be in the form of applied forces and/or enforced motions.

The important results obtained from a transient response analysis are typically displacements, velocities, and accelerations of grid points, and forces and stresses in elements.

While a frequency response analysis takes place in the frequency domain, a transient response analysis is a study in the time domain. Forces in a transient response are defined to vary with time. Another significant difference between frequency and transient response studies is the boundary conditions used. A frequency response analysis can be solved with no constraints and studied in a free state. Like static analyses, transient analyses must be fully constrained. Transient response analysis must be constrained in the same way that their static equivalents would be. Results are also more straightforward.

As we had in frequency response analysis, depending upon the structure and the nature of the loading, two different numerical methods can be used for a transient response analysis:

- Direct Transient Response Analysis: this direct method performs a numerical integration on the complete coupled equations of motion. It corresponds with SOL 109. You may impose initial displacements and/or velocities in direct transient response.
- Modal Transient Response Analysis: the modal method utilizes the mode shapes of the structure to reduce and uncouple the equations of motion, when modal or no damping is used. So the solution is obtained through the summation of the individual modal responses. It is SOL 112. Since the mode shapes are typically computed as part of the characterization of the structure, modal transient response is a natural extension of a normal modal analysis.

The guidelines that can help us to select between modal and direct transient response analysis are seen in the Figure 9:

	Modal	Direct
Small Model		X
Large Model	X	
Few Time Steps		X
Many Time Steps	X	
High Frequency Excitation		X
Normal Damping		X
Higher Accuracy		X
Initial Conditions	X	X

Figure 9: Modal versus Direct Transient Response

The input to a transient response study is also much like a static analysis except that a time-dependent function can be assigned to a load. Not all loads need a time-dependent definition, and multiple loads can have multiple time functions assigned. As we can guess, these studies are very flexible.

Moreover, one of the critical factors in the accuracy of a transient solution is the time step configuration used. Time stepping defines the time increment between calculated steps and output sets. If the time step is too large, peak responses could be missed or truncated. Conversely, if the time step is too small, the rum time may be excessive and the size of the calculated data set might become prohibitively large. Otherwise, finding the correct size of time steps will probably an iterative process.

1.5 Modal Analysis.

The building block of all dynamic analyses is the modal analysis, which reports the natural frequencies and corresponding with the principal mode shapes of the system under evaluation. It means that when we perform a modal analysis, you solve for the distinct deformation shapes that the vibrating system will assume at each of its preferred oscillating frequencies.

Modal or natural frequency analyses are used frequently when parts being designed or verified are subject to vibratory or cyclic loads. This solution type returns the resonant frequencies for a given structure under a specified constraint set. The mode shapes corresponding to those frequencies are also provided. Modal analysis is extremely important for products mounted on an automobile or a motorcycle that experience vibration resulting from the engines unbalanced forces for example.

The basic steps and terminology of modal analyses should be understood by all FEM users, regardless of the need for further dynamic analysis, due to some of the other uses for this solution that we can find later.

1.5.1 Basics of Modal Analysis.

In addition to the theoretical basis for modal analysis, it is important to have a practical feel for what physical characteristics of a structure affect its natural frequencies. To understand natural frequencies, it is helpful to first consider the mode shapes. In general, simple structures will have a first mode shape corresponding to their most flexible orientation.

For example, a cantilevered beam will bend at its base, and a disc with a pinned circumference will flex in its center. Another way of looking at this is that the first mode shape is the shape with the least potential or strain energy. The shapes of the second or third natural frequencies require more energy to generate and have higher internal strain energies.

While many problems involving harmonic input require computation of many higher order natural frequencies, this discussion focuses primarily on the first, or fundamental, natural frequency. We have to know that this is the most easily predicted and controlled and has many uses beyond dynamic studies.

As mentioned earlier, each mode shape relates to its corresponding modal frequency through the weight and stiffness of the structure as constrained in the analysis. The contribution of weight is best understood by considering inertia. The more weight or inertia, the harder it is to change directions when fluctuating. In addition, the mass moment of inertia, which results from the weight distribution, is inversely proportional to the magnitude of the first natural frequency. Thus, increased mass or weight lumped far from the constraint regions such that the mass moment of inertia about these constraints is high, will serve to reduce natural frequency due to inertial effects.

Now we introduce "spring-back", that is the force trying to keep the beam moving. When the beam is bent past its resting position, its elasticity tries to snap it back into place. However, inertial effects do not allow it to stop immediately and cause it to "overshoot" its mark. The interaction of these two parameters balances out to provide an oscillation speed, which is called the "natural frequency". The first mode or natural frequency is the one at which the beam will vibrate after all external excitations are removed. Additional natural frequencies represent the oscillation of the beam in other deformed shapes or modes. Modal vibration only occurs when the part that we are considering, is being shaken at a frequency that is near to the natural frequency.

In this step, we should talk about damping. Without damping, these two effects might keep an oscillating body motion forever. Damping represents inefficiencies of the material due to energy loss at a molecular level or of the system due to component interaction. Higher damping factors cause the oscillation's amplitude to decrease so the beam slowly, or not slowly, stabilizes. Remember that damping has very little effect on natural frequencies at typical structural values. Damping affects the physical, what is called ?real? response of the beam when shaken or bumped.

1.5.2 Industrial Applications of Modal Analysis

Modal analysis has become a standard approach in today's structural dynamic studies. Typical examples include:

- Analysis of a car body, car components like engine, suspension, exhaust, brake...
- Analysis of aircraft by Ground Vibration Tests, of aircraft components in flight testing in view of aero-elasticity satbility.
- Analysis os space launchers, of payloads, payload fixture.
- In the process industry: analysis of pumps, compressors, piping systems, shafts and bearings, turbine blades, machinery foundations, high precision equipment.
- Civil engineering studies related to bridges, dams, high-rise buildings, off-shore platforms.

- Audio and household systems such as washing machine, loudspeaker, CD-drive, computer rack.

In all these applications, it is the purpose to obtain adequate system models to perform troubleshooting as well as system optimization in terms of mission-critical functional performance parameters such as safety, stability, fatigue life, vibration confort, interior and exterior noise...

1.5.3 Preparing for a Dynamic Analysis.

A modal analysis should always be the first step when considering a dynamic analysis. If the modal study indicates that there are no natural frequencies near the operating speed, then the frequency response analysis may not be required. If the period or duration of the transient load is much longer than the period of the first natural frequency, then the input is less likely to excite a resonance.

1.5.3.1 Dealing with Resonant Frequencies. A natural frequency near an operating speed does not automatically mean that resonance will be excited and vibration will be a problem. In most cases, a first or second mode near an operating frequency will cause noticeable vibration amplification. Adjusting your geometry to move the natural frequencies is somewhat of an art. As we previously know, the natural frequency of a part is related to its weight and its stiffness. However, there are many techniques for increasing stiffness and also for add weight. The proper combination of increased stiffness, reduced weight, and redistributed weight is required to fine tune natural frequencies.

1.5.3.2 Meshing a Part for Modal Analysis. When the end results of a modal analysis are the natural frequencies of a structure, the mesh can be somewhat coarser than would be required for a detailed stress analysis. The density required is similar to that for a gross displacement analysis. Yet, as with a displacement analysis, an overly coarse mesh will result in an overly stiff structure and fictitiously high modes. A modal analysis must be converged as any other structural study.

When a modal analysis is to be used to pre-process a dynamic analysis that requires the actual modal results, the mesh must be developed to capture the behavior desired by the dynamic analysis. If obtaining stresses over a frequency range is the goal of a frequency response analysis, the mesh must be converged for all resultant deformations of interest. If displacement is the only output required, the mesh may be somewhat coarser.

We have already talked a little bit about symmetry, but if we focus on modal analysis, we should have clear that it is better avoiding symmetry in a modal and subsequent dynamic analysis. It is highly recommended. The danger of using symmetry is that there are many more asymmetric mode shapes for a general structure than symmetric. If symmetry is used, the only mode shapes calculated to the specified symmetry constraints. If the frequency of interest corresponds to an asymmetric mode shape in a full model, an important result will be missed in a symmetry model.

If for example, a frequency response or transient analysis will be using the calculated modes to develop more complex data, this problem is accentuated. A modal type dynamic solution bases the stress and deformation results on the pre-existing modes. If any mode that contributes to the final behavior has been missed due to the use of symmetry, the dynamic results will be meaningless.

To sum up with the preparation of our model to a modal analysis, we could say that modal analysis is a fundamental tool for most analysis. Few parts exist in an environment where there is no vibratory or transient loading at some point over the product life cycle.

In most cases where vibration is studied, only the modal analysis is required to make design decisions about the system?s behavior. Consider both natural frequencies and mode shapes to completely understand the modal response. Push the modes away from operating speeds and try to avoid mode shapes that are similar to the deflection resulting from the load input. Much can be gained by making a thoughtful interpretation of modal results.

2 State of the Art

Nowadays, FEM is used commonly in all of the engineering fields. In this section, we would like to show the most important applications and industries where we can find FEM as a tool of developing.

2.1 Applications

We would like to have a review in the most important applications of working with FEM.

- Acoustics: FEM is used to analyze and improve acoustic performance of our products early in product development cycle to gain competitive advantage. We are surrounded by sounds, some pleasant, and quite a few are not. In order for a product to be accepted by customers, manufacturers must pay close attention to its acoustic signature. As we can see an example in the Figure 10:

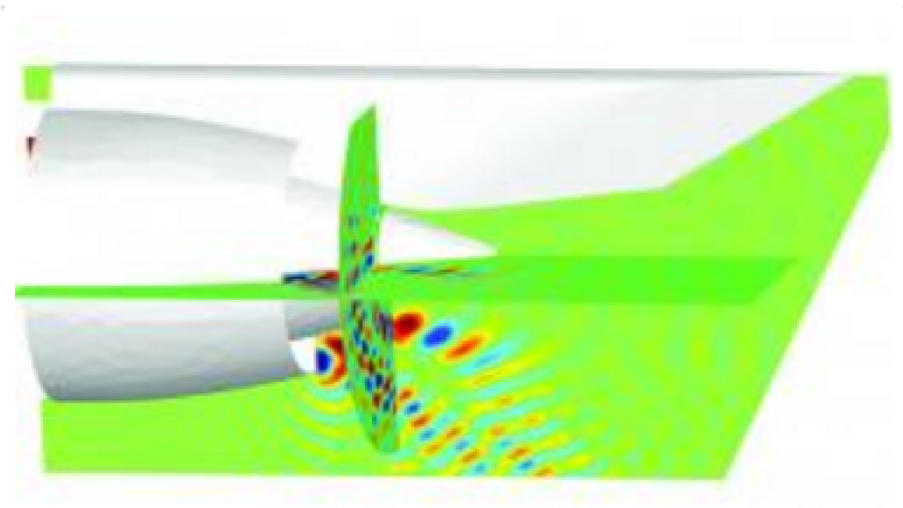

Figure 10: Acoustic Example

- Composites: defined as "engineered materials", composites offer product manufacturers several advantages in terms of weight and performance. However, they also come with several challenges during product design when compared to normal materials such as metals. So with FEM solution capabilities in composites help to analyze and enhance complex composites designs. The typical image of working with composites is seen in the Figure 11:

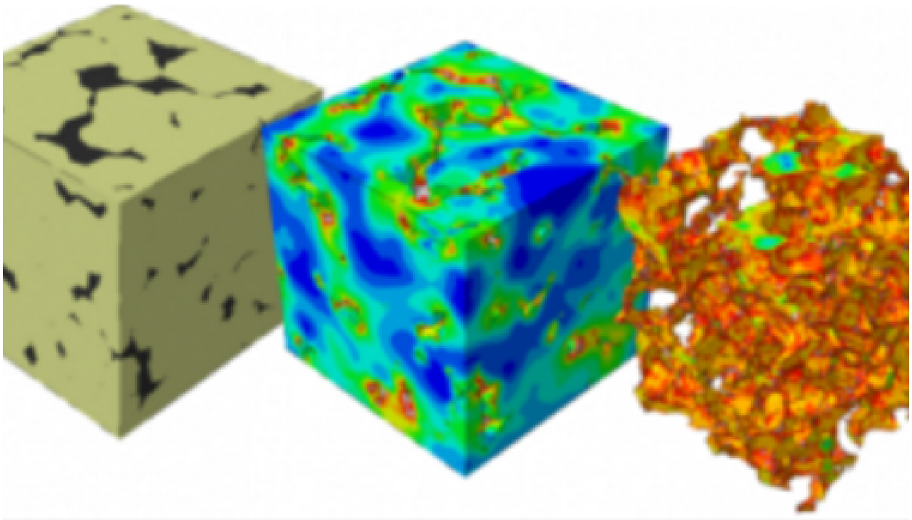

Figure 11: Composite Example

- Crash and safety: there exists many real world engineering situations involve severe loads applied over very brief time intervals. While testing is crucial to analyze these types of loading scenarios, it can be expensive and sometimes economically infeasible to conduct physical tests when the cost of each prototype is high. Moreover, data from a single physical test can be insufficient and companies cannot afford to conduct several of them for more detailed information. The main advantage using FEM is that we are able to built-in multi physics capabilities including fluid structure interaction ensure accurate and efficient solutions for a wide range of real world operating environments, as it seems in the Figure 12:

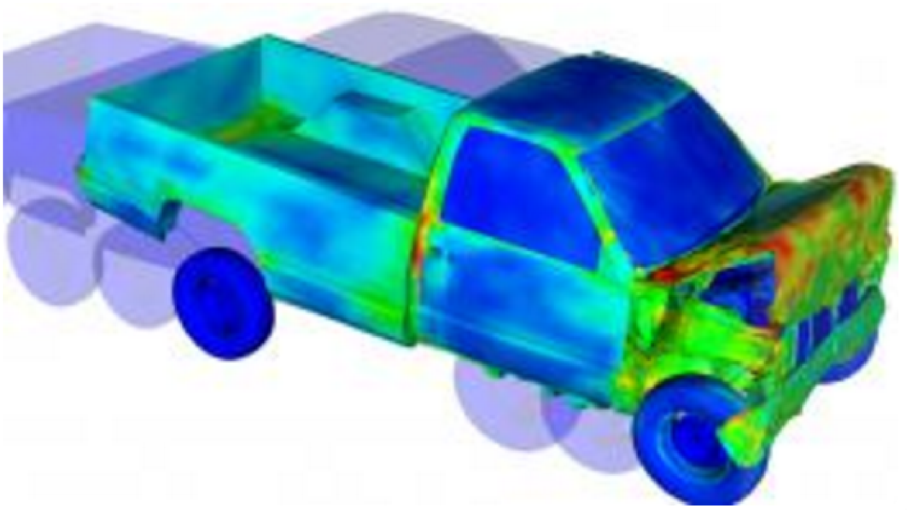

Figure 12: Crash and Safety Example

- Design Optimization: there are FEM softwares that offer a complete set of solutions for optimization, rangine from use of gradient based optimization methods, like sizing, shape and topology optimizations, or calculation of response surfaces to a broader process management to analyze and optimize design across multiple disciplines.
- Fatigue and Durability: one of the most challenging tasks of design and development process is prediction of failure over time. Without knowledge of how a structure might fail, it is harder to improve its safety performance. Physical testing for all possible failure scenarios can be cost prohibitive, as we had with "Crash and Safety" tests. Using FEM tools, we can have a accurate prediction of product life under any combination of time-dependent or frequency-dependent loading conditions. One of

the most typical examples is shown in the Figure 13:

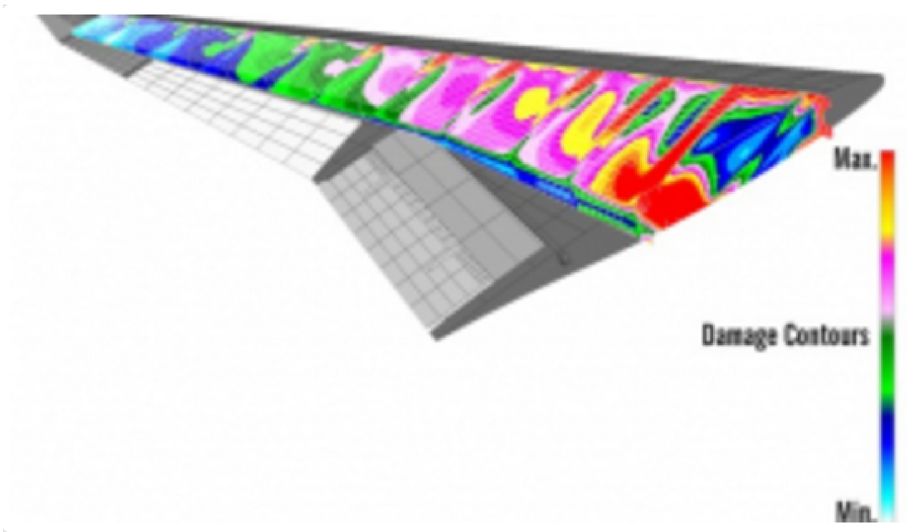

Figure 13: Fatigue and Durability Example

- Multi-body Dynamics: it is possible to easily simulate and test virtual prototypes of mechanical systems in a fraction of the time and cost required for physical build and test. A multi-body dynamic system is one that consists of solid bodies, or links, that ate connected to each other by joints that restrict their relative motion, under the influence of forced dynamics. Motion analysis is important because product design frequently requires an understanding of how multiple moving parts interacts with each other and their environment. We have a multiple examples as engineers, but as for example the one shown in the Figure 14 is an engine, to understand the importance of these analysis:

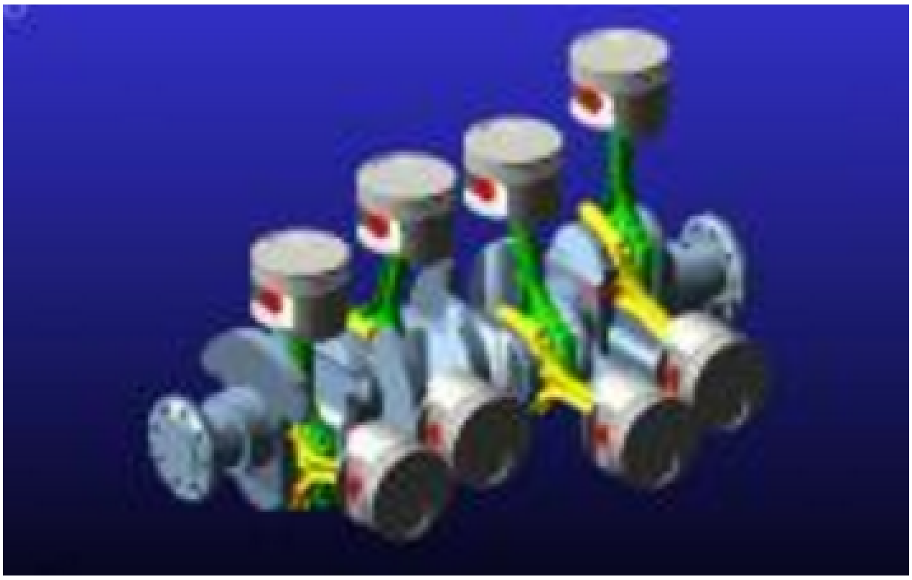

Figure 14: Multi-body Example

- Noise and Vibration: this application is just our main subject in our Master Thesis. So, we are going to develop this theme later. By using FEM, we are able to simulate and predict how a component or system will vibrate under varying operating conditions.
- Nonlinear analysis: all physical processes are inherently nonlinear to a certain extent. Nonlinear response could be caused by any of several characteristics of a system, and many of them exhibit combination of different kind. Using FEM provides solutions to simulate accurately and efficiently systems with any or all of the nonlinearities.
- Rotor dynamics: the field of rotor dynamics is concerned with the study of dynamic and stability characteristics of the rotating machinery and plays an important role in the improving safety and performance of the entire systems that they are part of. Using FEM, we are able to simulate the behavior of rotating machinery and predict critical speed and evaluate the effects of instabilities on virtual prototypes, saving time and money while improving safety. One of the most common examples is as

shown in the Figure 15 with the helicopter:

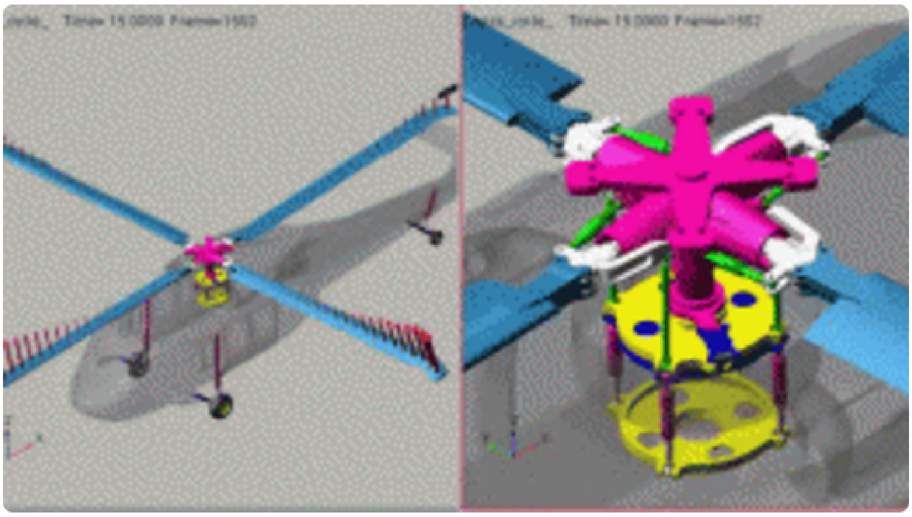

Figure 15: Rotor Dynamics Example

- Structural Analysis: this is one of the most developed fields of FEM. Nowadays, engineers are able to evaluate many different types of designs, giving high confidence that the final design will successfully meet prescribed requirements before the physical product is built. One example of many of them possible, is in the Figure 16:

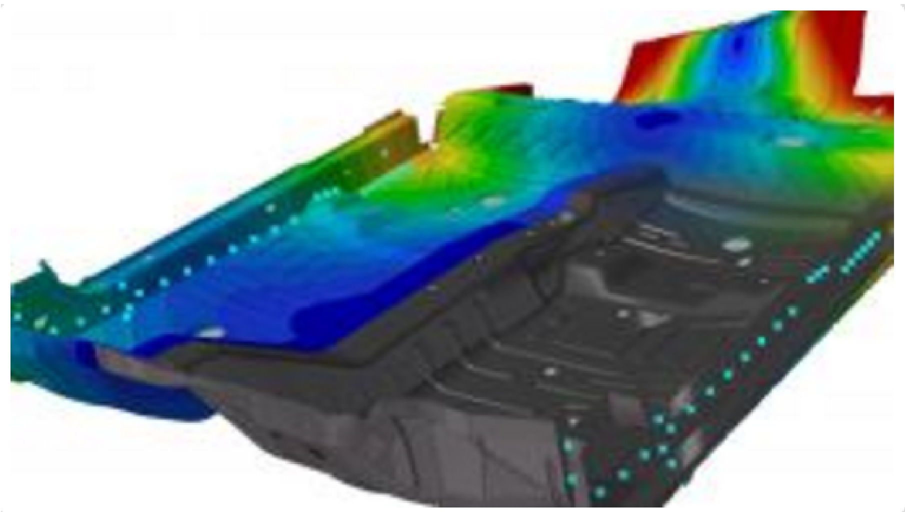

Figure 16: Structural Analysis Example

- Systems and Controls: most of the products we rely on today are actually mechatronic systems comprised of mechanical components, as pneumatic, hydraulic or electrical control subsystems with electronic control units. Engineering these complex systems is a true multidiscipline design challenge that is often not fully tested and validated until late in product development process, leading again to costly design changes. Traditional methods are isolated, so there software that allow us to validate and optimize complete mechatronic system performance.

- Thermal Analysis: FEM softwares enable us to model thermal responses including all the modes of heat transfer, as conduction, convection and radiation. The objective of a thermal study is often to understand the response and performance of a structure. Based on the modeling needs, chained or coupled, analysis can be performed by engineers to study temperature variations and effects on structural behavior, both in terms of the stress response and failure. There are multiple examples, but one of them is shown in the Figure 17:

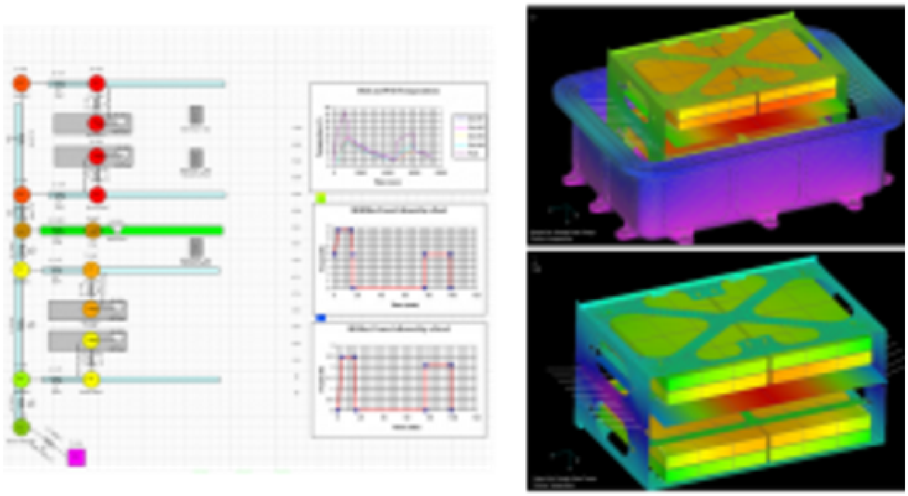

Figure 17: Thermal Analysis Example

2.2 Industries

In this section, we would like just simply to name the industries where FEM is used to see the scope and importance of this kind of software.

- Aerospace
- Automotive
- Consumer Products
- Defense

- Electronics
- Energy
- Heavy equipment
- Machinery
- Medical
- Motorsports
- Packaging
- Rail
- Shipbuilding

3 The Relationship between Finite Element Analysis and Modal Analysis

The properties and techniques of Modal Analysis and of Finite Element Analysis are identified, together with the present advantages and shortcomings of both methods. The interrelationship between these techniques is described, an the contributions of modal analysis to efficient finite element analysis are reviewed. It is noted that the term modal analysis is used to describe the following terms:

- A test procedure for obtaining structural data.
- An analytical procedure for efficient solution of structural dynamics problems.
- The same solution procedure for rotor dynamics analysis.

During the past ten years, the rapid development of specialized test equipment and efficient numerical methods for modal calculation of structures has revolutionized vibration analysis. Certain limitations, which affect the results attained by each procedure, are identified, and the restrictions, which these limitations impose on modal analysis and its application are discussed. The extent to which modal analysis and finite element analysis can be coordinated into an effective diagnostic procedure for vibration analysis is demonstrated by several case histories.

3.1 Types of Modal Analysis

As used in the general literature of vibration analysis, modal analysis may refer to either:

1. A formalized test procedure for identifying the dynamical properties of structures.
2. A mathematical procedure for increasing the efficiency of structural dynamics calculations.
3. A technique for rotor balancing.

Modal Testing is a formalized method for identification of natural frequencies and mode shapes of structures. it utilizes dedicated modal test equipment, and requires a formalized procedure for disturbing, rapping the structure into motion, and then, recording the distribution of the resulting motions throughout the structure. The end results of a modal test are the various natural frequencies, mode shapes, and impedance data of the structure. These data are identified from the digitized input signals using efficient curve-fitting routines. The results are subsequently displayed as impedance plots and mode shapes, and possibly animated as in our case, thanks to our post processor.

Mathematical Modal Analysis is an analytical procedure used to uncouple the structural equations of motion by use of a known transformation, as outlined in the following section. The resulting analysis is then readily achieved by solution of the uncoupled equations. The modal response of the structure is then found through a reverse transformation, followed by a summing of the respective modal responses, in accordance with their degree of participation in the structural motion.

Modal Balancing is a rotor balancing procedure in which the respective modes of a rotor system are first isolated and then corrected for residual unbalance in sequence. The balance corrections used for one mode are carefully arranged in accordance with modal principles so as not to re-introduce the other modes of the rotor system.

The above modal procedures have the following items in common:

- Identification of structural modes and frequencies for further analysis.
- The theory of each procedure is based on an analytical technique known as modal analysis, which uncouples the equations of motion to make possible their efficient solution.
- The orthogonal properties of structural dynamics matrices are utilized either directly in the analysis, or implicitly in the test procedure.

3.2 Practical Modal Analysis Procedures

There exist some general steps to do a practical modal analysis, or modal testing, and we would like to describe some general operations:

- The structural response amplitude is acquired in digital format throughout a prescribed frequency domain, at a given displacement point for excitation applied at a certain point.
- The modal computer automatically develops and stores this digitalized frequency response data in a designated memory for subsequent processing.
- Curve-fit routines are applied to the frequency response data to identify the natural frequencies within the given frequency range. The corresponding mode shapes are extracted from the digitized amplitude data at the natural frequencies.
- The mode shapes may be animated in terms of the simplified structural model, corresponding to those locations at which the response has been determined.
- The modal damping is estimated from the magnitude of the response at each natural frequency. This is often the most approximate structural parameter obtained by

modal testing.

- Modal matrix data are identified for the structure. Output is developed for mass, stiffness, and damping matrices suitable for further computations, based on the structural modal properties. These data are printed out for subsequent use.
- Some software packages permit modifications to be made to the matrix data, to evaluate the influence of possible changes on the natural frequencies and mode shapes. These packages can be run on certain commercially available modal analyzers.

3.3 Finite Element Analysis

Finite element analysis is a computerized procedure for the analysis of structures and other continua. Rapid engineering analyses can be performed because the structure is represented using the known properties of standard geometric shapes. Efficient, large, general-purpose computer codes now exist with appropriate matrix assembler routines and equation solvers for calculation of the following structural properties:

- Static displacement and static stress.
- Natural frequencies and mode shapes.
- Forced harmonic response amplitude and dynamic stress.
- Transient dynamic response and transient stress.
- Random forced response, random dynamic stress.

Finite element analysis used in this manner provides the dynamic properties of structures, including mode shapes and corresponding natural frequencies.

3.4 Advantages of Modal Analysis

The mode shapes and natural frequencies of a structure are its basic dynamic properties. Modal testing is used to rapidly identify these modes and their natural frequencies, and to provide the structural matrices, which governs the modes and natural frequencies. Thus the basic structural dynamic data, when obtained accurately from a valid test also provides a true identification of the structural properties for the modes of interest. These derived matrices are based on the measured participation of the mass, stiffness and damping properties in the modes of interest, for the actual boundary conditions, which the structure is experiencing. These data can then be used directly in a finite element model for the structure or component, for subsequent problem solving, or re-designing the equipment for more optimum dynamic response.

Modern modal analysis test equipment has been developed to provide the maximum convenience in testing and data reduction, and to provide the above mentioned dynamic properties of the structure. All modal analyzers contain dedicated mini-computers for efficient high speed data processing, performed in a prescribed manner in accordance with a specialized test routine. In the hands of an experienced modal analyst, this leads to economical extraction of the data mentioned above.

The advantages of modal analysis are, first of all, that a modal test provides the most rapid and effective procedure available for the acquisition of data on the dynamic properties of a structure. Such testing can often be performed by a skilled technician for later interpretation by a dynamics engineer. Second, modal analysis is an effective analytical procedure for the solution of large sets of structural dynamics equations because it reduces coupled matrix equations (which must otherwise be solved by some iterative procedure) to a set of independent linear equations, each with the well-known closed-form solution given above. Modal solutions can therefore be obtained directly, without further numerical operations. These solutions are then re-combined to form the complete solution to the structural response problem in question.

3.5 Shortcomings of Modal Analysis

The output from modal testing consists of natural frequencies, mode shapes, modal stiffness, modal damping, and modal mass matrices. The main assumption involved in the acquisition of this information is that the structural system is linear, it means that structural displacements are directly proportional to applied loads. In practical structures this condition is not always met. Structural systems may be non-linear to some degree, due to those causes listed below. Non-linearities complicate the extraction of modal data and, where their effect is strong, they may invalidate the results obtained by linear analysis. Non-linear effects may be represent in a structural system due to several causes:

- The material properties may be non-linear, composite structures, viscoelastic materials, elastic-plastic materials, where displacement is non-linearly related to force.
- Where large amplitudes are involved, the geometry may result in displacements, which are non.linearly related to load, like for example large deflections of plate and shell.type structures.
- The structural boundary conditions may introduce nonlinearities, as structures where the number of support points changes, or where he structure is a rotor mounted in fluid-film bearings experiencing relatively large whirl amplitudes.

Such non-linear effects complicate the analysis and tend to introduce errors into the data reduction and curve-fitting estimates of natural frequencies. Such results cannot always be adequately represented by a linear analysis, because the properties change according to the magnitude of the applied load. Errors can range form small errors where minor non-linearities are present to large errors where the non-linear effects are substantial, such as in multiple support structural contact problems (load-dependent indeterminacy).

A further limitation to modal testing is that it does not directly address the forced response problem, nor problems of transient response nor of random response. For problems in which the response to such loadings is of interest, modal amplitude data can be obtained by testing to formulate an efficient structural model for finite element analysis. Once the structural model is available in matrix form, the forcing data can be loaded into a finite element program, and the response to dynamic loading (harmonic, transient, or random) can then be obtained by calculation. The accuracy of such analyses depends of course on the validity of the model, which is generated from the modal test data. It is good practice to make a preliminary natural frequency/mode shape calculation with such data, to verify that the test modal data is consistent with the structural modes and frequencies upon which it is based.

Another limitation of modal testing is that it cannot, by itself, predict threshold conditions for structural stability problems, such as structural buckling, and rotor whirl stability in fluid-film bearings. Again, the modal test structural matrix data from such problems can be developed for subsequent linear finite element analysis, such as the prediction of stability threshold conditions. However, the nonlinear limitation again applies to the post.threshold behavior of such structures. Following the development of an unstable condition, the structure characteristically undergoes large displacements until a new equilibrium condition is found. Such behavior may be highly non-linear, and so beyond the capabilities of modal analysis, and of the structural matrices developed by modal testing.

3.6 Advantages of Finite Element Analysis

Finite element analysis in conjunction with the high-speed digital computer permits the efficient solution of large, complex structural dynamic problems. As the majority of structural dynamics problems are linear they can be solved in the frequency domain using a modal transformation as noted above, subject to certain simplifying assumptions concerning the nature of damping.

Many efficient and comprehensive finite element computer codes are now available to perform structural dynamics response calculations involving harmonic response, transient response, and random response of complex structures. Provision is made in many large codes for storing specific solutions on tape and using these solutions as input to a second related problem, involving the same structure. For example dynamics problems where high temperatures cause changes in the elastic properties of the structure may be addressed by solving for the temperature distribution prior to the natural frequency calculations. The temperature distribution is first obtained for known input conditions, and this solution is used to solve the structural dynamics problem with temperature-dependent elasticity. Similar comments apply to fluid/structural interactions, where the equivalent mass properties of the fluid must be incorporated within the structural mass matrix.

The finite element method therefore offers a very efficient procedure for the calculation of complex linear structures under a variety of dynamic excitation conditions, and under environmental conditions, which may include temperature effects and entrained fluid effects. Where the structure is nonlinear, modal testing may still be used to estimate initial values for mass, stiffness and damping parameters, which can be modified to suit more advanced structural models. This is perfectly related with our model.

3.7 Shortcomings of Finite Element Analysis

Although most linear structural dynamics problems may now be solved accurately and economically, it is still costly to solve most non-linear problems. For such cases a solution strategy must usually be developed on a case by case basis. In such instances the structural geometry and elasticity may be needed i considerable detail in the input data, and the formulation time for such cases may be significant unless suitable pre-processors are available within the code.

The finite element analysis of recurrent structures where a specific segment of the structure geometry is repeated a number of times, are still costly to solve. Problems of recurrent geometry are relatively common, like for example bladed turbomachine structures, axisymmetric structures, building structures and many types of rotating machinery. The geometry of such structures often closes on itself, that are called "ring structures". The total structural matrix is still symmetrical and tridiagonal, but the dynamical matrix contains off-diagonal elements, which may substantially increase the local matrix bandwidth. This causes a corresponding increase in computation time. Efficient computation of such recurrent components has been undertaken by special finite difference procedures, but sub-routines to undertake such computations are not yet in widespread use.

4 Starting our Learning Process

Along our project we are going to use three different software, as we have mentioned before in the introduction. The main reason as we are going to use this three software is also explained before. For that reason, as we had not any experience with this new softwares, during almost the first month, we spend out time learning how to use them.

To do so, we had the opportunity of learning using the tutorials from ANSA and µETA, where we could start to understand more or less how they worked and their features.

Moreover, we would like to add that this period of self-learning was very complicated and hard for us.

4.1 ANSA

ANSA is a software designed to work with the automotive industry. With ANSA we are able to modify and take care of the majority of the parts of our car body. Moreover ANSA offers us a special module to use with NASTRAN, that allows us to mesh and create accurately elements.

ANSA was the first software that we started to learn. In the first tutorials, we were taught the basic commands that later we would use. So, the way that we learnt with this tutorials were so easy: we had the ”pdf.file” and we followed the instructions to reach the same results. Sometimes as you could probably imagine, that it was hard because our results were not the same as in the files. Maybe it took a long time to know what was our error, but we are sure now that it had a positive impact because we had to find the error by ourselves and that was really helpful.

Another important field with the ANSA tutorials was related with fixing some problems of one geometry, that later it could give us some mistakes when we wanted to mesh it. In that step, we learnt the basic geometric commands, and also the difference between add or remove some parts from the piece.

The mesh module was the most important one. As we are working with FEM, we know that the results are totally related with the mesh that we create. We learnt how to make a different kind of mesh, depending if the geometry was simple or complicated, we could use different commands. This commands were specially to use NASTRAN as processor.

We observed the difference between the same geometry but with different mesh, and it was very clear the importance of the person who is meshing, because you could appreciate his experience in that field and how it works perfectly.

To sum up with this paragraph, our project was not focused on meshing the car body, but we had to learn and understand how ANSA mesh because maybe along our research we are needed to change some parameters to achieve good results.

4.2 MSC-NASTRAN

NASTRAN is our processor. We are not going to work literally with NASTRAN, because as we are going to explain later, we were given a "text.file" where we are able to change all the parameters and simulation conditions of our model.

This file is also possible to get it from ANSA. However, it is easier to try to understand the "text.file" an change directly all that we want.

Related with our processor, it is necessary to talk about the time of simulation and the number of CPUs required. One of the most problematic issues that we had to face during the development of our project was the time required for our computers to finish the simulation of the car body.

One of the first things that we had to discover was the number of CPUs that our computers had. Each of them had 8 CPUs. So with that information we were able to run each of our simulations with a concrete number of CPUs, using the command "dmp=number of CPUs" that the computer would use to solve our model" for LINUX.

Whereas it would seem that as much as CPUs we used, faster would be the resolution time, it was not true at all. The reason why it did not work that simple fact we do not still yet, but every time that we tried one simulation with one or more CPUs, the computer could not finish the simulation or there were "FATAL MESSAGES".

So after many attempts, we decided to run the simulation without assign the number of CPUs, so we let the computer to use alternatively its own CPUs. The problem is obviously the time that each simulation takes. After all of our simulations, we are able to affirm that the time depends exclusively in the changes that you ask to the computer. For example, if we change the stiffness of our spring and we set a "stupid" value that it does not correspond with the reality, the time of that simulation will be so long. One example is that one of our first ten attempts took more than 35 hours.

One of the first "FATAL MESSAGES" that we obtained in our first simulations was the capacity of the "SCRATCH" folder. NASTRAN uses this folder when it is running to put certain files that it would use in the background of its simulation. So for that reason, after each of our simulations, we have to be sure that this folder is empty, because the files that are in this folder are not relevant and they are only related with the simulation process and not with the results.

4.3 µETA

µETA is the perfect "partner" of ANSA. Both of them are made by the same company, so they are totally complementary. This post processor is also very used in the automotive industry, thanks to the multiple options integrated to make a range variety of analysis.

With µETA we are able to show the animation of the hole car body, and that information is very useful for us, as we can read the output asked before very easy. Moreover, with the animations, we are able to appreciate clearly the behavior of the car body against the different signals.

Another way of introducing our results is using curves. µETA allows us to export the data in the DIAdem format, that it is a software developed to work with a big files and make curves easily. We use also that software because our mate Mr. Krampe had used it to do his Bachelor Thesis.

One of the advantages of this program is that we are able to analysis every point individually, or two points in the same graphic. And it is very intuitive to use it.

5 Analysis of our NASTRAN program

In this section we are going to explain the most significant points of the file that we used for our multiple simulations. But we have to clarify first that this file was made by VW, and our job ,as we are going to explain later, is to change some parameters to obtain good results.

We are going to explain this program in some steps, focus in the main points that at the end will have a great influence in our model.

5.1 First Steps

The first step is to explain the kind of solution that we are looking for. In the Figure 18 we can see two important aspects:

```
$=====================================================================
$
$       Uebergabemodell Golf Cabriolet Verdeck geschlossen
$
$       von Volkswagen Osnabr.ck (O-TEGF-1)
$
$       an die Hochschule Osnabrueck
$
$=====================================================================
$
$---------------------------------------------------------------------
Nastran BUFFSIZE=32769
ID vw365
SOL 111
CEND
```

Figure 18: Solution Definition

The first one corresponds with the command "BUFFSIZE". We use this command after our first simulation was done, because it happened one problem with the space required from the computer to achieve the hole simulation. So with this command, we solved this problem and it never occurred again the problem with the capacity of our computer.

The second one and probably one of the most important aspects is the "SOL 111". As we could see later in another steps of the program, there are two "SUBCASES" created. In each of these cases, we asked different things to the processor, but the main requirement is this type of solution.

We can describe the "SOL 111"as "Modal Frequency Response". Frequency response is the quantitative measure of the output spectrum of a system or device in response to a stimulus, and is used to characterize the dynamics of the system. It is a measure of magnitude and phase of the output as a function of frequency, in comparison to the input. It is used to compute structural response when is applied a steady oscillation, like rotating machinery, helicopter blades, etc. Forces can be in the form of applied forces and/or enforced motions (displacements,velocities, or accelerations). In nature the oscillatory loading is sinusoidal. The steady-state oscillatory response occurs at the same frequency as the loading. The response may be shifted in time due to damping in the system.

The important results obtained from a frequency response analysis usually include the displacements, velocities, and accelerations of grid points as well as the forces and stresses of elements. The computed responses are complex numbers defined as magnitude and phase (with respect to the applied force) or as real and imaginary components, which are vector components of the response in the real/imaginary plane. Two different numerical methods can be used in frequency response analysis. The direct method solves the coupled equations of motion in terms of forcing frequency. The modal method utilizes the mode shapes of the structure to reduce and uncouple the equations of motion. The choice of the method depends on the problem.

In a Frequency Response Analysis, referred to the excitation definition, the force must be defined as a function of frequency. Forces are defined in the same manner for the direct and modal method. The following Bulk Data entries are used for the frequency-dependent load definition:

- RLOAD1: Tabular input-real and imaginary
- RLOAD2: Tabular input-magnitude and phase
- DAREA: Spatial distribution of dynamic load
- LSEQ: Generates the spatial distribution of dynamic loads from static load entries
- DLOAD: Combines dynamic load sets
- TABLEDi: Tabular values versus frequency
- DELAY: Time delay
- DPHASE: Phase lead

As we will observe later, we use some of these entries for our car body, to get the right Frequency Response Analysis. Moreover, each of these input are defined in the Reference Guide, where we can take a look if we have any question about how to define one or the meaning of one position. That guide is really useful and it has helped us a lot during all our project.

As the force must be defined as a function of frequency, there are six Bulk Data entries that you can use to select the frequency at which the solution is to be performed. Each specified frequency results in an independent solution at the specified excitation frequency.

5.2 Set Definition

We have mentioned before that in NASTRAN we can work creating different subcases depending in our interests. That way of working is perfectly understood if we are able to create SETs definition of a concrete number of grid point or elements that we are looking for their results. In our case, it is quite obvious that we are going to ask the results for the same points where the sensors of the real car has been tested.

As we can see in the Figure 19, we have chosen the number of the grid points that were the same in the real car in the laboratory.

```
SET 100 = 51,52,6301,6304,1100,1105,1200,1205,
          5000003, 5000004, 6000003, 6000004,
          5710001, 5710002, 6801006, 6802026,
          9001001, 9000901, 9001011, 9000911, 9005002,
          1051787, 34799, 36077, 254546, 104208,
          313600, 300388, 59646, 254717, 1048578,
          30404, 27499, 54565, 41814,
          41216, 42989, 257906, 40664
```

Figure 19: SET Grid Points Definition

The way to get the number of this points was using ANSA directly. In the model we have defined exactly the same car, so more or less in the position of the sensors, we take the information of that grid points that we are interested in the results.

In the next Figure 20, we observe another SET made in this case with elements. In our case, we are not paying too much attention to this elements, because we are more interested in the grid points, but it is just an clarification that we can also make a SET with elements.

```
SET 200 = 152, 155,
          6229, 6230, 6231, 6232,
          5000063, 5000073, 6000043, 6000053,
          9000001, 9000002, 9000003,
          9000016, 9000017, 9000018,
          5000823, 5000833, 6000823, 6000833
```

Figure 20: SET Elements Definition

So at the end, this tool of creating "SETs" is very useful because we could change so easy the points or elements that we would like to study.

5.3 SUBCASES

The SUBCASES are made to obtain different results in function of the input. Moreover we can ask different results for each on of them, as for example, displacements, displacements only for all the nodes in one SET, or just defined the number of mode shapes.

The first SUBCASE that we are looking in is in the Figure 21:

```
SUBCASE 101
  LABEL =MODALANALYSE
  SET 111 = 1 THRU 35
  OMODES = 111
  DISPLACEMENT(PLOT) = ALL
  ANALYSIS=MODES
```

Figure 21: SUBCASE 101

This case corresponds with the Modal Analysis, referred to the SET 111 created before. The mode shapes that we ask are from the 1 to the 35. This is a parameter that perfectly we can change in other simulations. Displacements are set up for all the grid points, as we could appreciate better in the video that our post processor can make.

The second SUBCASE is called 300 as we can see in the Figure 22. We can call them as we want.

```
SUBCASE 300
   LABEL =  1mm verschraenkt-SPCD
   DLOAD = 300
   DISP(SORT2,PUNCH,PHASE) = 100
   ACCE(SORT2,PUNCH,PHASE) = 100
   MEFFMASS(PRINT,PUNCH,SUMMARY,PARTFAC)
   ELFORCE(SORT2,PUNCH,PHASE) = 200
```

Figure 22: SUBCASE 300

In this point, we have the input signal amplitude of 1mm. This is the same value that has been introduced in the experimental car. Another part of this case is the input introduced. As we have seen before, DLOAD is the command used to combine dynamic load sets. To get a better knowledge of the definition of this load, we could look in the Figure 23:

```
DLOAD,300,1.0,1.0,101,-1.0,102,-1.0,201
,1.0,202
```

Figure 23: Dynamic Load 300

The dynamic load is defined crossed. It means that for the grid points selected, the input signal in this SUBCASE is going to represent the bending moment of our car body. The points that appear here are the points that in our model correspond with the contact points of the four wheels with the entrance signal source. We have checked this using ANSA.

Moreover, in this case the displacements and the accelerations are asked only to he SET 100.

The last SUBCASE is seen in the Figure 24:

```
SUBCASE 310
   LABEL =  1mm gleichphasig-SPCD
   DLOAD = 310
   DISP(SORT2,PUNCH,PHASE) = 100
   ACCE(SORT2,PUNCH,PHASE) = 100
   MEFFMASS(PRINT,PUNCH,SUMMARY,PARTFAC)
   ELFORCE(SORT2,PUNCH,PHASE) = 200
```

Figure 24: SUBCASE 310

The only difference between this SUBCASES is the input signal. For this SUBCASE we have another DLOAD as we can see in the Figure 25:

```
DLOAD,310,1.0,1.0,101,1.0,102,1.0,201
,1.0,202
```

Figure 25: Dynamic Load 310

5.4 Tabular input

We introduce the amplitude of 1mm using "RLOAD1! command. It means that we introduce the input using a table. This table is going to be defined by two main values. Therefore in the first table, as we can see in the Figure 26, we are going to have for 1000 Hz, 1mm of amplitude in the input signal. So in the second table, we are going to have for the same frequency, a value of 0 mm:

```
$--- Table y = 1.0 ---------------------------------------------
TABLED1        1
            0.0     1.0  10000.     1.0    ENDT
$--- Table y = 0.0 ---------------------------------------------
TABLED1        2
            0.0     0.0  10000.     0.0    ENDT
```

Figure 26: Tabular Input

This table is going to be used to define the RLOAD1, as a factor that we can multiply. So for that reason, we can observe in the Figure 27, that as we want an amplitude of 1 mm, we do not need to multiply for another value.

```
$ VL
RLOAD1       101     111       0       0       1       2     DISP
SPCD         111 5000003       3       1
$
$ VR
RLOAD1       102     112       0       0       1       2     DISP
SPCD         112 5000004       3       1
$
$ HL
RLOAD1       201     211       0       0       1       2     DISP
SPCD         211 6000003       3       1
$
$ HR
RLOAD1       202     212       0       0       1       2     DISP
SPCD         212 6000004       3       1
```

Figure 27: RLOAD1 Definition

5.5 Include files

The "include files" are all the files that are created to simulate the car body. Each component is described in one file. For that reason, it is really simple working like this, because if we are looking for a change of one parameter in one part of the car, we know before where we could change the value. All the includes files are written like in our main file, as we can appreciate in the Figure 28:

```
INCLUDE 'Includes/vw365_03_tb_rohbau_dyn_003.ninc'
INCLUDE 'Includes/vw365_02_tb_SQRT_100203_stat_mod_001.ninc'
INCLUDE 'Includes/vw365_02_tb_Frontend_100205_stat_mod_003.ninc'
INCLUDE 'Includes/vw365_02_tb_abschiermung_vo_100316_stat_mod_003.ninc'
INCLUDE 'Includes/vw365_02_tb_Streben_Hi40x10_100125_stat_mod_001.ninc'
INCLUDE 'Includes/vw365_02_tb_HiRa_Hi_100423_stat_mod_003.ninc'
INCLUDE 'Includes/vw365_02_tb_HiRa_Vo_100203_stat_mod_002.ninc'
INCLUDE 'Includes/vw365_02_tb_MQT_100503_stat_mod_003.ninc'
INCLUDE 'Includes/vw365_02_tb_Material_100114_mod_stat.ninc'
INCLUDE 'Includes/vw365_02_tb_Anbindung_100416_mod_008.ninc'
$
$INCLUDE 'vw365_02_tb_Koppelinclude_mod_008.ninc'
INCLUDE 'Includes/vw365_02_tb_Koppelinclude_mod_008zu.ninc'
INCLUDE 'Includes/vw365_03_tb_karo_100705_mod_002_NSM.ninc'
INCLUDE 'Includes/vw365_02_tb_Antrieb_mod_100203_001.ninc'
INCLUDE 'Includes/vw365_02_tb_HKL_100401_mod_001.ninc'
INCLUDE 'Includes/vw365_02_tb_Kotfluegel_mod_100113_001_MAT.ninc'
INCLUDE 'Includes/vw365_02_tb_Kotfluegel_mod_100218_001.ninc'
INCLUDE 'Includes/vw365_02_tb_VWPQ35GP_April08_ok.ninc'
INCLUDE 'Includes/vw365_02_tb_VWPQ35GP_April08_ok_MAT.ninc'
$
INCLUDE 'Includes/vw365_02_tb_Modulsitze_vorn_100223_mod_001a.ninc'
INCLUDE 'Includes/vw365_02_tb_FKL_100401_mod_002.ninc'
$INCLUDE 'vw365_02_tb_Verdeck_mod_100401_001.ninc''
INCLUDE 'Includes/vw365_02_tb_Verdeckzu_100913_001.ninc'
INCLUDE 'Includes/vw365_02_tb_Tank_mod_100122_002.ninc'
INCLUDE 'Includes/vw365_03_tb_Tuer_li_100801_mod_001.ninc'
INCLUDE 'Includes/vw365_02_tb_Tuer_li_100218_001_MAT.ninc'
INCLUDE 'Includes/vw365_03_tb_Tuer_re_100801_mod_001.ninc'
INCLUDE 'Includes/vw365_02_tb_Tuer_re_100218_001_MAT.ninc'
INCLUDE 'Includes/vw365_02_tb_SQRTHI_100222_stat_mod_001.ninc'
```

Figure 28: Include Files

6 Our Car Body-VW Golf Cabriolet

Our project consists on trying to get a validation of a FEM car model from a real one. For so, all our data are going to be compared with the simulations of our mate who is working with the real car. For that reason, we would like to introduce the way of testing from our colleague before explaining our FEM model.

6.1 Real Model Car Body

Describing the real experiment from our colleague, it will be helpful to explain better our tests. The car is located in the laboratory. The structure used to do the test with the car body is the one seen in the Figure 29:

Figure 29: Test Structure

The structure is composed basically of four pillars that are going to be the attendant to create the input signal of the car body. That four pillars are connected to a central station where our colleague could modify this signal to make different kind of tests and where all the sensors are also connected. This central station is also used to get all the test results, as we can see in the Figure 30:

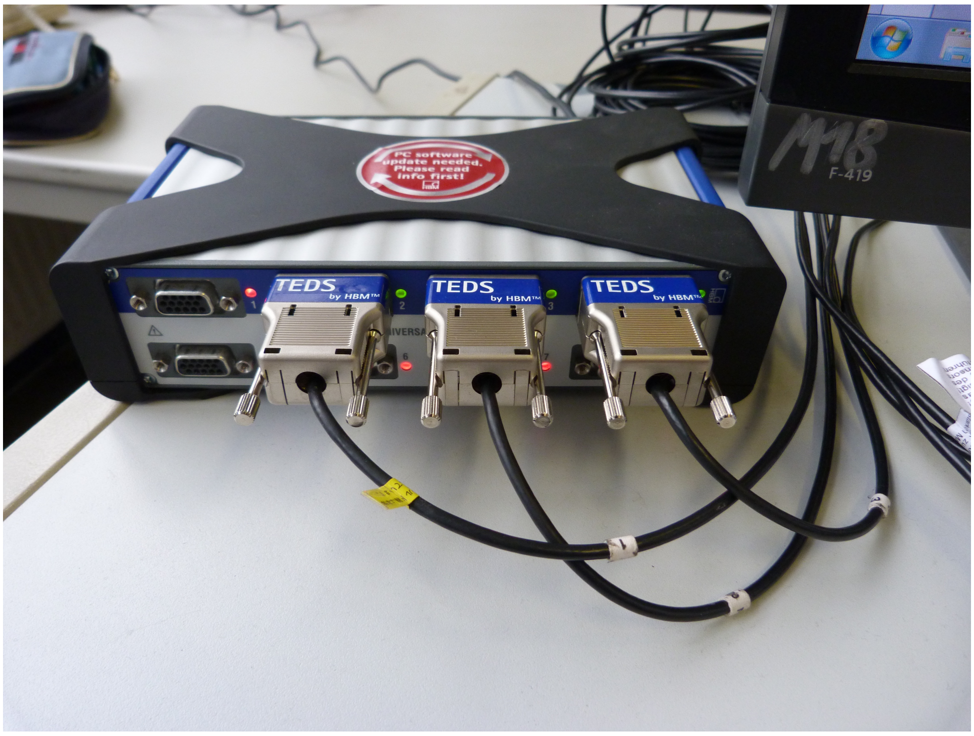

Figure 30: Test Station

In the Figure 31 we are going to see the car located in its correct position in the structure:

Figure 31: Car Located in the Structure

As we can appreciate here, the four wheels are exactly located right above the four pillars. This is quite obvious, because the wheels are the elements in contact with the road, so they are the parts that are going to transmit all the input signals to our car body.

6.1.1 Test Points

There are ten points selected by each of the faces in the car body to do the tests. These points are selected around all the car body to try to get good results from all the most important parts of it. In the Figure 32 we can appreciate one face of the car body with all the points:

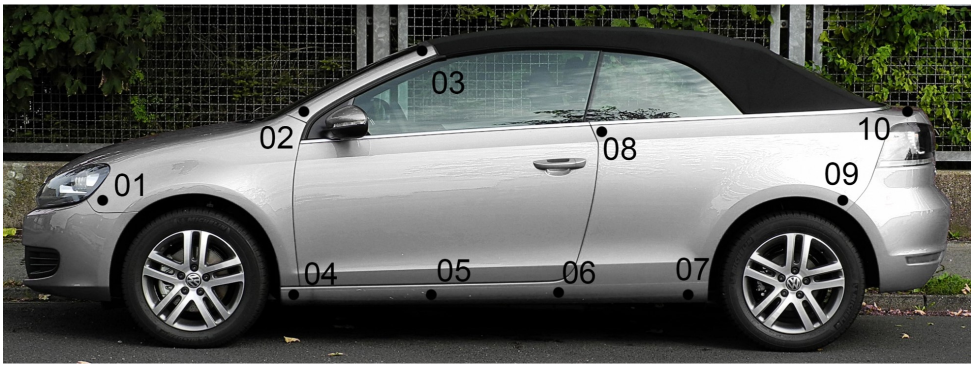

Figure 32: Test Points

However, we would like to pay more attention to two points that are going to be more important to our project (the importance of these tow points will be explained later). These two points are the number 3 and the middle point of the wheel.

In the Figure 33 we can see the A pillar point. This point is really important for the comfort of the driver and occupants of the car. Further in a cabriolet car, the influence of having opened or closed the roof will take more importance and studying this point will be really useful.

Figure 33: A Pillar point

There is a difference between the middle point of the wheel and the rest of the points. The main difference is that the mid wheel point has not been tested by Mr. Krampe, but for our project, this point will have an enormous influence to study the behavior of the car body changing the wheel stiffness properties. The midpoint is the one seen in the

Figure 34, that it corresponds with the left front wheel:

Figure 34: Mid wheel Point

6.2 FEM Model Car Body

As we have mentioned before, we were given the data file from Volkswagen. In this data file is included everything, it means that we could open it using ANSA to have a better look in the structure of the car. As we said, ANSA is the preprocessor that we would like to use to get the correct information of the parameters and parts of the car body that we were looking to change.

We can say more clearer that opening the data file in ANSA allowed us to get a better approach to the real car, and from that view, we knew which include files we could modify to change some parameters or boundary conditions for the next simulations.

In the next Figure 35 we can have a great overview about our car body:

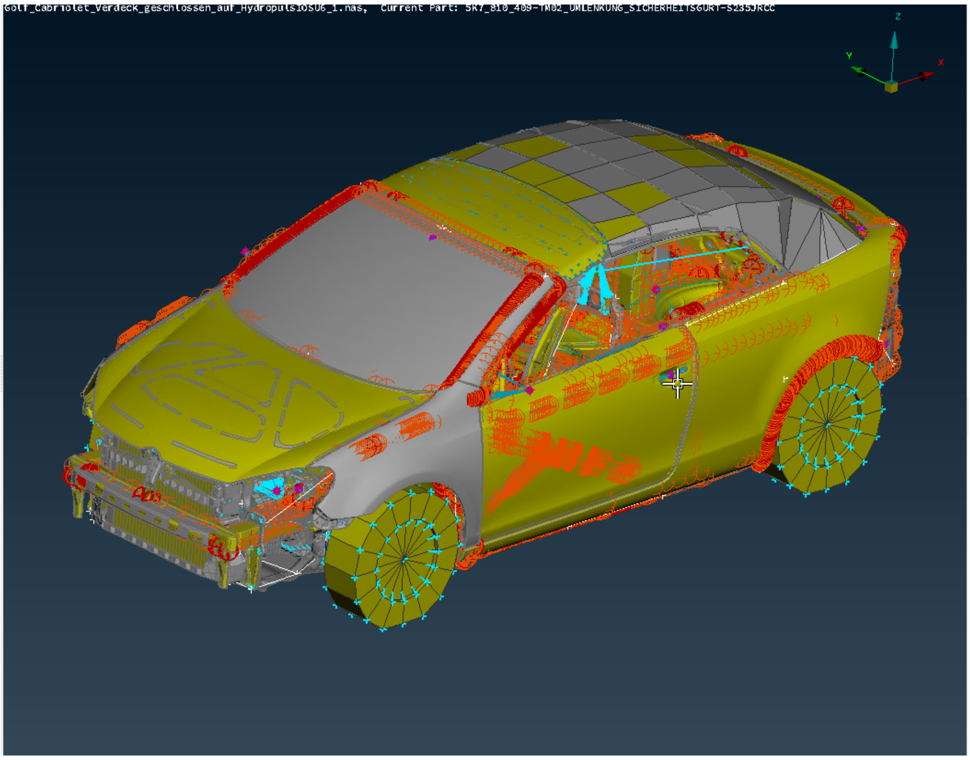

Figure 35: Car Model Overview

One of the best advantages of using ANSA is that we can do a lot of things with our model, like removing some parts, only having a look in some elements, etc.. As we can appreciate in the next Figure 36, it is clearly seen how easy is removing one part that we are not interested using the command "NOT":

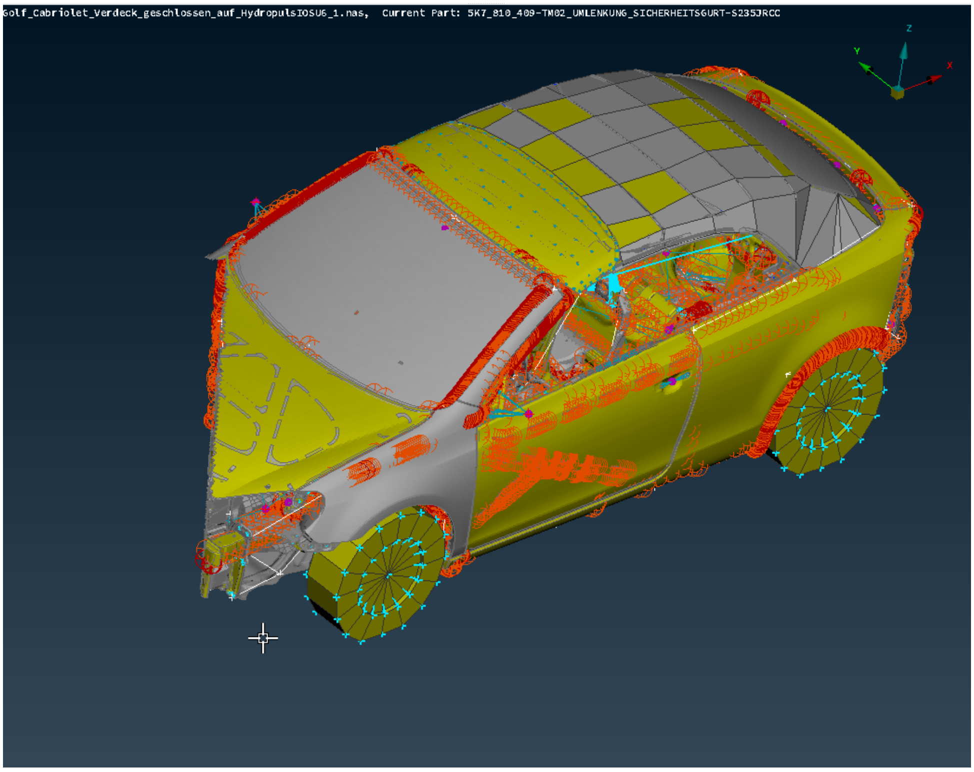

Figure 36: Modification Using ANSA

In the next Figure 37, it is seen what happened to our car body and we have a lot of opportunities to look carefully in some parts:

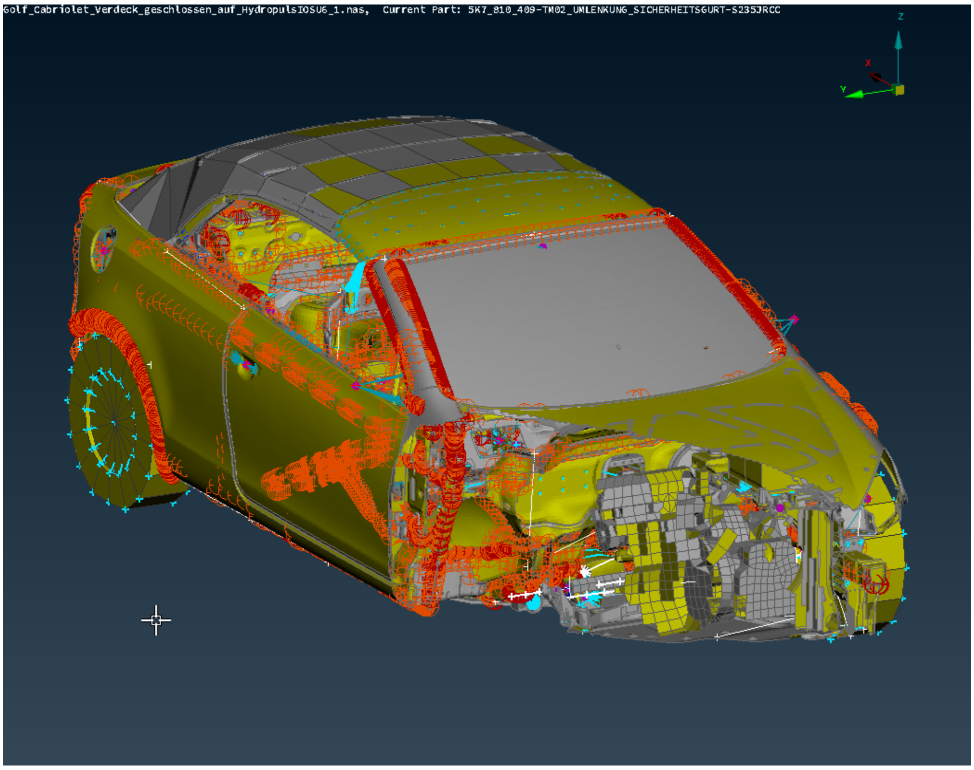

Figure 37: Another View Using "NOT" Command in ANSA

Another useful tool is only to work with the elements that you want. As for example, we would like to take care only about the "RBE2" elements. So in the left tree that appears, we only have to select this elements and the rest of the car disappears, as we can see in the Figure 38:

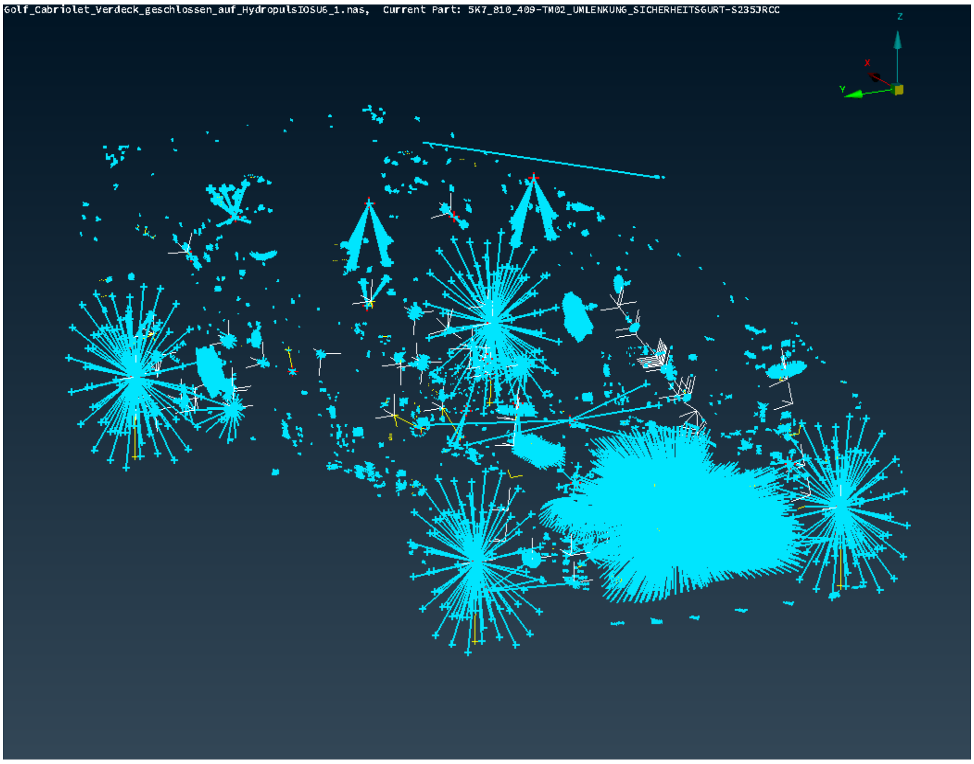

Figure 38: Selecting only one Type of Elements in ANSA

Another useful tool from ANSA is that we can select one element and the software gives you the information about that element. We used this tool too many times. Instead of trying to guess what kind of element would be, we directly select the element and it was said everything, as we see in the Figure 39:

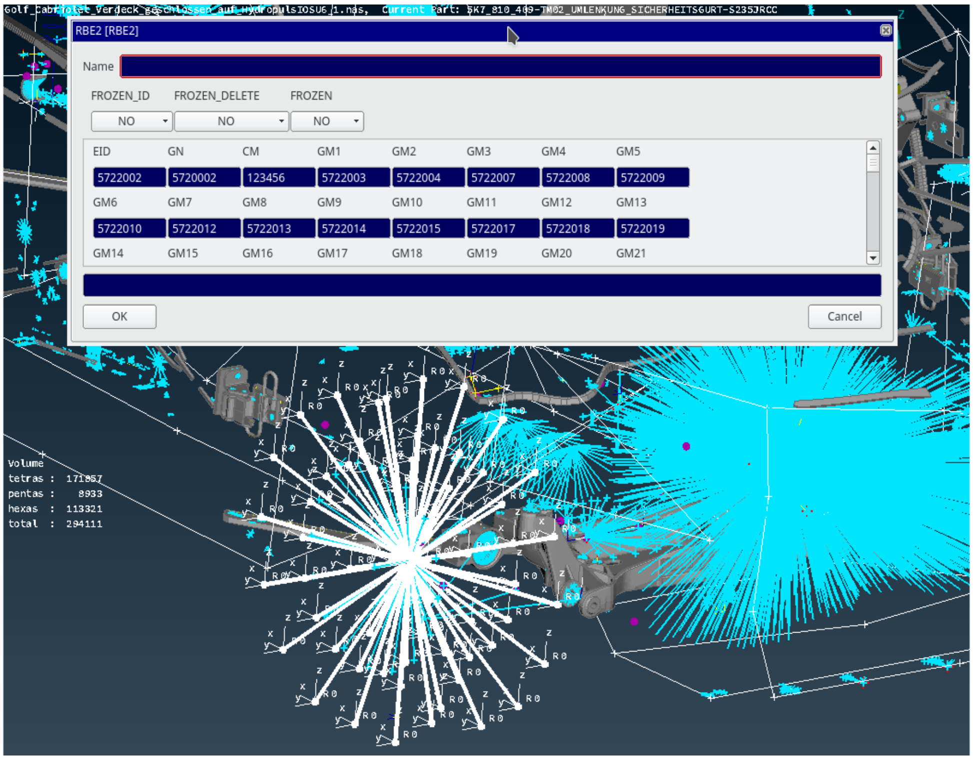

Figure 39: Picking one Element in ANSA

Moreover, this is probably one of the most useful tools that we have used along the project, because the program gives us the enough information to know in which include file we could take a look for changing some properties. In this picture, we can see that this element represent the wheel of the car, and it corresponds with the "EID 5722002", so from this information, we could look in the correct include file if we would like to make any change in this property.

6.3 Test Definition

A modern modal analysis test can easily comprise more than 1000 degrees of freedom. This is due to the complexity of many built-up products, where many components can exhibit a local behavior. In particular, when modal analysis is applied to validate and update a FEM model, a sufficiently dense grid of test points is needed. The FEM model grid is anyway much finer than the test one, requiring a substantial reduction. So optimally selecting the minimal set of needed test points is very valuable in optimizing the test efficiency.

In case a FEM model is available, this model can be used precisely to this purpose: to select the optimal location of a minimal set of measurement sensors and excitations for the optimal control and observation of a number of selected target modes.

However, in our project, we are trying to validate our FEM model from some points that we were given before of having had the FEM model. At the end, it could be the same, because thank to the experience, we know that the points that have been simulated were the most important ones.

7 First Comparison Between the Results of the Real Car and Our Model

As we have defined before, the main objective of our project is to achieve a validate FEM model from a real car body. To do so, we have to compare the results from the real car with the results from our FEM model for some points in the car body. And at then end, we should have the same diagrams respect to the frequency. If we obtain the same graphics, we will be able to say that our FEM model is a correct validation of the real car body.

In this way, as we had two computers to do the simulations, the first step was to test if we ran the the same NASTRAN file in both of the computers, we got the same results. Moreover, we should had to check the duration of these simulations.

After that the first simulations were done and we got the same results, we were concerned about the fact that each of the simulations took a lot of time. We looked to solve this problem using more CPUs. As we know, if we use more than one CPU, it will mean theoretically that the simulation would take less time than if we only use one.

Unfortunately, we discovered that our computers had 8 CPUs, but the problem was that if we ran the simulation with more than 1 CPU, NASTRAN detected a "FATAL MESSAGE", so we could not work with more than one CPU at each time. For that reason, one of our main disadvantages to develop this thesis was the time of simulation, because we only could try one simulation per day, and only during the "night" because the computers were too slow when they were running with the solution.

Another important problem that we had since the beginning, was related with the size of the SCRATCH. The problem was related with the size of the folder that NASTRAN uses to do some calculus in its background. The fatal message is show in the Figure 40:

```
USER FATAL MESSAGE 1012 (GALLOC)
DBSET SCRATCH IS FULL AND NEEDS TO BE EXPANDED.
USER ACTION: SEE THE MSC.NASTRAN CONFIGURATION AND OPERATIONS GUIDE OR
             KB8012329 ON THE MSC WEB SITE FOR METHODS TO MAKE LARGER DATABASES.
```

Figure 40: DBSET SCRATCH Fatal Message

To solve this problem, we looked in many sites, and we tried with one command written at the beginning of our NASTRAN file. The command is "Nastran BUFFSIZE=32769". So from this point, all our simulations had this initial command.

As we said before, the aim of the project is to approach the results of the NASTRAN model, to the real ones. For this, we changed the elements and their parameters of the original ANSA file.

In the NASTRAN document, we chose different points of the car to sight their outputs, displacements or accelerations. They were programmed in NASTRAN as a common set to assign them easier. The reason of the selection of these points is because of the duration solving the program. If we decided to get the outputs of more than 1 million points that there were in the model, it would have been clearly slower than if we chose just a few. Therefore, in µETA post processor, we just had the results of this set of points that we had selected. However, we could choose the most important points of the car to observe what was happening there in each frequency.

7.1 Output of the Initial Model

Initially, we disposed of the results of the real car in two significant points. The first one was the highest point of the car in the upper left corner of the A-pillar. The second one was one point inside of the back right wheel. This second point was difficult to compare with the ANSA model because that part of the wheel was not well represented with solid bodies. As we will study later, that part of the suspension system was represented with curves and lines and was more difficult to find the exactly point.

However, the A-pillar point was really easy to detect in ANSA because this vertical component of the lead window was made with a lot of grid points. Thus, we chose the grid point 27499 as the point that the sensor, accelerometer, was disposed in the real car.

This point is shown in the in the Figure 41:

Figure 41: A Pillar Point

Then, we will compare the results of the real car with the results of the initial model in the A-pillar point. As we have said before, the NASTRAN file contrasts two subcases, the torsion one (in the file is the subcase 300) and the flexion one (in the file is the subcase 310). To observe clearly the outputs, we separate the subcases in two graphs.

In addition, we had two different kind of curves that depended of the car geometry. In the bottom of the car there were two rods in a diagonal position which function was to absorb the efforts of that part of the car. We can observe these components in the the Figure 42. These bars were not present in the commercial cars, we just used it to observe the different behavior of the car with and without these components. Logically, if we added this element to the car, the whole car was more comfortable and the car could absorb the efforts of that part of the car.. So, we had two different kind of curves describing the behavior of the car with and without the diagonal bars. However, we just studied these elements in the first part of our project and in the rest of it, we will assume that the bars are in the car.

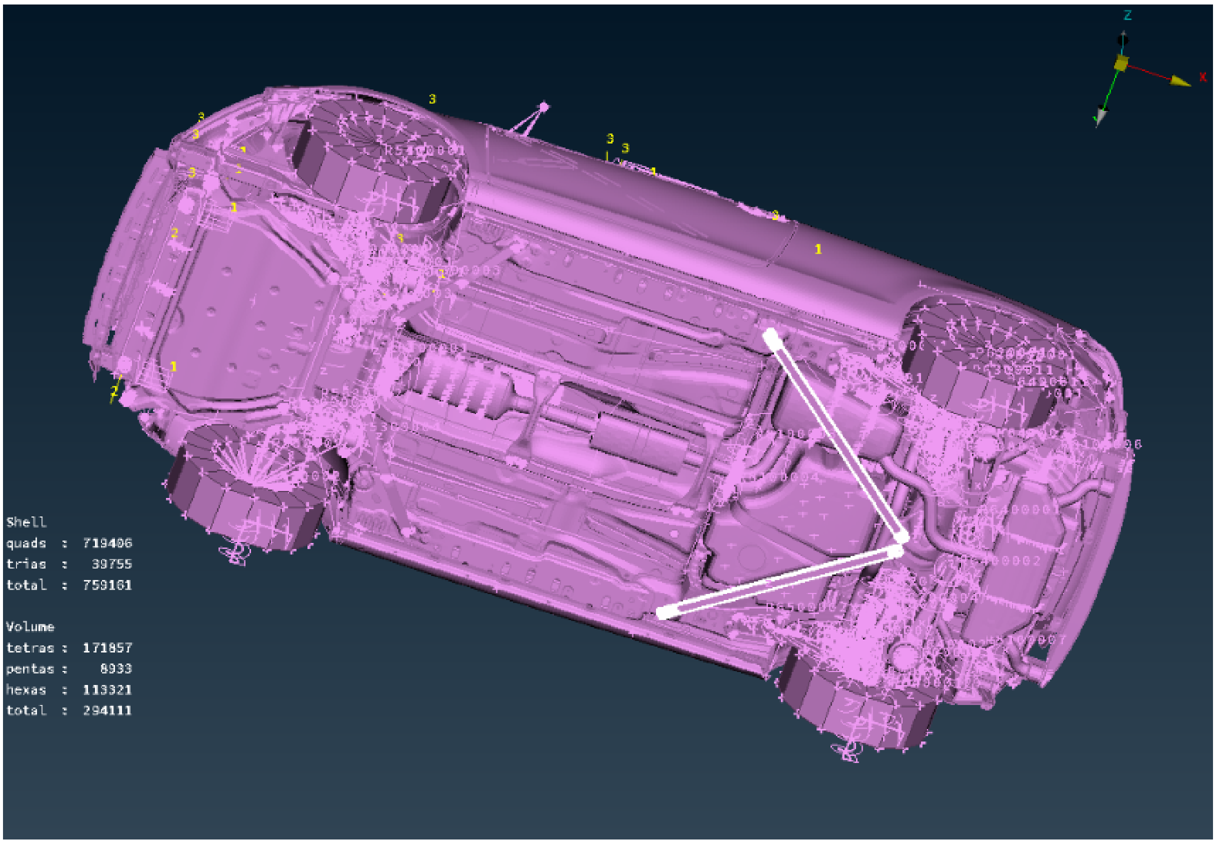

Figure 42: Diagonal Striving

In the real tests, the input was a signal with 1mm of amplitude. The frequency was changing over time to get the final output that we had. Obviously, the input was different in the two cases studied. In the torsion one, the opposite wheels (for example the front left wheel and the back right one) had the same phase and contrary to the other wheels.

Other important aspect was the magnitude that we were using. As we know, the transfer function is the relation between the input and the output of a system. Therefore, we wanted to know which was the transfer function of the real model to apply it in our µETA solutions. After studying the dates of the results that we had, we observe that the transfer function was as follows:

$$Transfer\ Function = \frac{Acceleration\ A - Pillar}{Aceleration\ Wheels} = \frac{Displacement\ A - Pillar}{Displacement\ Wheels}$$

Figure 43: Transfer Function

In the real curves, the transfer function was related with the displacement. As the displacement in the wheels was a constant with the value of 1mm, the transfer function had the same shape than the displacement curve. However, the amplitude of these curves was important because it depended of the magnitude of the accelerations and displacements. Thus, the curves in the real model were in (m/s^2) and in our model they were in (mm/s^2). So, we chose the acceleration as our output because it was similar than the transfer function in the real car.

Besides, we studied the curves in the range between 10 and 30 Hz. The reason was that we had just the real results in these frequencies. Therefore, although we could check in µETA other frequencies, we just focus our work in these range.

7.1.1 Torsion in the A-pillar in the Initial Model

The first perception that we had when we saw the graph of the acceleration in the A-pillar point in the torsion case of our model was that it had the same shape than the real output. The three curves that represented the three different axes were disposed in the same position than the real ones. However, when we look deeply, we saw that there were differences between them.

First of all, the amplitude of the output was quite different. As we can see in the Figure 44, the highest peak was around 22 (m/s^2) and in our model, as we can see in the Figure 45, the highest peak was around 17,6 (m/s^2). Also the real curve was more clean. Thus, our model had more peaks in the curve which conferred to the model a lot of harmful frequencies.

However, this graph was quite similar to the real one because the axes were disposed in the same position (the highest amplitude corresponded to the Y-axis) and the highest peak was in both graphs around 17,6 Hz.

Besides, the curves of the car without the diagonal striving had the same relation with the curves with these elements in our model than in the real case. Thus, these curves had a slightly smaller amplitude than the curves with the diagonal file in the NASTRAN program. Also, we observed that the curves were displaced around 1Hz to the left of the graph. Therefore, the highest peaks were in smaller frequencies.

Finally, we have to remind that all the curves that we are studying are with the car without the roof. So, in the real case the roof was opened and in the initial model we deleted properly this part of the body car.

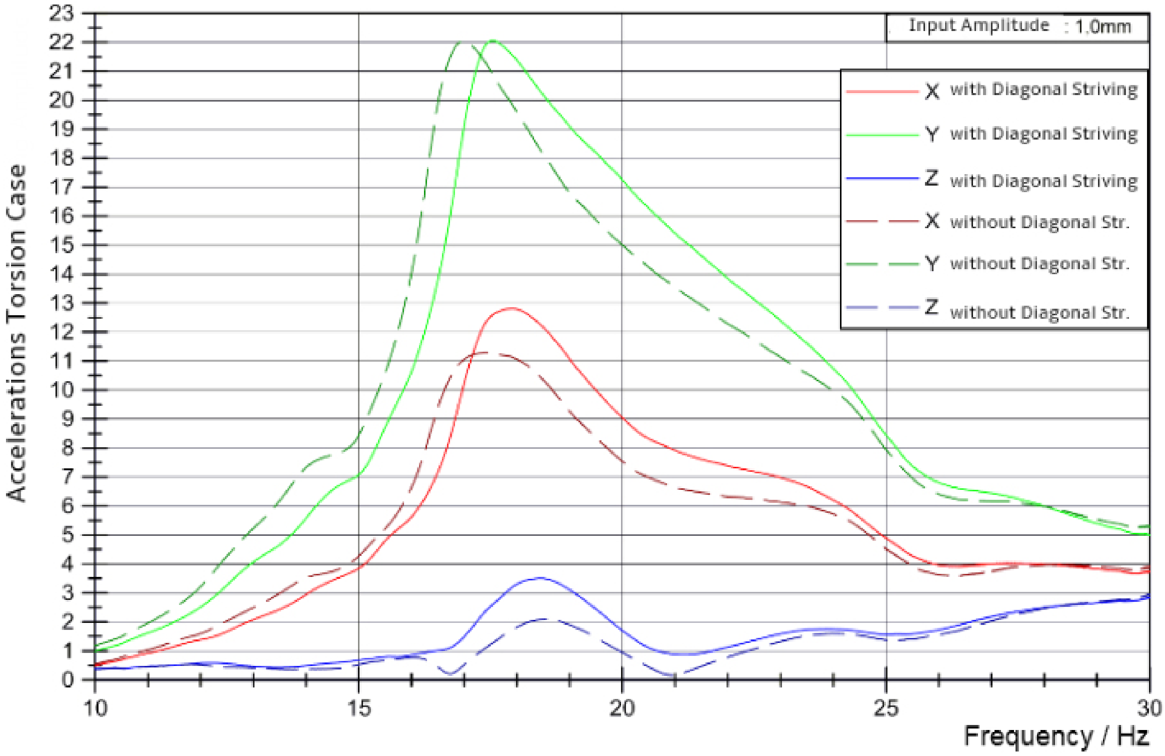

Figure 44: A-Pillar Real Torsion Case

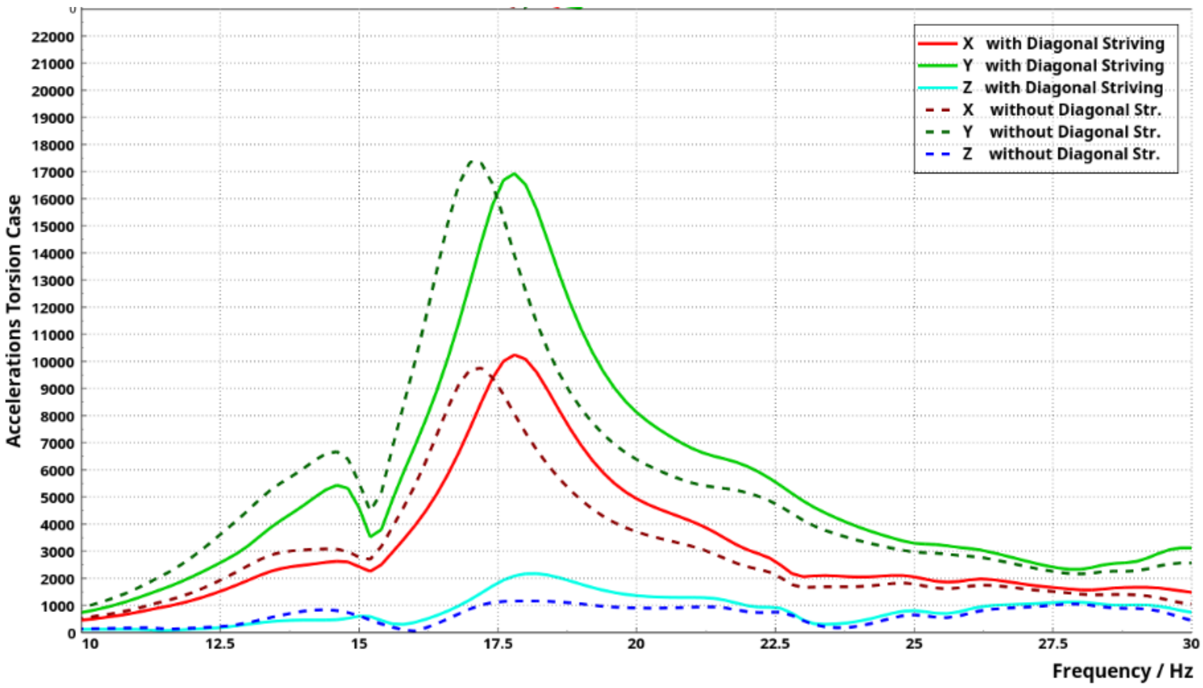

Figure 45: A-Pillar Initial Model Torsion Case

7.1.2 Flexion in the A-pillar in the Initial Model

Although the torsion curves were approximately like the real ones, the flexion curves were quite different.

As we can see in the Figure 46, the shape of the curves are completely different in both cases. In the real model, the curves had one remarkable peak (except the acceleration in the X axis that it had a constant peak in the range between 18,5 and 22Hz) and they were crossing each other, specially the Y axis. Conversely, as we can see in the Figure 47, the curves of the initial model had also one peak, but they were not crossing each other in the range between 10 and 30 Hz. This feature was also in the torsion curves but in both cases, the real one and the initial one.

Also, the curves of our model had the highest peak in other frequency than the real ones. In the real car, the X and Z axes curves were in 18,5Hz and they were around 21,5Hz in the initial model. In the Y axis case, although in the real model it was in other frequency than the other axes. Actually, there was not any representative peak in this axis in our model. The amplitude was constant along the range of frequencies.

However, we could observe that the graphs had some characteristics in common. First, the position of the axes was the same in both cases. Thus, the axis with the highest amplitude is the Z axis and the lowest amplitude corresponds to the Y axis. In addition, in the low frequencies, the amplitude in the Y axis was even higher than in the other two axes. This feature was also common in both graphs.

In addition, we could observe again some characteristics between the real curves and our initial model curves with and without the diagonal striving. In the flexion case, the curves without those components were slightly higher. Also, like in the torsion case, the curves were a little displaced to other smaller frequencies.

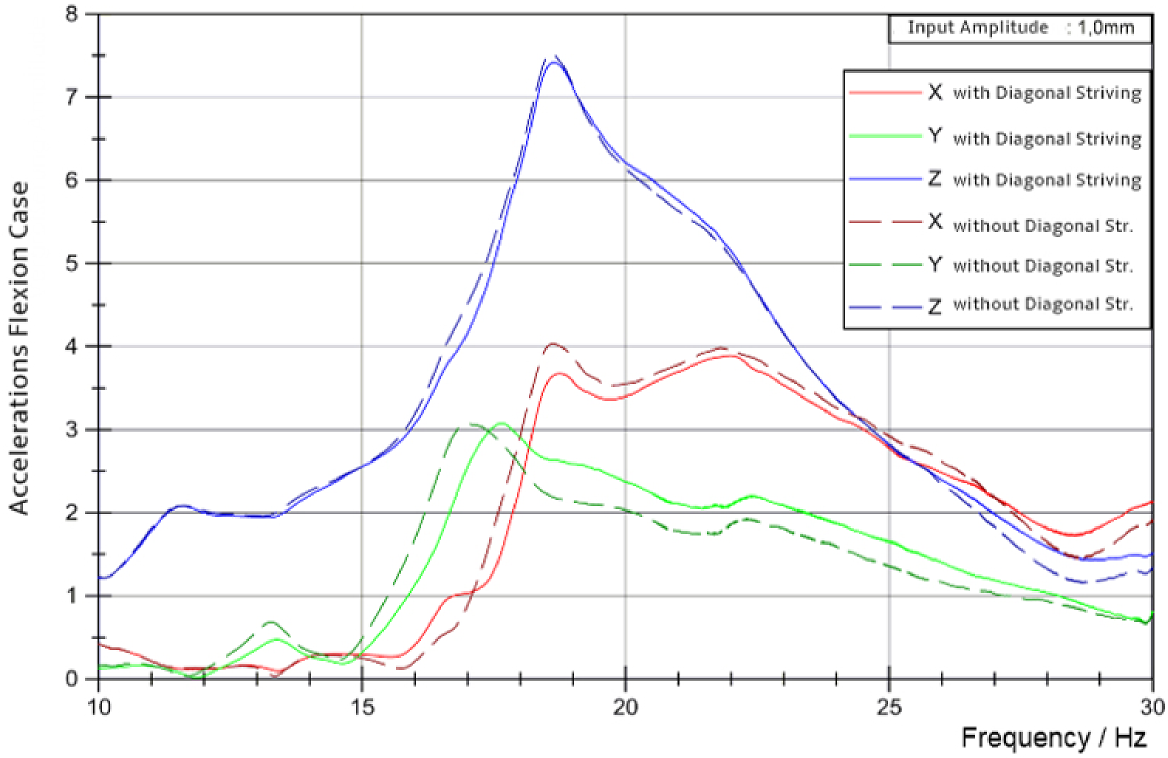

Figure 46: A-Pillar Real Flexion Case

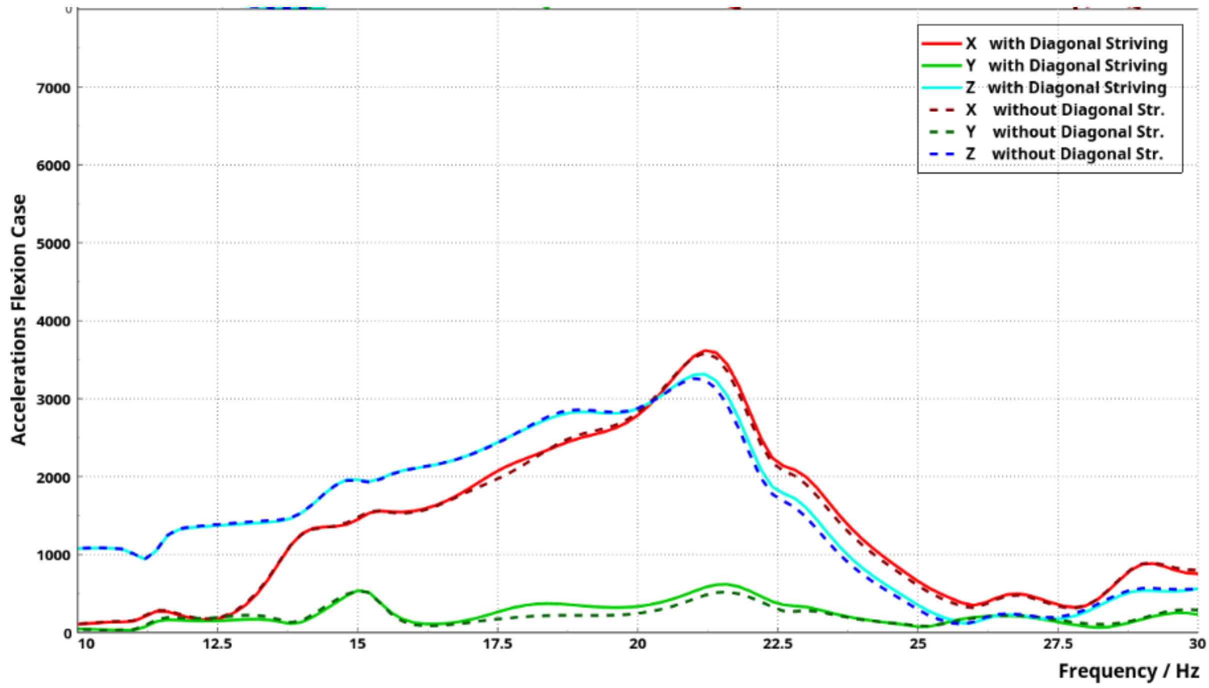

Figure 47: A-Pillar Initial Model Flexion Case

Therefore, we had to improve both graphs to approach the results to the real ones. To do this, we changed the parameters,commands and other elements in the ANSA file. This part will be shown in the following chapters.

8 Fundamental Parts of our Car Body

After that we were sure that our both computers obtained the same results from the same data file, we started to change some fundamental parameters of the car that would have a big influential in its behavior. We are talking about all the elements that transmit the input signal to the rest of the car body: suspension system.

8.1 Suspension System

In a case of forced vibrations, as in our model, the amplitude of the resulting vibration tends to be amplified around the natural frequency. This amplitude will be higher, the smaller the damping. Therefore, the presence of damping always limits the amplitude of vibration.

If the frequencies of excitation forces are known, it is possible to avoid the resonances changing the natural frequency of the system. However, if the system has to operate in a specific speed range, as in a car or other kind of vehicle, it might not be possible to avoid the resonance along that range. In such cases, the main solution is to provide damping, through the suspension system, in order to control the dynamic response.

In addition, the suspension system mades the vehicle safer and more comfortable for the passengers. The reason is that the damping absorbs the reactions produced in the wheels passing over the ground's irregularities. It avoids to transmit the vibrations to the bodywork.

8.1.1 Properties and Parts of the Suspension System

The suspension system is the item set which joins the chassis with the rolling surface of the vehicle. As explained before, it is necessary to assure the contact between the wheels and the road and to reduce the vibrations transmitted.

The suspension needs two qualities: elasticity and damping.

- Elasticity to avoid the transmission of the road's irregularities in raps. The elasticity depends on the vehicle's weight. The more compression, the larger period of oscillation. Therefore, the vehicle will be more comfortable but less secure. The element which fulfills that function is the spring that it does not just reduce the raps transmitted to the frame, also the reaction of the weight of the car to the wheels. The elastic suspension elements are designed to store kinetic energy possessed by the body car to return it later. Ideally, an elastic element should return a hundred percent of that absorbed energy, but in reality it does not. Although there are different elements which are part of the elastic suspension of the vehicle like steel plates joined by clamps, torsion bars, etc... we only will study the springs.

- Damping to avoid the excessive swinging and to assure the contact between the wheels and the road. It pretends to return the spring to its original position as quick as possible, absorbing the kinetic energy transmitted to the car body. Without the damping, a vibration would never finish because the energy stored in the springs would not be dissipated in some way. In this case, it is the damper, commonly called shock absorber, that performs this function. We are not going to study the operation of the damper, just say that it consists in a piston which moves inside a cylinder with the up and down movement of the wheels.

The suspension system it is perfectly seen in the Figure 48:

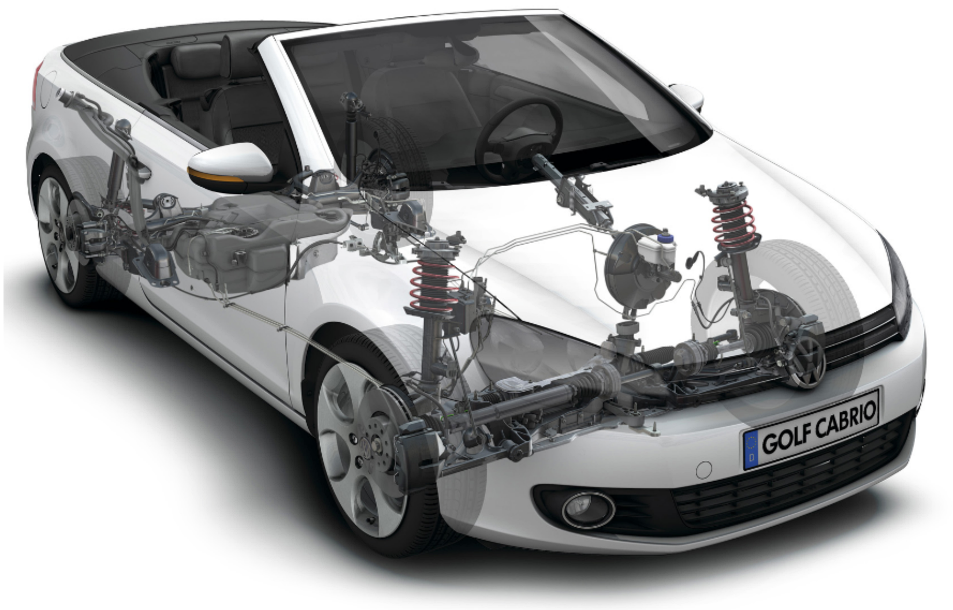

Figure 48: Suspension System in a Golf Cabriolet

8.1.2 The Suspension System in our Model

Thus, the damping and spring of the car are directly related with the results of our project so for, we will study them deeply.

The real Golf Cabriolet, as any other kind of car, includes a damper and a spring connected with each wheel. Their function is to connect them with the chassis. Each damper and spring are related with their correspondent wheel. However, we can differentiate the front suspension (related with the front wheels) and the back one. Then we discuss each of these suspensions.

8.1.2.1 Suspension System Elements Before starting with all the description of the suspension system, we would like to define before some basic ANSA elements that help us to understand better how is represented the suspension system in compared to the real model.

- CELAS1: scalar spring element which joins two geometric grid points. We can see how it is defined in the Figure 49:

CELAS1	EID	PID	G1	C1	G2	C2

Field	Contents
EID	Element identification number
PID	Property identification number of a PELAS entry
G1,G2	Grid point identification number
C1,C2	Component number

Figure 49: CELAS1 Elements Definition

- RBE2: specifies a single point with independent degrees of freedom and an arbitrary number of points with dependent degrees of freedom. Thus, there is no relative displacement between the points that have been selected with dependent degrees of freedom. Its definition is shown in the Figure 50:

RBE2	EID	GN	CM	GM

Field	Contents
EID	Element identification number
GN	Identification number of point to which all the independent degrees of freedom are assigned
CM	Component numbers of the dependent degrees of freedom
GM	Identification number of point to which the dependent degrees of freedom are assigned

Figure 50: RBE2 elements definition

- CDAMP1: it corresponds with the damper. Its definition is in the Figure 51:

CDAMP1	EID	PID	G1

Field	Contents
EID	Element identification number
PID	Property identification number of a PDAMP entry
G1	Grid point identification number

Figure 51: CDAMP1 elements definition

8.1.2.2 Front Suspension In our model, the front suspension connects the middle point of the wheel with the top point of the damper. We can see this last point opening the car hood.

Although we can see the bodywork as a 3D element in ANSA, the parts of the suspension system are represented as lines, curves or spirals. Now, we will study their components.

The suspension system starts in the middle of the wheel. The tires are represented as spring elements and rigid bodies which join all the points of the wheels as in an individual piece and give its properties. These elements are connected with the 3D solid pieces which compose the tires. Therefore, the program approaches the influence of the wheel in our model.

Similarly, the damper ends in a set of rigid rods. The aim of these elements is to lock the damper to the car hood. Once again, in the top of this element, there are springs which emulate the behavior of the suspension system. Also, the damper is close to the vertical position, because this is the best way to damp the movement of the car.

In the Figure 52 we can appreciate better the front suspension:

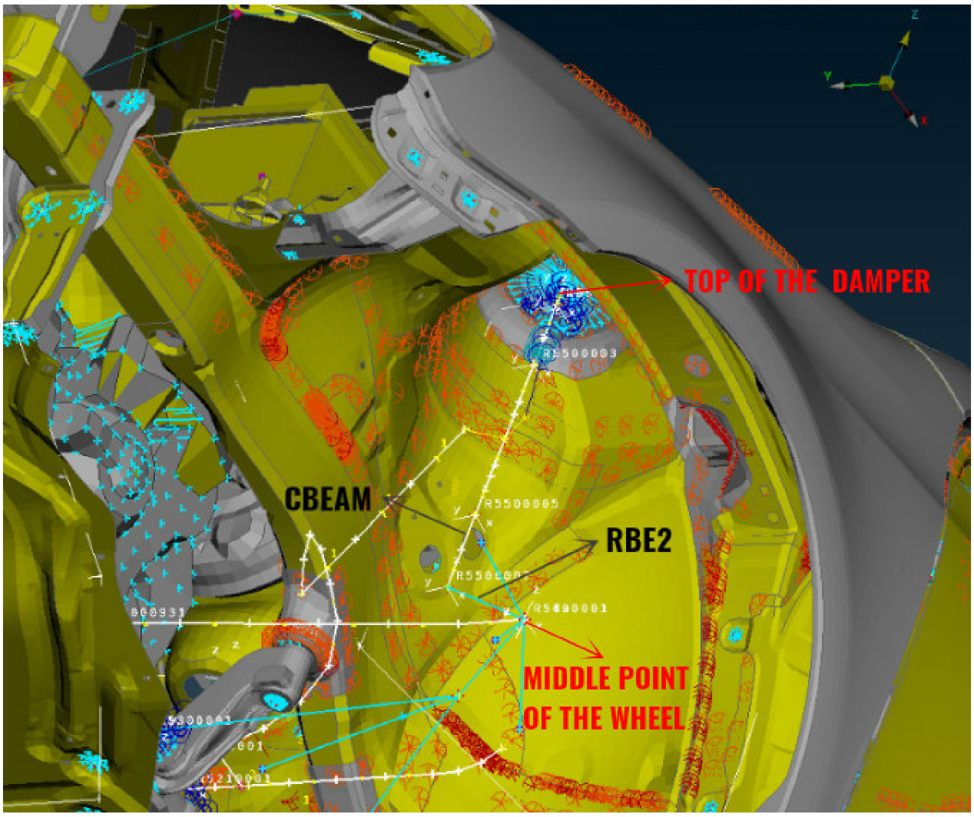

Figure 52: Front Suspension System Coupled to the Chassis

A RBE2 element is connected to the middle point of the wheel. For this reason, the suspension system can absorb the reactions produced by the ground's irregularities. This rigid body is connected with a series of beams joined together one after the other before (CBEAM component).

This is the rigid part of the suspension system, but the model has also the set spring-damper. It is simulated different in the front suspension than in the back one, but this part will be studied later.

On the one hand, the damper is represented as a CDAMP1 element. It is connected to the set of beams and it makes the function to absorb the kinetic energy. As we will see later, we can modify de properties of this component.

So in the Figure 53 we can see the different parts of the front suspension system:

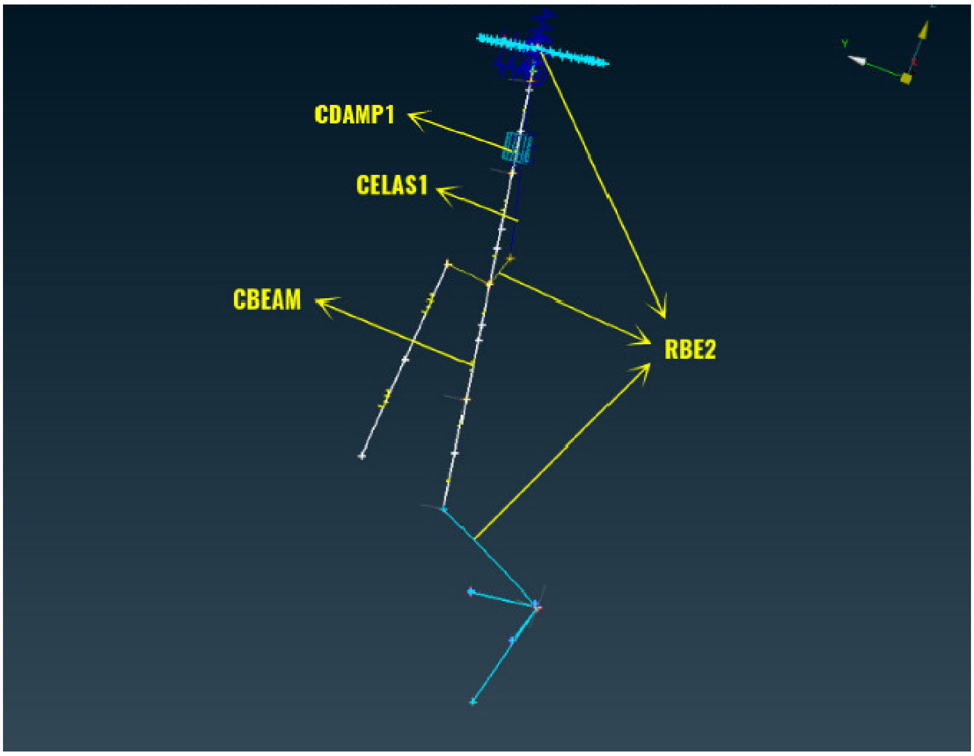

Figure 53: Parts of the Front Suspension System

On the other hand, in one extreme of a beam, there is a RBE2 connected, which join this rigid part with the spring. Thus, the CELAS element, which represents the spring of the car, is connected at its ends with 2 RBE2 rigid bodies. Also, the higher RBE2 connects with other RBE2 which ends in the top of the damper or in the car hood.

It is clearly seen in the Figure 54:

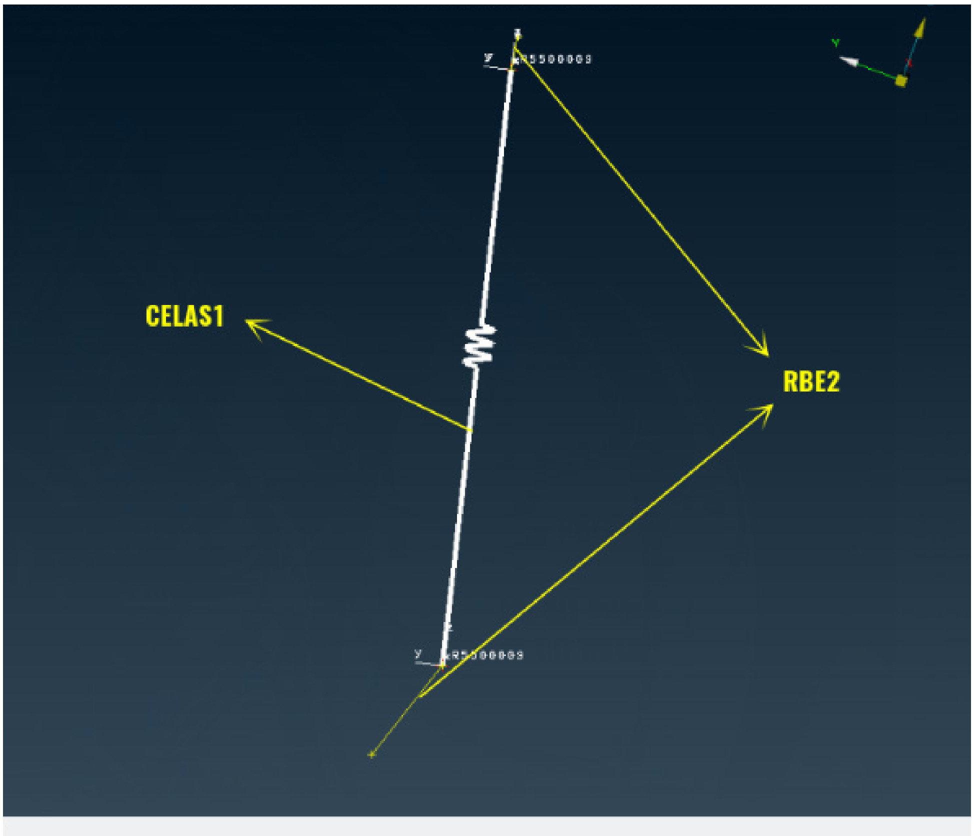

Figure 54: Set RBE2-Spring-RBE2 of the Front Suspension System

8.1.2.3 Back Suspension As we said before, the back suspension is divided in two different parts, the set torsional axis-spring and the damper.

On the one hand, the suspension system consists in a torsional axis which is a solid tube that can be deformed in a certain angle when the wheels find an obstacle, and after that it can return to the original position. With this component, the vehicle is more balanced because this axis is joined with its two corresponding springs. The torsional axis is directly joined to the wheel to be able to absorb the irregularities of the ground.

So, the spring starts in a circular hole that the solid axis has, and it ends in a component joined to the bodywork in the back part of the trunk, as we can appreciate in the Figure

55:

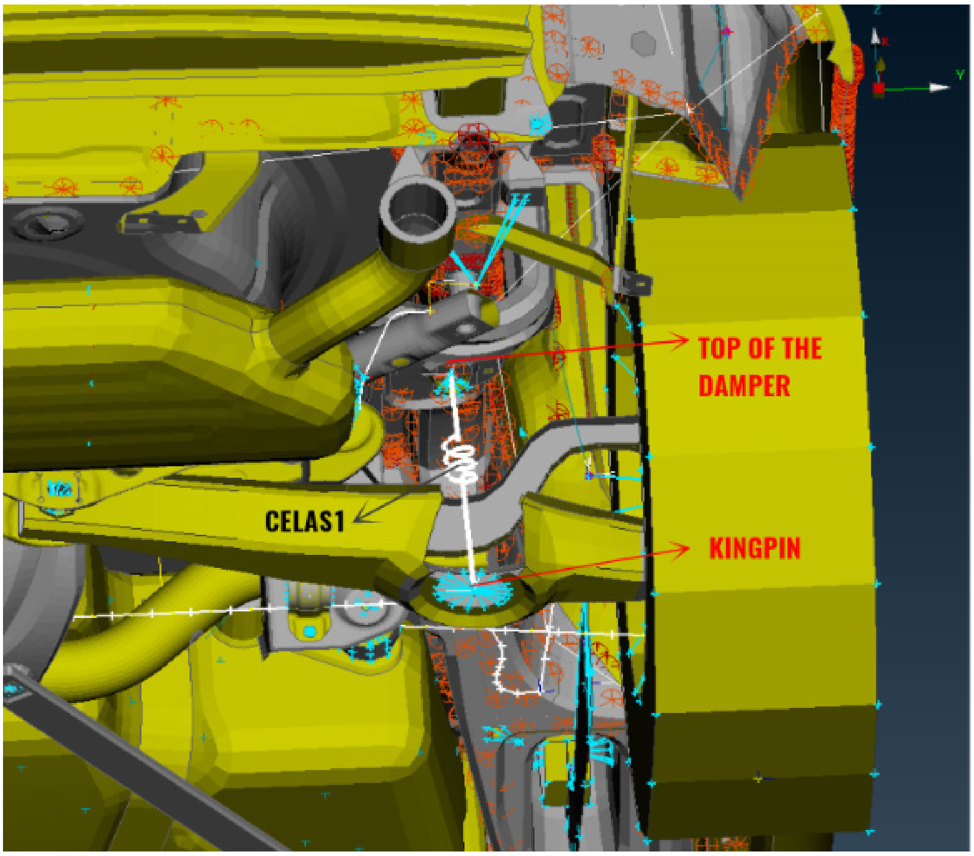

Figure 55: Spring of the Back Suspension System Coupled to the Chassis

As in the front case, we find two set of RBE2 rigid bars that connect the spring with the solid parts. Therefore, there is not any kind of displacement between the points that are connected with those elements, as it is shown in the Figure 56:

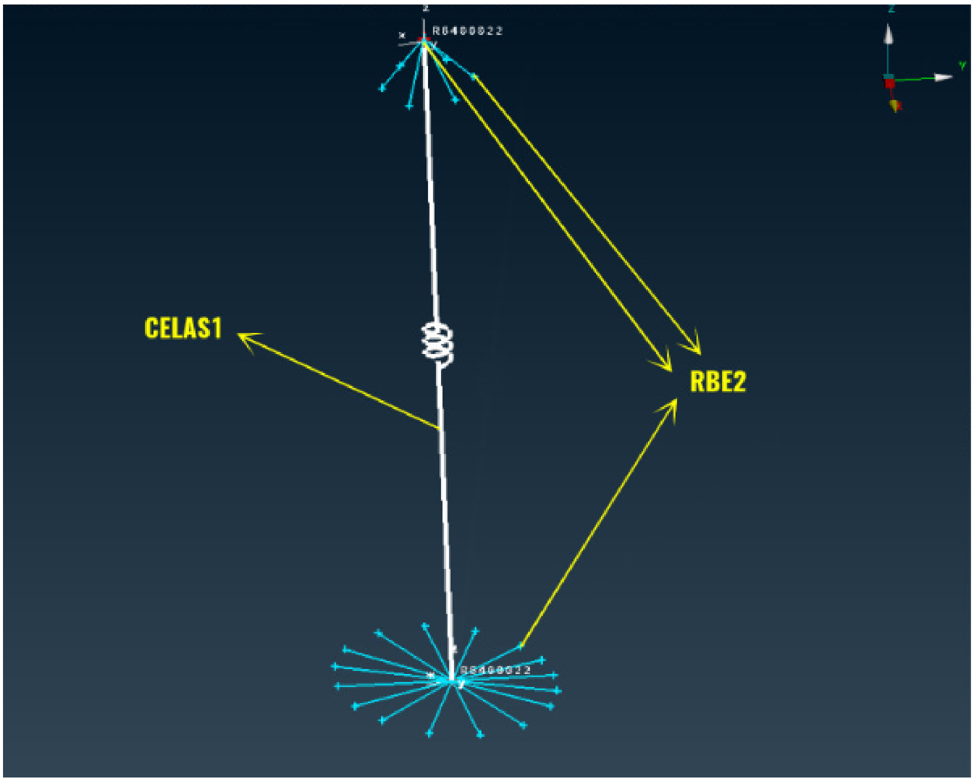

Figure 56: Set RBE2-Spring-RBE2 of the Back Suspension System

On the other hand, the suspension is guaranteed by the damper. In this case, the damper is allocated independently from the spring. With this element in this position, we assure the contact between the wheels and the ground. The reason is because of the lower weight of the torsional axis that gets the parameters of the wheels constant.

We can see in the Figure 57 the place of the damper related in this case with the back suspension:

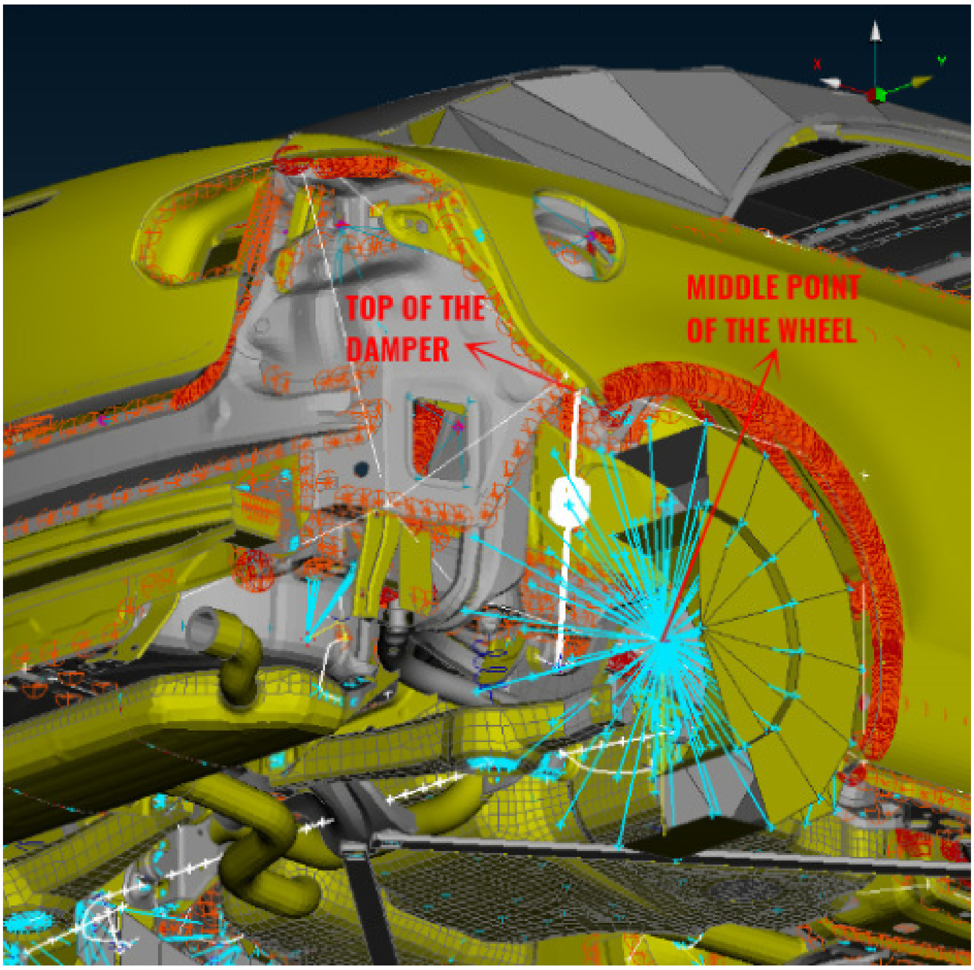

Figure 57: Damper of the Back Suspension System Coupled to the Chassis

Once again, the damper is joined to the wheel and to the bodywork through two rigid bars. As in the front suspension, there is a RBE2 element which is connected with the middle point of the wheel. Similarly, in the top o the suspension there is other RBE2 that joins the damper with the chassis.

Also, the extremes of the rigid bars ends with some springs which simulate the behavior of the suspension system. In that case, we can change the parameters of the spring, damper and these little springs.

We can get a global view of this SET in the Figure 58:

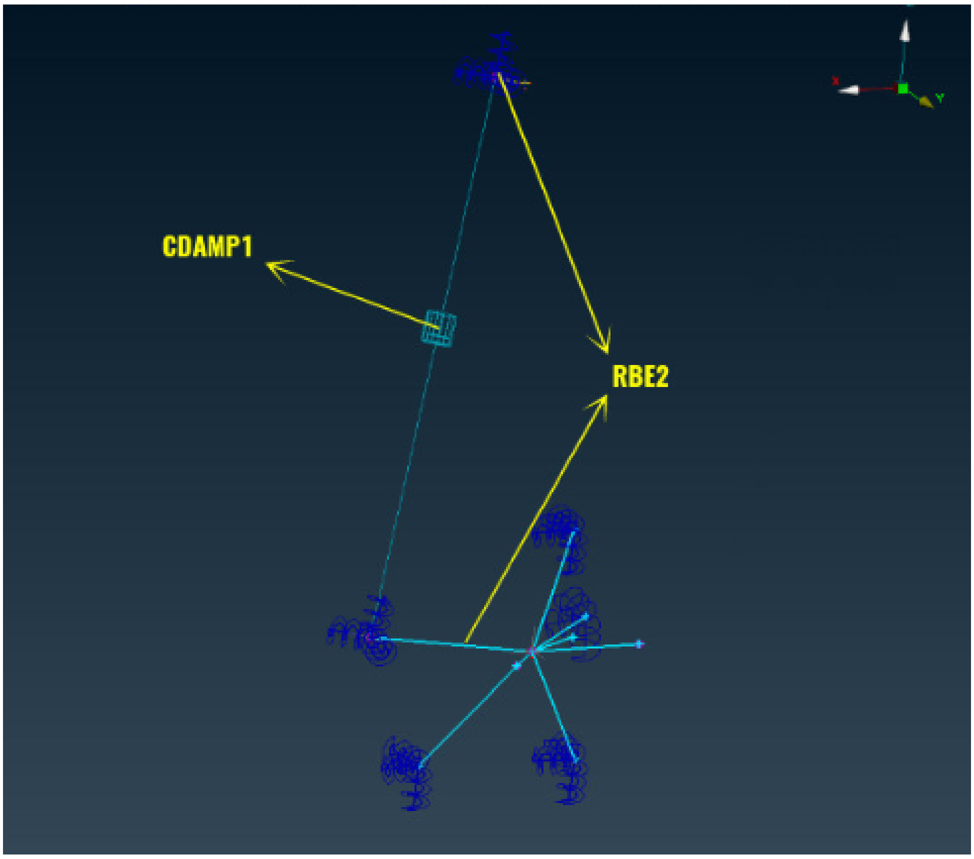

Figure 58: Set-RBE2-Damper-RBE2 of the Back Suspension System

8.1.3 Changing Elastic Properties

After describing the suspension system of our model, we started to find in the include files the possible elements that could affect to the behavior of our car.

8.1.3.1 CELAS Elements The first elements that we looked carefully were the CELAS elements that were in our car. Specifically, they were CELAS1. This command defines a scalar spring element which joins two geometric grid points. In the next Figure 59 we can see how are defined these commands.

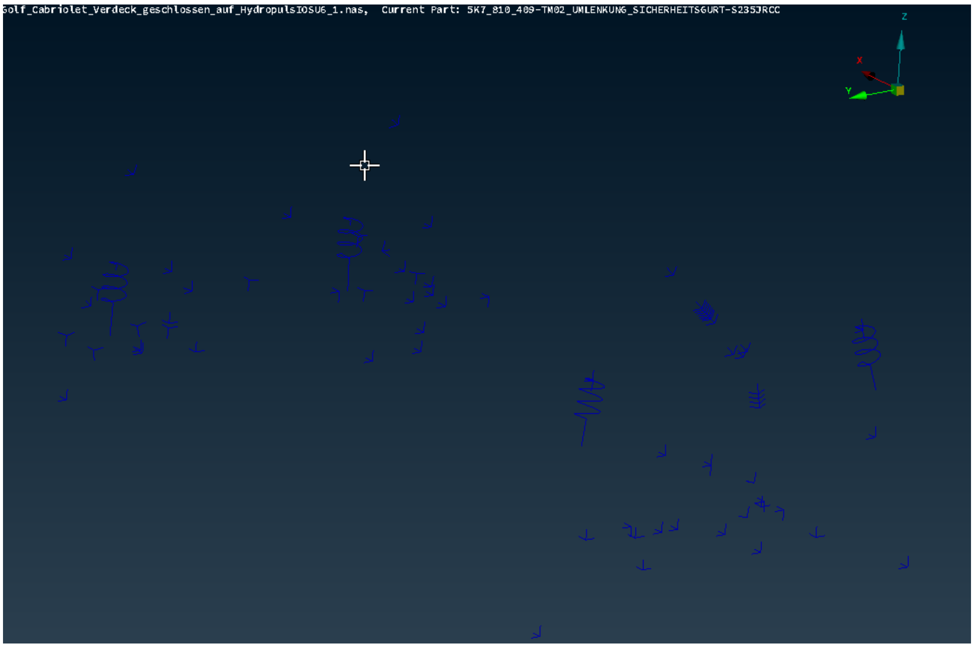

Figure 59: CELAS1 elements definition

In addition, it is associated with a PELAS entry which determines the properties of the spring, as we explain in the Figure 60:

PELAS	PID1	K1	GE1	S1

Field	Contents
PID	Property identification number
K	Elastic property value
GE	Damping coefficient
S	Stress coefficient

Figure 60: PELAS definition

The first step was to identify the elements that represent the suspension in our car body.

As we can see in the Figure 61, there are four springs.

CELAS1	EID	PID	G1	C1	G2	C2

Field	Contents
EID	Element identification number
PID	Property identification number of a PELAS entry
G1,G2	Grid point identification number
C1,C2	Component number

Figure 61: Suspension Elements of Our Car Body

If we pick one of the spring elements, we would see the identification property. So for example, as we can see in the Figure 62, the spring selected is one of the rear ones, related with the identification number of "6000003":

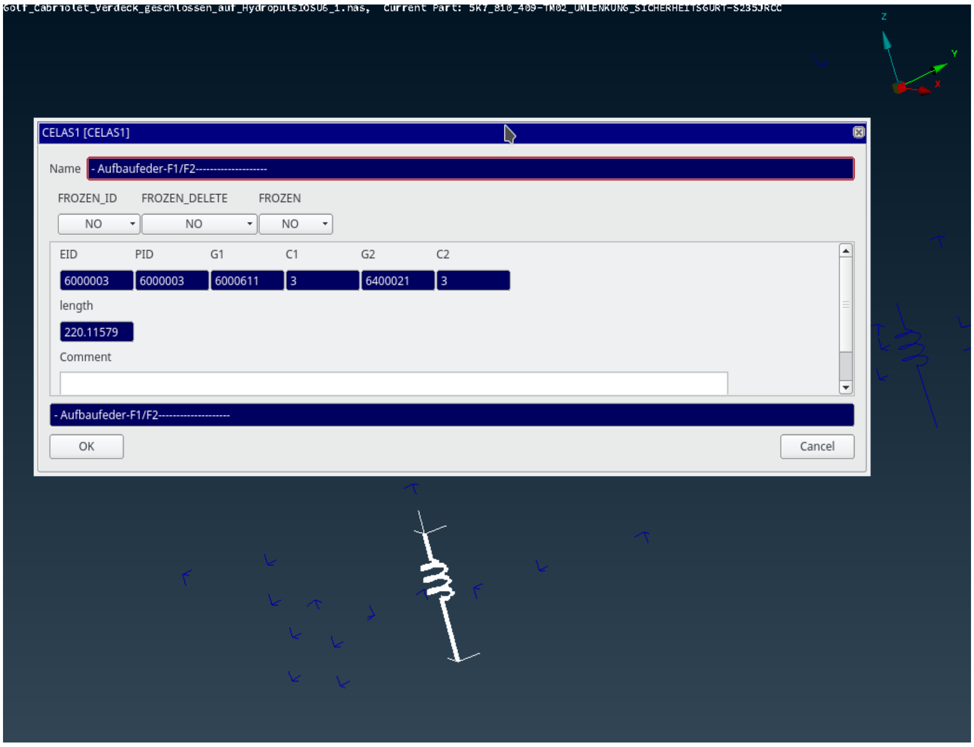

Figure 62: Spring Element Information

This information was very important for us, because since we knew the include file were the spring was described, we were able to identify manually the spring and then change the properties for doing different kind of simulations.

There were two different types of suspension: the lead and rear suspension. Both of them were identified with another two elastic properties. So if we looked in the INCLUDE files that were described in the original NASTRAN file, we found the next information as we can appreciate in the Figure 63:

```
$ --- VORDERACHSE
$
INCLUDE 'Includes/0_VA_0101_ohnevat.inc'
INCLUDE '/daten/iosu/volkswagen/test_012/
Includes/0L_VA_01014Aggwz5_d1_1000.inc'
INCLUDE 'Includes/0_LA_0101_neu.inc'
$
$ --- HINTERACHSE
$
INCLUDE 'Includes/0_HA_0101_ohnehat.inc'
INCLUDE '/daten/iosu/volkswagen/test_012/
Includes/0L_HA_0101qst6cwy_d1_1000.inc'
INCLUDE 'Includes/hatpottvorn.inc'
$
```

Figure 63: Suspension Include Files

From here, we have to open these both data files. If we start looking to the lead suspension, it is related with the property 5000003, as we can see below in the Figure 64:

```
$- Aufbaufeder---------------------------
PELAS    5000003   23.00    0.01
```

Figure 64: Lead Suspension File

The information that we were interested was only that two values. These values match with the stiffness and the damping coefficient.

In the next Figure 65 we can see the rear suspension property:

```
$- Aufbaufeder-F1/F2--------------------
$  enstsprechend Fahrzeugzustandsblatt
PELAS    6000003    24.0    0.01
```

Figure 65: Rear Suspension File

So, the process that we followed was really simple in concept but not in facts. It is clear that the spring elements that were attached to these properties should have had an important effect in the behavior of our car body. As we know, the suspension is the responsible of carrying with the inputs from the road. For that, we changed the properties trying to guess the behavior of the results if we changed both values. That was our first idea, but starting the simulation process that we are going to explain now, we got to the fact that we were changing the inappropriate parameters.

In the initial Nastran model, the results that we obtained in the simulation were not the same as in the real model (this is obvious because the aim of the project is to get a model as close as possible to the real one). For that reason, we started changing the parameters of the suspension system to observe the variations in the results using the post processor. Therefore, we decided to modify the parameters of the 4 main springs. Each one was related with its correspondent wheel, as we have shown before.

The aim was to know how the modification of the parameters affected to the final results. So, we changed the value of the "Elastic property value" and the "Damping coefficient" because in the initial Nastran model there was no mention of the "Stress coefficient". After that, we compared with the real results and we decided if the modification made was better than the initial file.

The first attempt was to reduce significantly the value of the Elastic property value. We reduced the initial value (23 in the front dampers and 24 in the back ones) to 15 in both cases.

However, we had a problem because our program does not have any FATAL messages, the results were similar in both cases (when the Elastic property values were 23 and 24, and when they were 15).

Trying to increase that parameter instead of decreasing and also changing the damping coefficient, we found that the results were again similar. Thus, we were modifying the parameters of the four main springs of the suspension system but the results that the post processor gave us were the same.

In addition, the whole suspension system is simulated with CELAS elements. This is because this kind of components represent efficiently the behavior of the suspension parts. For example, in the bottom of the wheel, there were four springs which pretend to simulate the characteristics of the tire. There, we could change again the parameters of the stiffness and the damping coefficient.

In a new simulation, we tried to change all the values of the stiffness in these springs. The aim was again to approach our results with the real ones. However, the difference that we obtained making this substantial change was quite small. For that reason, we decided to let these values as in the first model.

The first results were not acceptable because we always obtained the same results from different parameters. So that was really annoying and it took us a long time to solve these problems.

8.1.3.2 CBUSH Elements Another change that we tried was to change all the CELAS1 elements that we had modified before, for CBUSH elements. The reason to do that was simply to compare the results and the influence of these elements. Moreover we would like to see if that elements were more similar to the real car, and maybe the diagrams were more similar than using CELAS1.

The CBUSH is a spring and damper structural scalar element. It has many advantages of using the CBUSH element over CELAS elements, for instance, if we use CELAS elements and the geometry is not aligned properly, internal constraints may be induced. The CBUSH element contains all the features of the CELAS elements plus it avoids the internal constraint problem. In addition, it is associated with a PELAS entry which determines the properties of the spring. In the next tables we can see how are defined these commands.

The definition of both CBUSH and PBUSH is explained in the Figure 66:

CBUSH	EID	PID	GA	GB

Field	Contents
EID	Element identification number
PID	Property identification number of a CBUSH entry
GA,GB	Grid point identification number

PBUSH	PID	K	B	GE	RCV	M

Field	Contents
PID	Property identification number
K	Stiffness values in 6 fields
B	Force per velocity damping in 6 fields
GE	Damping constrants in 6 fields
RCV	Stress or strain coefficients in 4 fields
M	Lumped mass of the CBUSH

Figure 66: CBUSH and PBUSH definition

We solved the new model with the same values for the parameters of the spring. Thus, we just introduced the values for the stiffness and the damping constant. We wanted to see what was to change with the new results.

Obviously, the output was not the same that in the initial model. Especially in the flexion subcase because the accelerations in each axis were not in the right order. It was easy to know if the model was on the right way to get good results, because we could compare in the graphics if the curves were similar and with their respective axis. In the flexion subcase, the three curves of each axis were not in the same position than in the CELAS's model. In the last one, the acceleration in the Z-direction was the highest, like in the real output . This aspect was not like this in the CBUSH's model.

Although the results were worse, we tried to modify the parameters of the CBUSH element to see if we could approach to the real results. Once again, we reduced the initial value (23 in the front dampers and 24 in the back ones) to 15 in both cases.

However, we had the same problem because the results were similar in both cases (when the Stiffness is 23 and 24, and when it is 15).

So we decide that the CELAS1 spring is not the trouble in our model because we had had the same problem with the CBUSH element. Therefore, now we decided to use the initial model with CELAS1 command because it was easier to use (it has less parameters) and also because the output in our model was closer to the real results.

8.1.3.3 CDAMP-PDAMP Elements After all of these last attempts, we realized that we had another two another elements that we had not modified. These two elements were the dampers and the wheels.

We would like to start talking about the definition of the dampers and its properties, so we are helped by the Figure 67:

CDAMP1	EID	PID	G1

Field	Contents
EID	Element identification number
PID	Property identification number of a PDAMP entry
G1	Grid point identification number

PDAMP	PID1	B1

Field	Contents
PID1	Property identification number
B1	Force per unit velocity

Figure 67: CDAMP and PDAMP definition

To place better the dampers that we are talking about, we would like to show you in the

Figure 68:

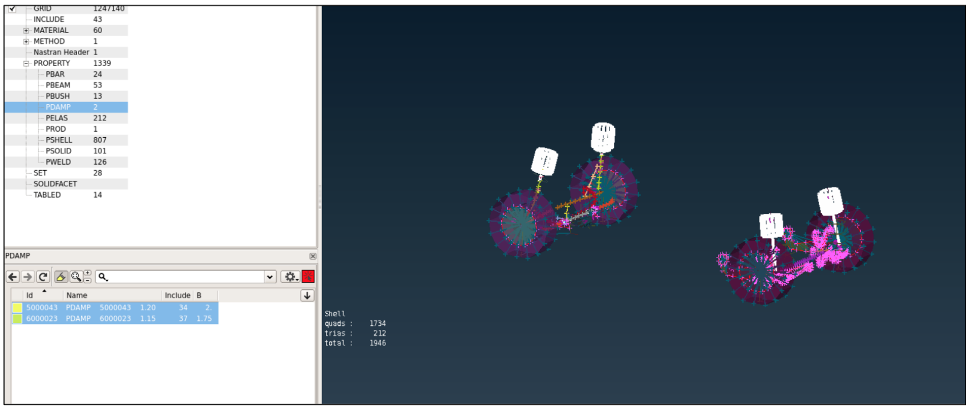

Figure 68: Dampers in our FEM Model

As we can see before, the dampers are different in the front suspension than in the back one. It?s because in the back suspension the damper is independent of the set spring-torsional axis.

In this case, the damper has just one property, the Force per unit velocity. Its units are Newton meter divided by square second, that indicates the quantity of energy that the damper can absorb in a determinate time.

Again, we tried to change the parameters of the suspension system to observe which were the variations in the results of the post processor. Therefore, we decided to modify the parameters of the four main dampers. This time we hope to get different results than in the initial model, thus, without change any parameter.

Then, the first attempt was to reduce significantly the value of the Force per unit velocity. We reduced the initial values of the dampers (2,00 in the front dampers and 1,75 in the back ones) to 1,20 and 1,15 respectively.

In this case, the output results were different than in the initial model. We could notice that the shape of the curves were similar than before, but we had changed the amplitude of the output signal. For example, the highest peak of the acceleration in the A-pillar was 17490 and now it is 11620 (mm/s^2). Therefore, we could modify the force per unit velocity to get the same amplitude than the curves of the real model.

Trying to reduce just the parameters of the front dampers, that they are more important because they are part of the traction and direction system of the car, we decrease also the amplitude of the output.

So we decided to use this parameter, the Force per unit velocity of the CDAMP1 elements, to bring closer the amplitude of the model curves to the real ones.

For that reason, the first mode shapes that we got were not correct. The problem was that we set up the simulation with the values of the stiffness and damping of the suspension system of our car without any knowledge of the real ones. For that reason, we decided that first of all, we should have found the correct value of the stiffness and damping for our car body, and then we would be able to obtain the good mode shapes.

8.2 Study of the Wheels

As we said before, the wheels were composed with a lot of RBE2 elements which joined all the points of the extremes with the central point (which was the independent point). Thus, there was not relative displacement between the different points of the wheel.

However, it was really important the movement of the point in the bottom of the wheels. This was the point which in NASTRAN program was assigned to be the responsible of the movement of the whole car. The reason was that the input signal was in these points. Therefore, this input signal was induced just in one point of each wheel. The point was the bottom point of the middle of the wheel. It means that in the NASTRAN program, just the middle point of the width of the wheel was touching the floor and had an input signal. Also, for the µETA, the output of the displacement in those points was zero because the SPC constraints were defined in them.

The middle point of the wheel was connected with the bottom one through a RBE2 element, as we can see in the Figure 69. Therefore, the movement produced in the wheels passing over the ground irregularities was transmitted directly to the middle point of the wheel. This last point is connected with the whole wheel and also with the suspension system as we said before.

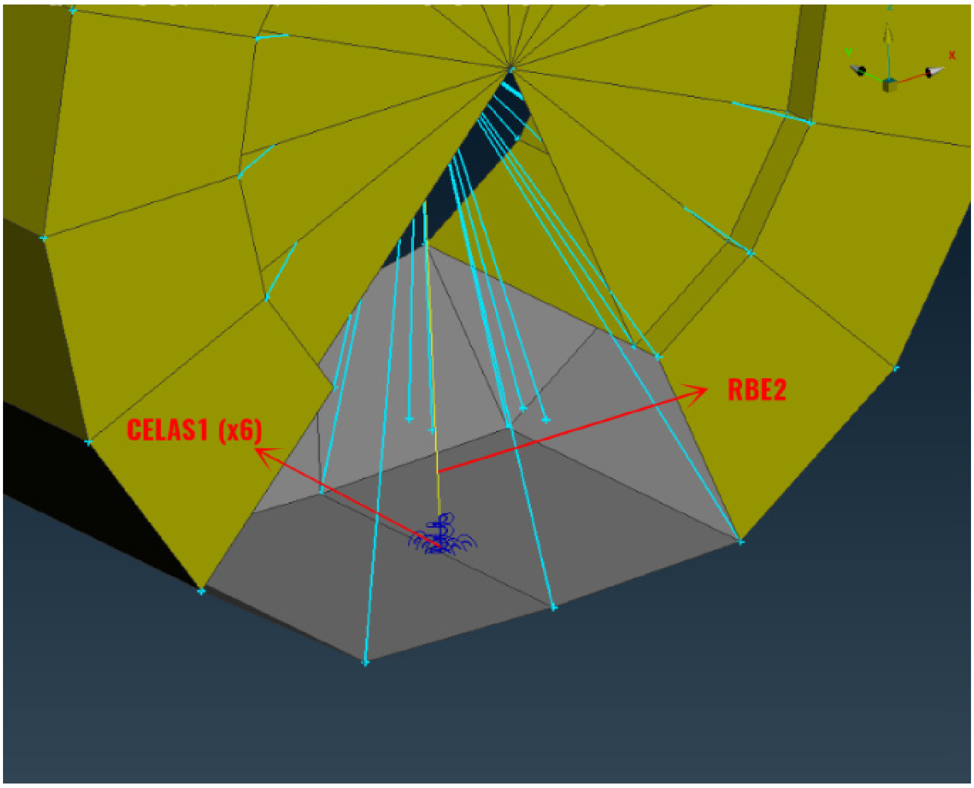

Figure 69: Wheels Parts

Although in the real car there were more than one point which connect with the ground, in our model this point had to emulate the behavior of the whole wheel. For that reason, although the ANSA file seemed that there was just one point in the bottom of the wheel, there were two. Thus, the point where the input signal was produced (for example, as we can see in the point 5000003 in the front left wheel of the Figure 70) was connected with a set of six spring elements. These CELAS1 components ended in other point (in this case the grid number 5720003) which was connected with the RBE2 which started in the middle point of the wheel.

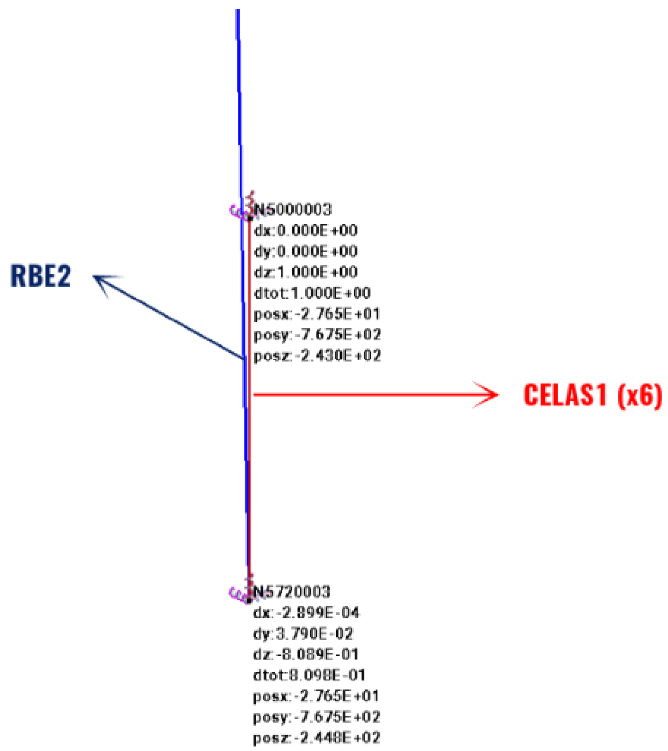

Figure 70: Wheels 2points

Studying this part of the wheel deeply, we could observe that the movement of the middle point of the wheels was related with the movement of that set of springs. As we said in the chapter 1.5.4.2.2 Frequency Response Analysis, there was a phase shift between the input signal in the bottom point of the wheels and the ended point of the springs set. Thus, the peaks did not occur at the same time. For example, when the input signal was in its highest moment, the ended point of the springs (that was directly related with the middle point of the wheel) was nearly to the lowest zone. This phase shift depended of the frequency that were induced.

Also, the amplitude of the ended point of the springs was higher than the amplitude of the input signal (1mm). So, the movement transmitted to the middle point of the car had also an amplitude higher than the input.

This characteristics were constant for the same nodes in the simulation at each frequency. Both the phase shift and the amplitude were a function of the damping system. Thus, ideally the output of all the points of the car would be the same without damping.

Moreover, the time between each peak in the ended point of the springs set was the same as the input signal. So, the period of the response was the same in both cases.

In this occasion, we are not going to study the behavior of the car when we changed the parameters of the wheel. We will study this in the chapter ?Trying to Approach to the Results? how this change affects to our final results.

9 Engine Test

Everybody knows the importance and the influence of the engine in each car. It is not only related with the power, is also related with the vibrations that the engine usually does in a normal state of working. Moreover, the engine could be a source of non desirable vibrations that can have a bad influential effect in the comfort of our car.

So for, as our German partner had done with his own research, we had to try to do the same tests with the engine. The main problem for us was basically that in our model, the engine was only represented by "RBE2" elements, that in a colloquial way, we could say that it was represented by springs.

The springs are a tool really used with FEM models, because they are more or less easy to set up and they represent accurately the function that are supposed to play. However, for us it was a problem. As Mr. Krampe did his tests modifying the real engine removing or changing some specified parts, we had to try to implant another boundary conditions that allowed us to get the same results.

So first of all, we would like to place the original changes made in the real engine by Mr. Krampe. Our car body has three mounts. However, Mr. Krampe had only done two changes in the mounts, that were on the right and left side of the engine. As we can see in the Figure 72, the change made on the right side is simply a modifying between two components:

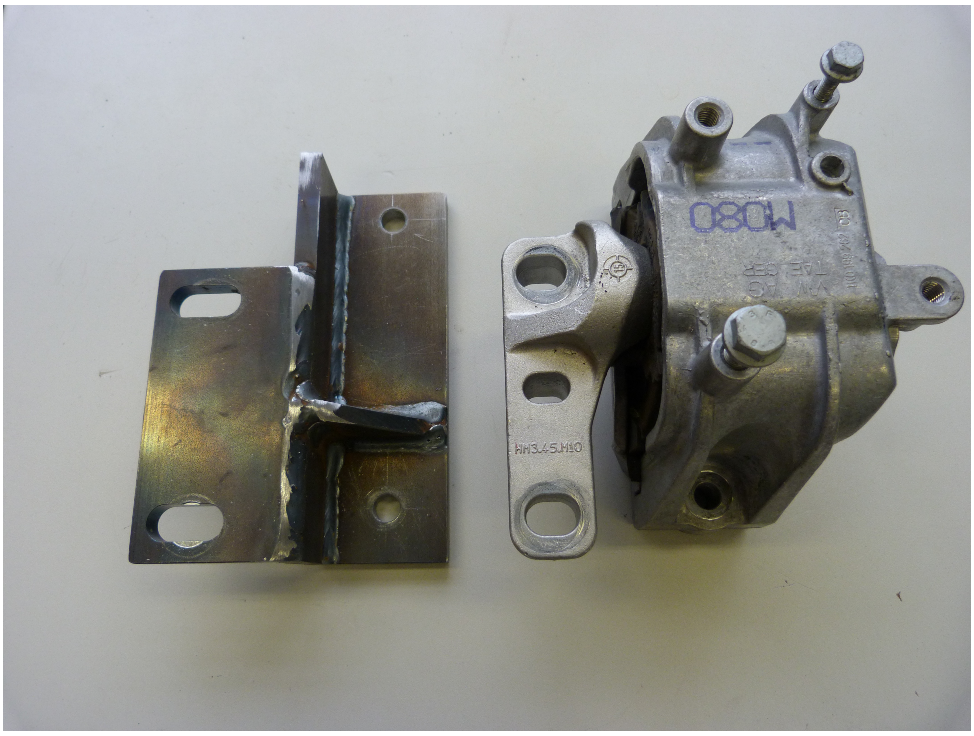

Figure 71: Changes made on the Right Side in the Real Engine

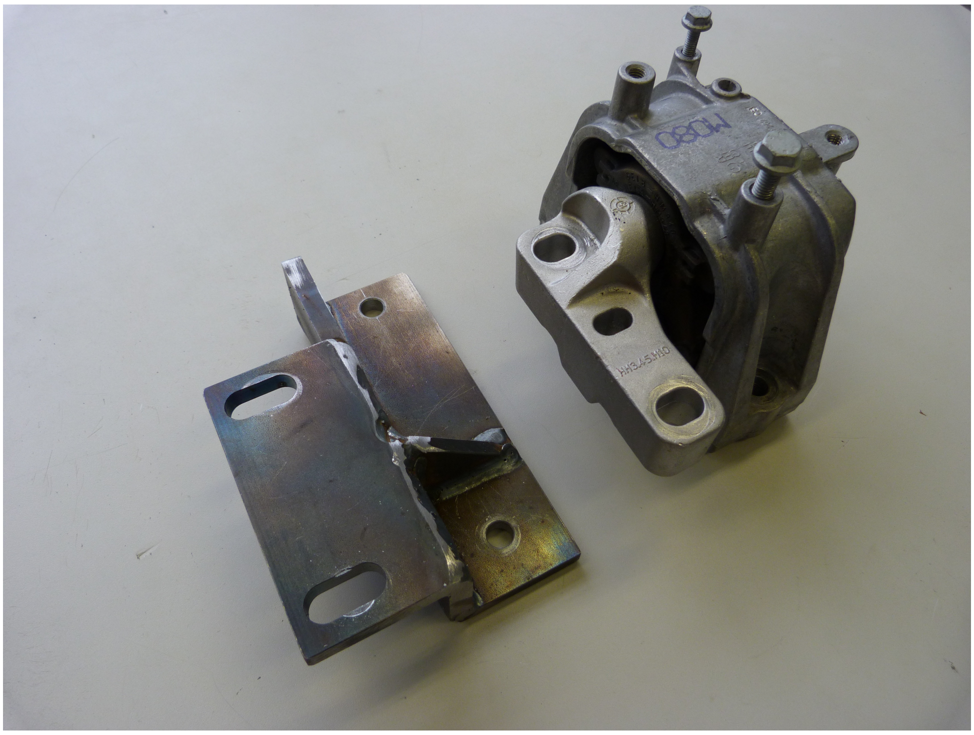

Figure 72: Changes made on the Right Side in the Real Engine

The original part is the right one, and the piece that is going to lock the engine is the located in the left view. To clarify better the location of the lock part in the real engine, we can have a better look in the Figure 73:

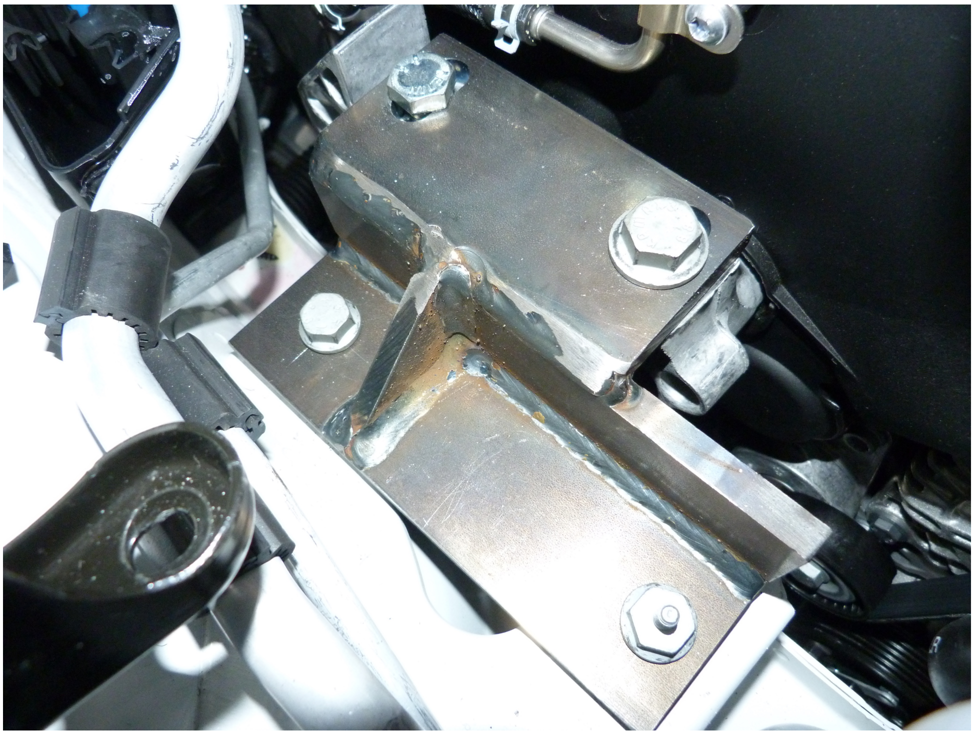

Figure 73: Part Locking the Real Engine on the Right Side

For the other side, we are going to change another component from the real car, just because are not the same parts, so for that reason, we will need another different piece, as we can see in the Figure 74:

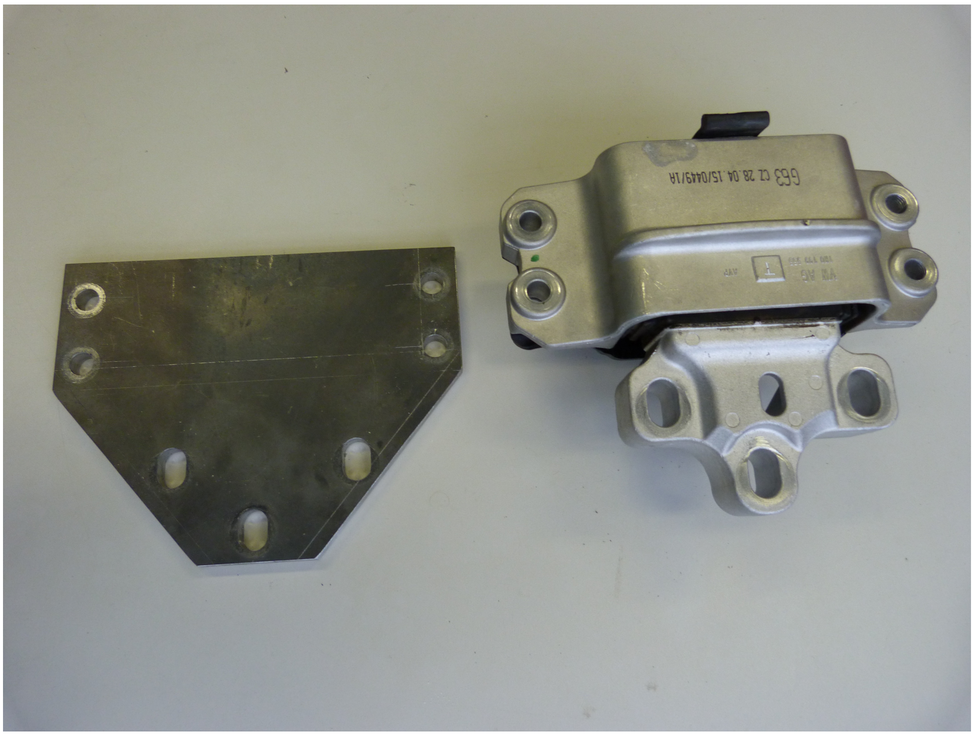

Figure 74: Change made on the Left Side in the Real Engine

So for, now we are going to try to approach better to our engine and try to find a way to lock the engine in the most real way possible using our FEM model.

In the next Figure 81 we are going to see the process that we follow to get till the engine, using the "NOT" command form ANSA:

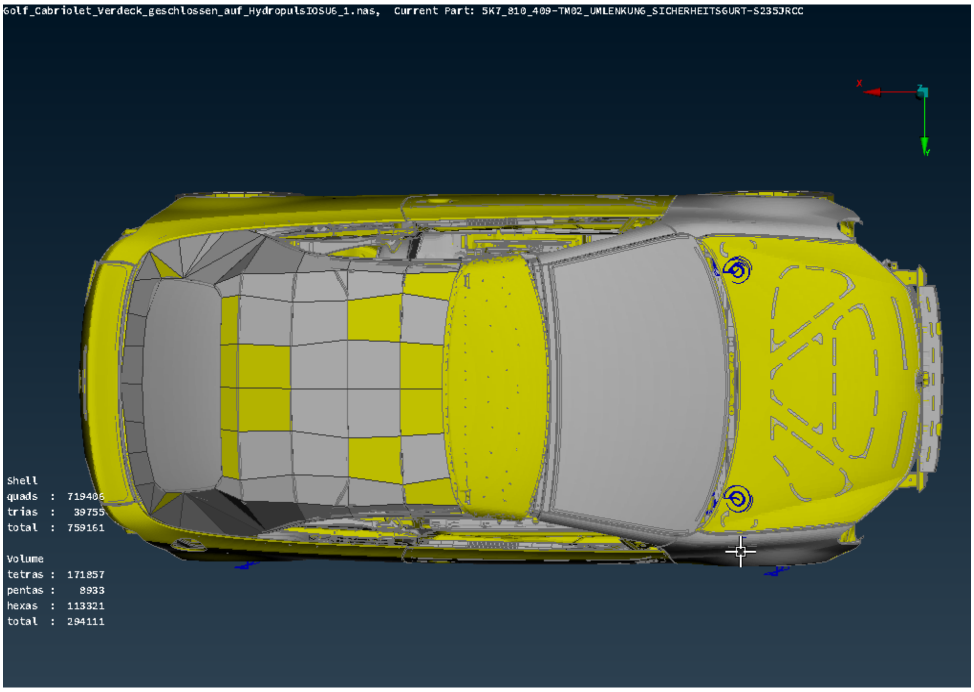

Figure 75: Working with the Engine

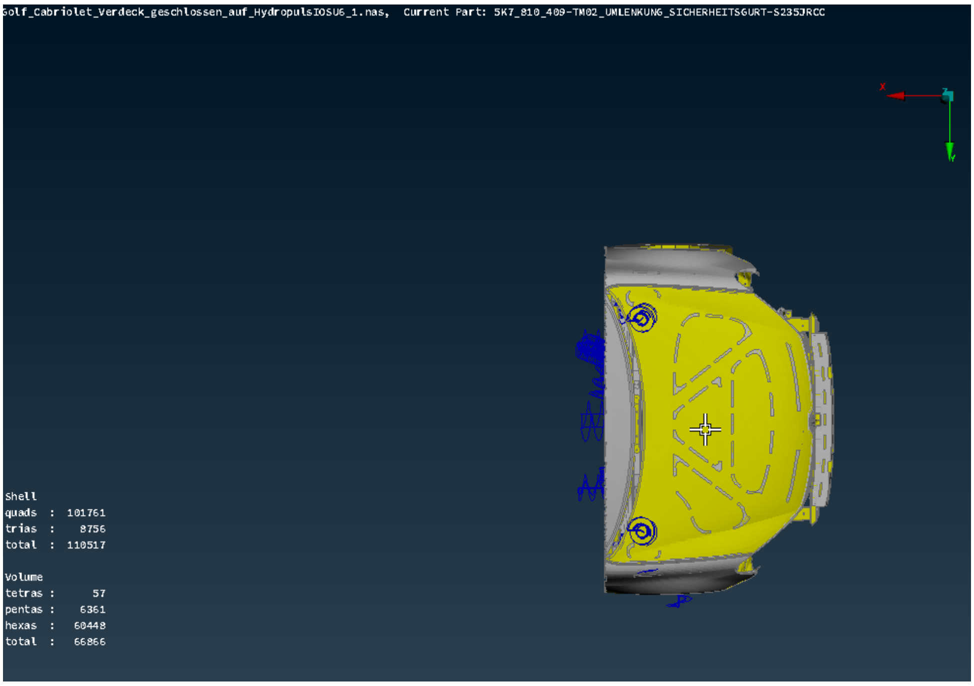

Figure 76: Working with the Engine

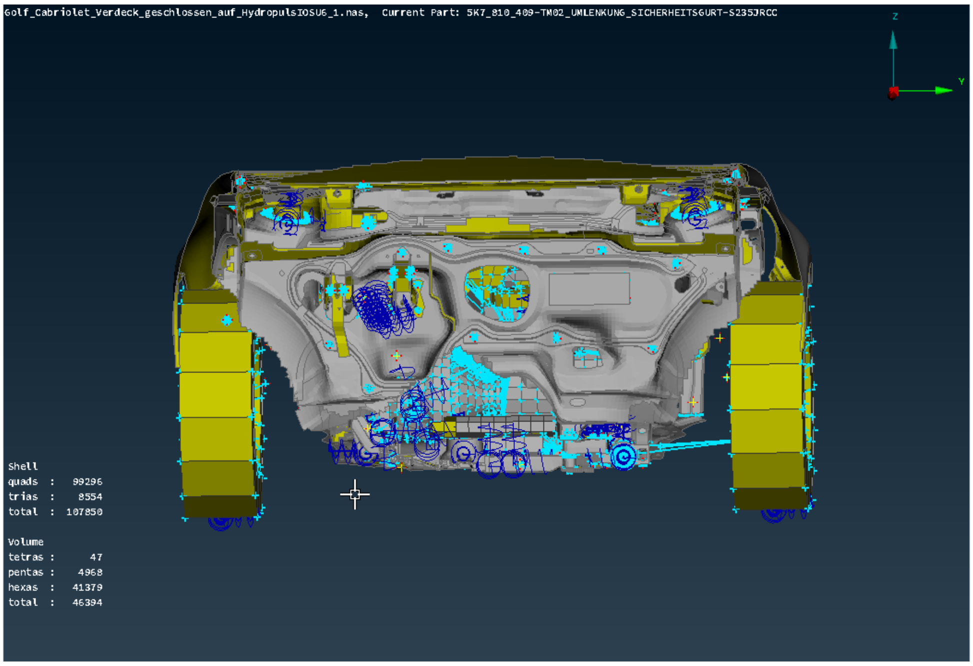

Figure 77: Working with the Engine

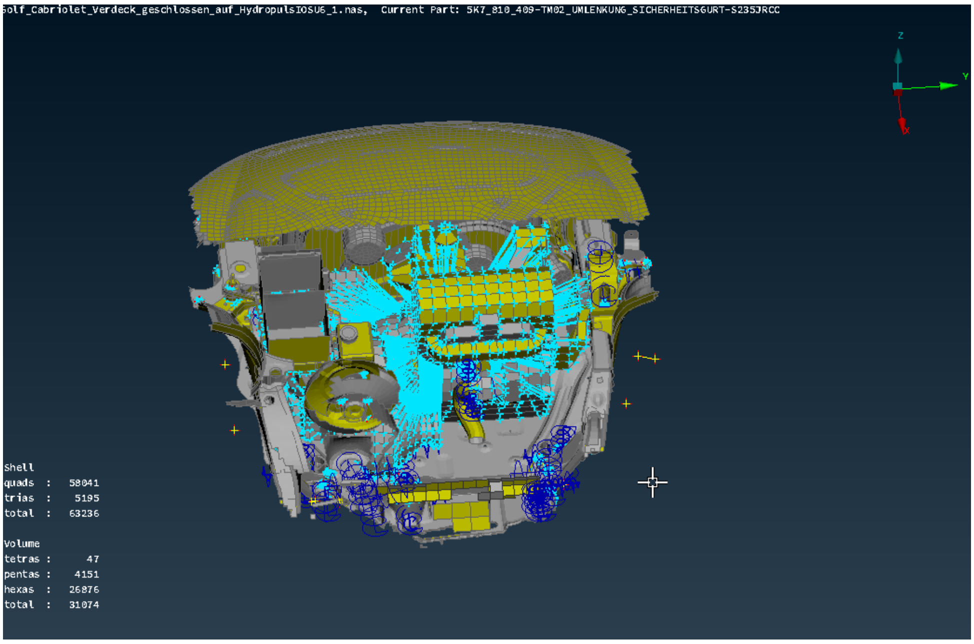

Figure 78: Working with the Engine

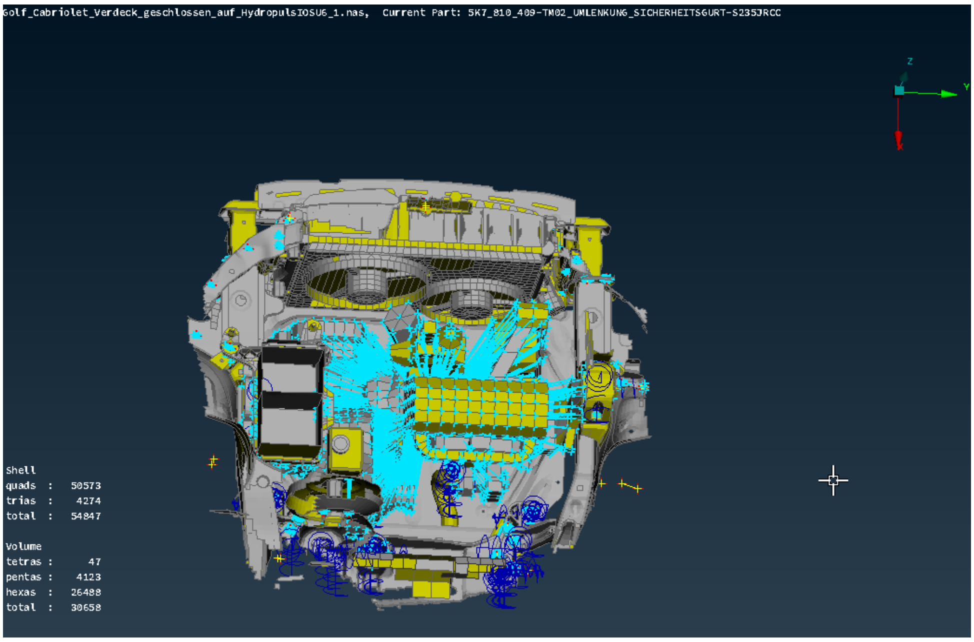

Figure 79: Working with the Engine

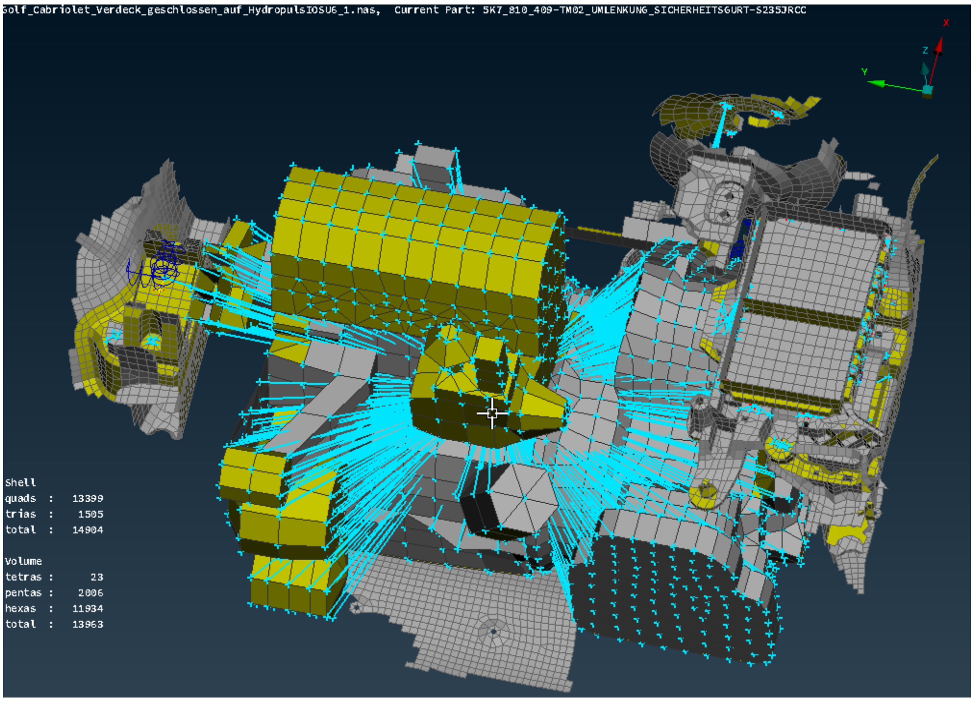

Figure 80: Working with the Engine

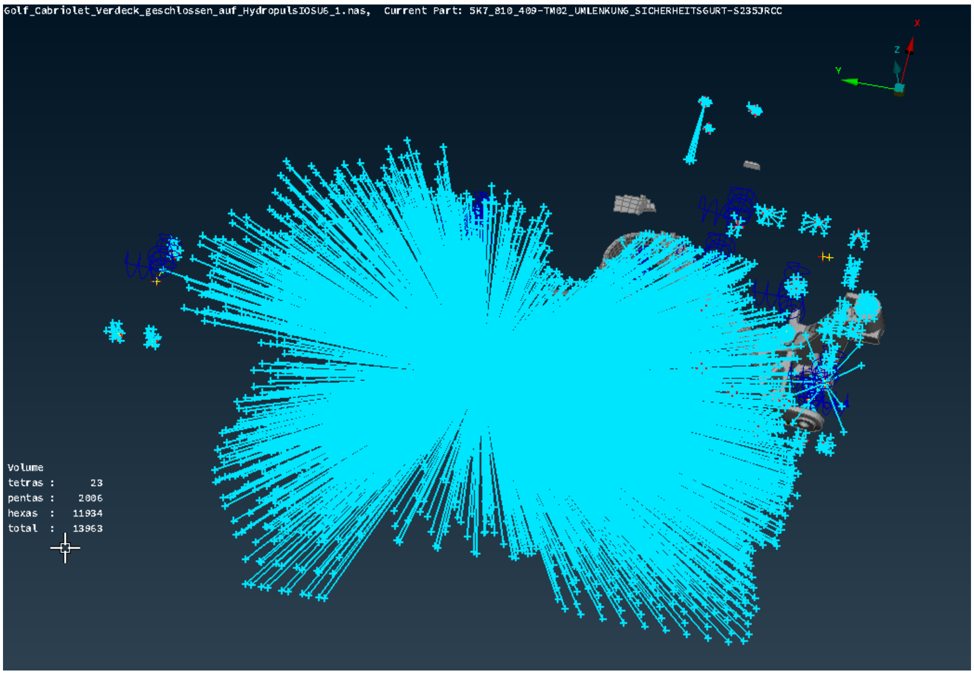

Figure 81: Working with the Engine

Along these pictures, we have seen all the elements that we needed to remove to get till the engine. It was very complicated because we have to be careful of not removing a shell element that was close to the engine, because it was so important to have all the extreme points that the spring were joined to.

To clarify better the information about the engine and the possible changes that we could try, we selected the engine and as it appears in the Figure 82:

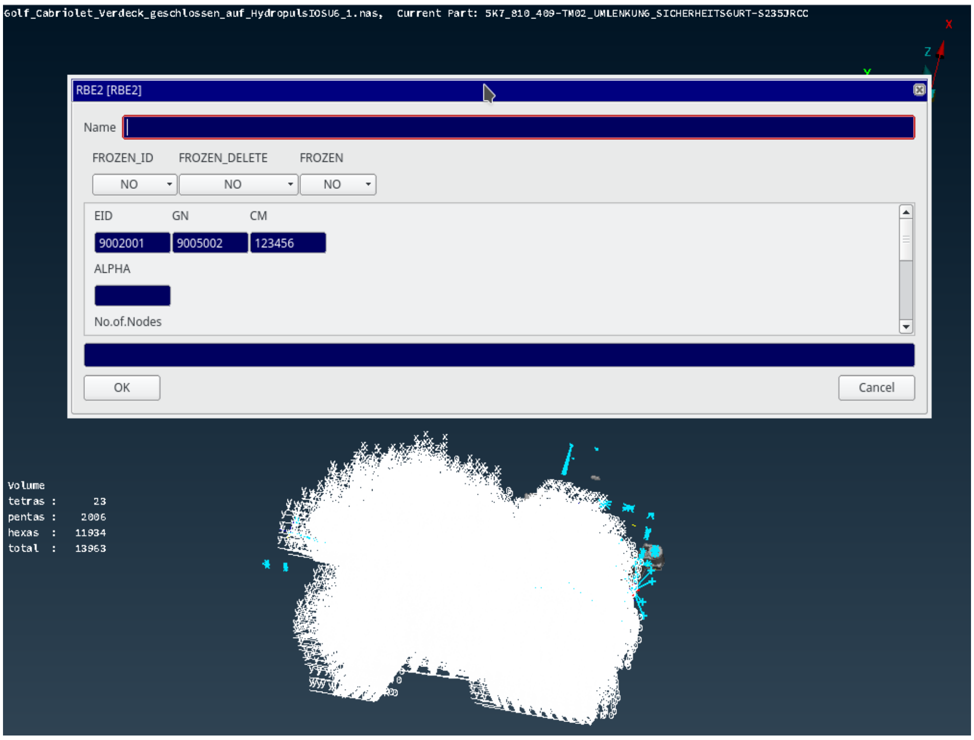

Figure 82: Engine Properties Information

Another possibility of getting a better approach to the engine is using the "INCLUDE" files. ANSA allows us to show only the include file that we are looking for. Moreover, using the "NEIGHB" command, we are able to look in the coupled parts of the include file that we considering in that moment. Moreover, we can lock the current view to work in another moment.

9.1 Locking the Engine in our FEM Model

So using the "INCLUDE" file and "NEIGHB" command, we are able to work better with the engine. As we told before, our car has three different mounts, but we only would like to change two of them. As we are not able to modify in the same way as Mr. Krampe, our changes were to remove the springs that were coupled the engine with the car body. And then, we pasted the "RBE2" element directly with the grid point that was joined with the car.

To get a better idea of the process that we followed, in the Figure 83 we could appreciate the change finally made to lock the engine:

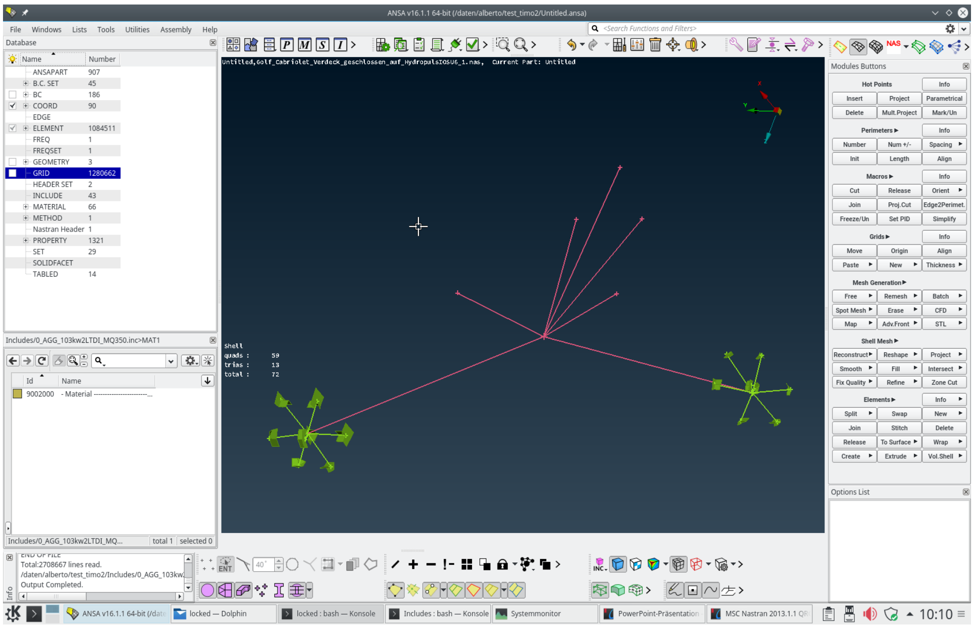

Figure 83: Engine totally Locked

In this picture we can see the two mounts of the car that we are talking about. It is remarkable that the behavior of such an important part as the engine, is only represented by that elements with FEM. So, using the command "NEIGHB" we would like to show the global situation of the engine in our car body, as we can see in the Figure 87:

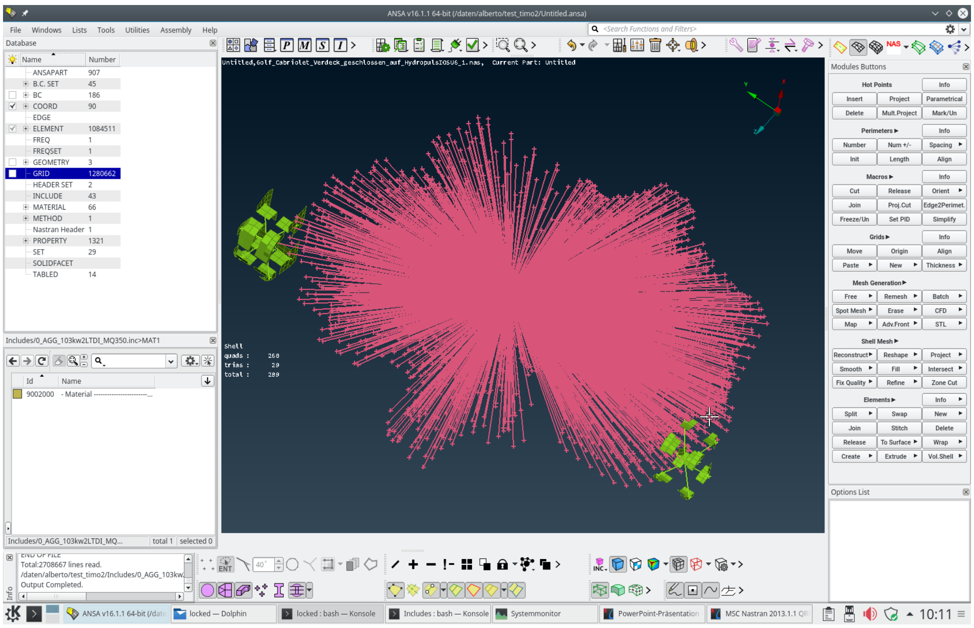

Figure 84: "NEIGHB" command used with the Engine

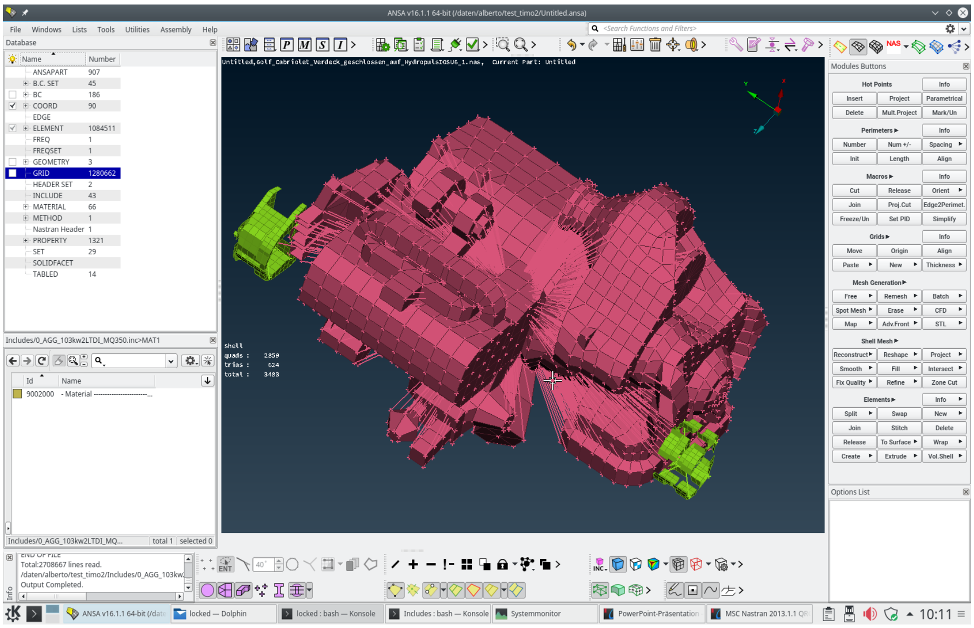

Figure 85: "NEIGHB" command used with the Engine

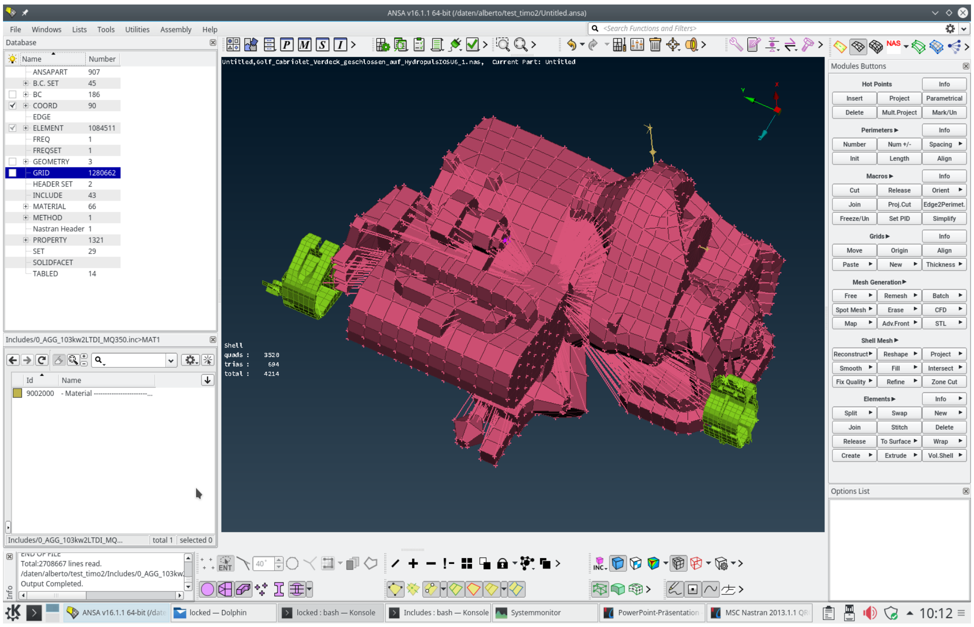

Figure 86: "NEIGHB" command used with the Engine

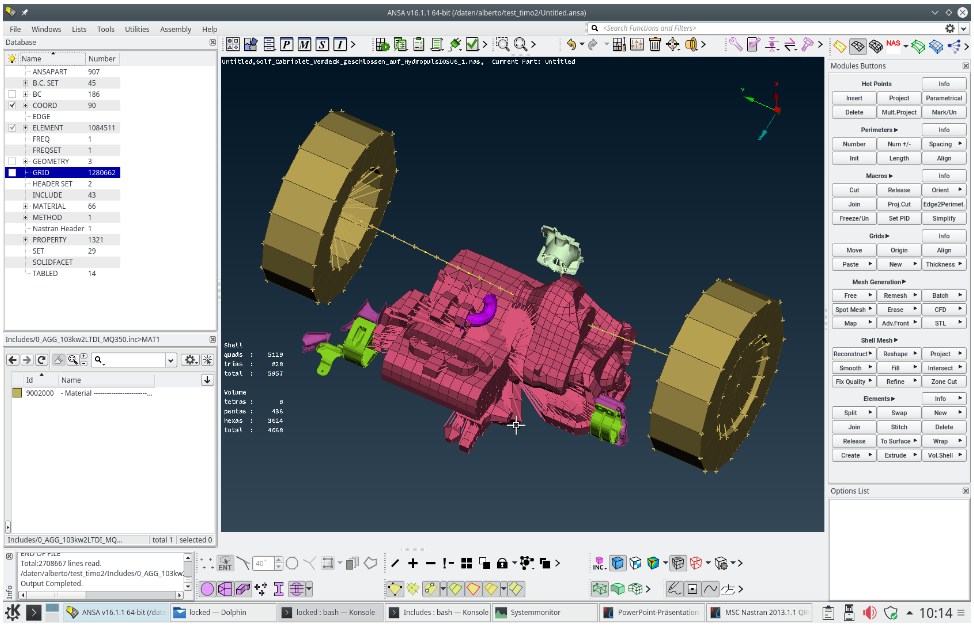

Figure 87: "NEIGHB" command used with the Engine

In the last picture of these ones, we can see more or less the main structure of the engine and how it is coupled with the wheels.

9.2 Comparison of the Results with the Engine Locked and Unlocked

To observe the difference between the car with and without the engine we studied again the output curves on the A-Pillar point and the middle point of the wheel. It is important to study this two points because we want to approach our results to the real ones and we need to get the same curves in all the points of the car. As we said before, if we contrast these two point, we would make a good comparison. Thus, we got the curves by an accelerometer located in the A-Pillar point (the grid number 27499 in our ANSA file) and in the middle point of the wheel. This last point will be in the front left wheel and in the back right one, as our colleague put the accelerometers in the tests of the real car.

Then, we compared the results of the real car with the results of the initial model in these two points. So, each graph had two different kind of curves. The first ones were the curves corresponding to the car with the engine unlocked and the second ones were with that component locked.

In addition, again and as all the tests performed, the amplitude of the input signal was

1mm. The frequency was changing over time between 10 and 30Hz. Besides, the output of our graphs was the acceleration in the point studied, corresponding to the transfer function that we explained in a previous chapter.

In this case, we wanted to differentiate the behavior of the car when the engine was locked and unlocked. Then, we did not differentiate between the torsion case and the flexion one (they were in different graphs but only because their curves were completely different). It means that we want to know the influence of the roof, independently if it is flexion or torsion case. studied first the graphs of the real car to observe the similarities and the differences.

First, we studied the graphs of the A-Pillar point to observe the similarities and the differences between the two kind of curves. After that, we observed the results in the middle point of the wheel to compare again what the repercussion was when we locked the engine. Then, we could decide if the changes were the same in both cases.

9.2.1 Difference between the Engine Locked and Unlocked in the A Pillar Point

As we can see, the Figure **??** and the Figure 89 had the same characteristics features.

First of all, in both graphs, when the engine was locked, the Y axis increased their amplitude considerably, the X axis increased this parameter a little bit, and contrary the Z axis decreased it slightly. For example, in the real case, the higher peak of the Y axis was 23(m/s^2) when the engine was unlocked and this parameter increased to almost 29 when we locked the engine. In the same way, the higher peak of the Z axis in the flexion case was 7,25(m/s^2) with the engine unlocked and the peak was reduced to 6,6 when we locked the motor.

Also, we observed that the curves with the engine locked were displaced to the left in both graphs. Therefore, the highest peaks were in smaller frequencies than when that component was free. In the real case the natural frequency of the peaks with the engine of the car unlocked was around 1,5Hz higher than with this element unlocked. However, in the flexion case the phase shift depended of the axes. In the X and Z axes the difference between the peaks was 1Hz and in the Y axis the difference was 1,5Hz as in the torsion case.

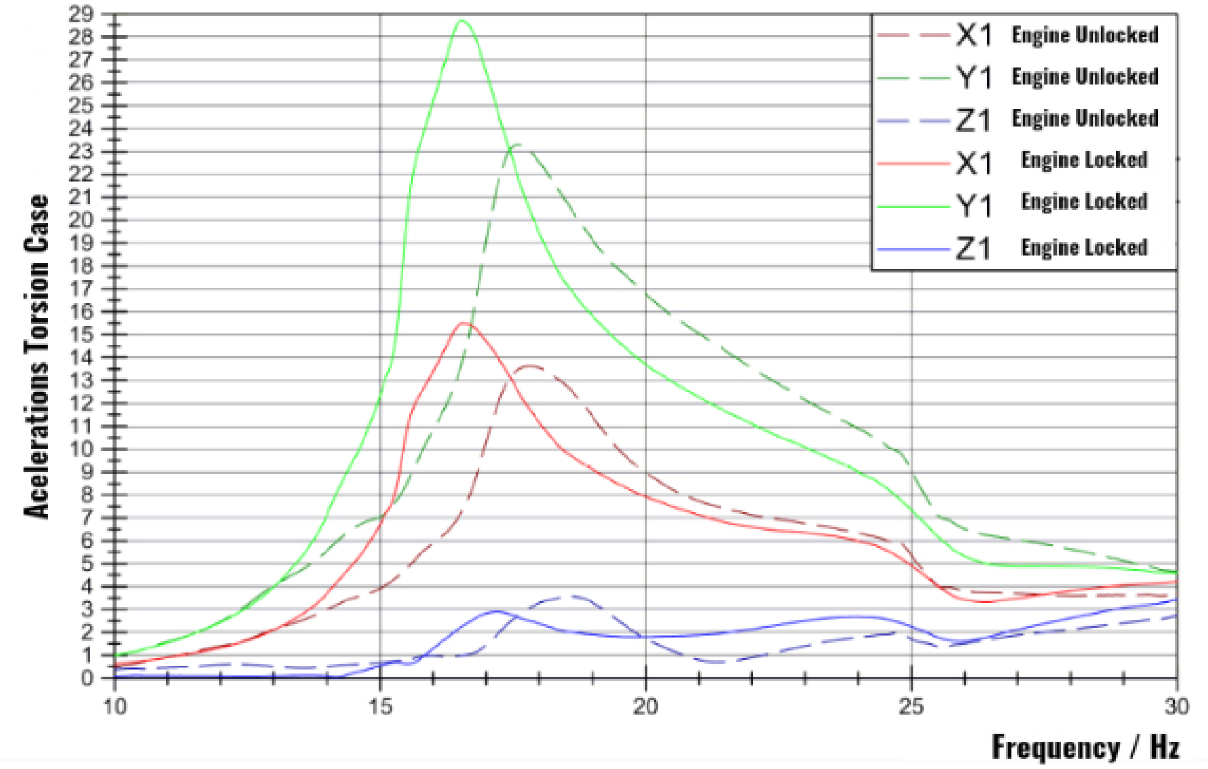

Figure 88: Torsion Engine Locked-Unlocked A-Pillar

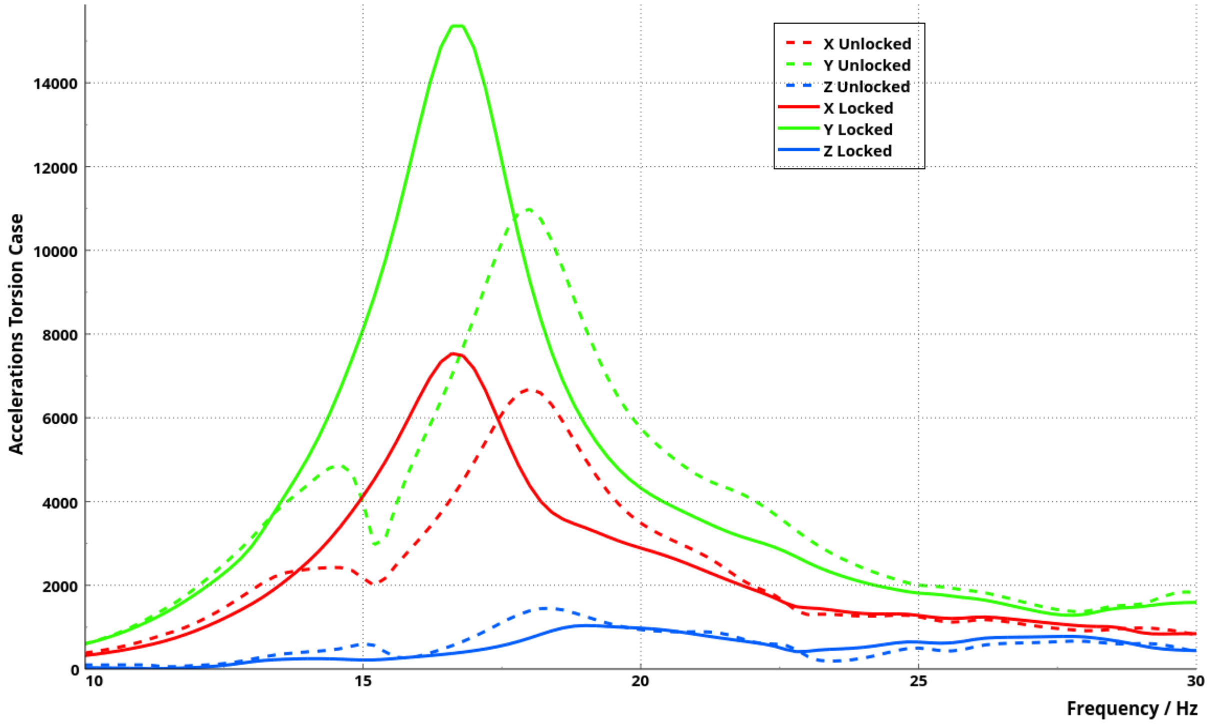

Figure 89: Model Torsion Engine Locked-Unlocked A-Pillar

In the flexion case, as we can see in the Figure **??** and the Figure 91, the curves that we obtained in the real case and in our model were completely different. Along this project we are saying that the flexion case did not give us the results as good as the torsion case. Now we are going to say that the problem could be in the SUBCASE 310 in the NASTRAN file. It is possible that the input signal did not represent in an optimum way the real movement of the car. We did not have time to study this problem, but we offer this solution, to change that SUBCASE in the NASTRAN file. So, the results were better in the torsion case because the curves of the flexion case were not so similar to the real results than the curves of the torsion one. Therefore, it was easier to observe more reliable results in the torsion graph.

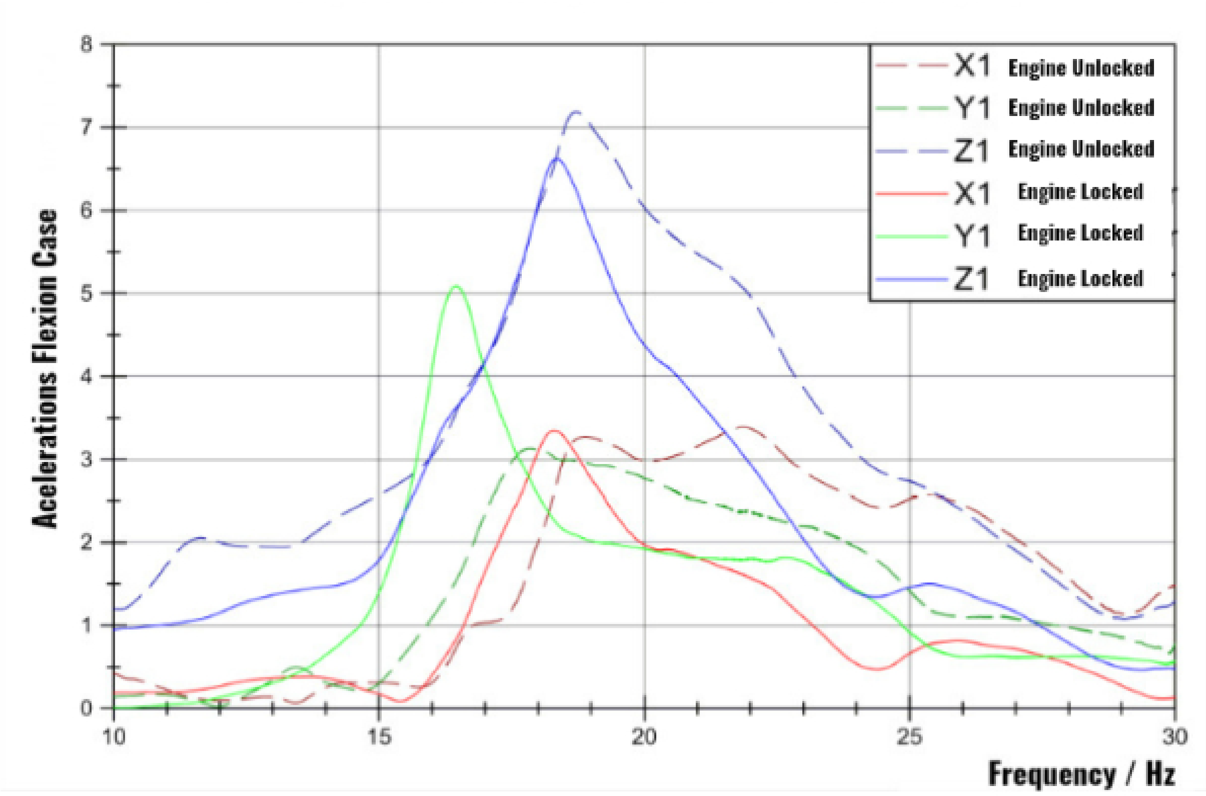

Figure 90: Flexion Engine Locked-Unlocked A-Pillar

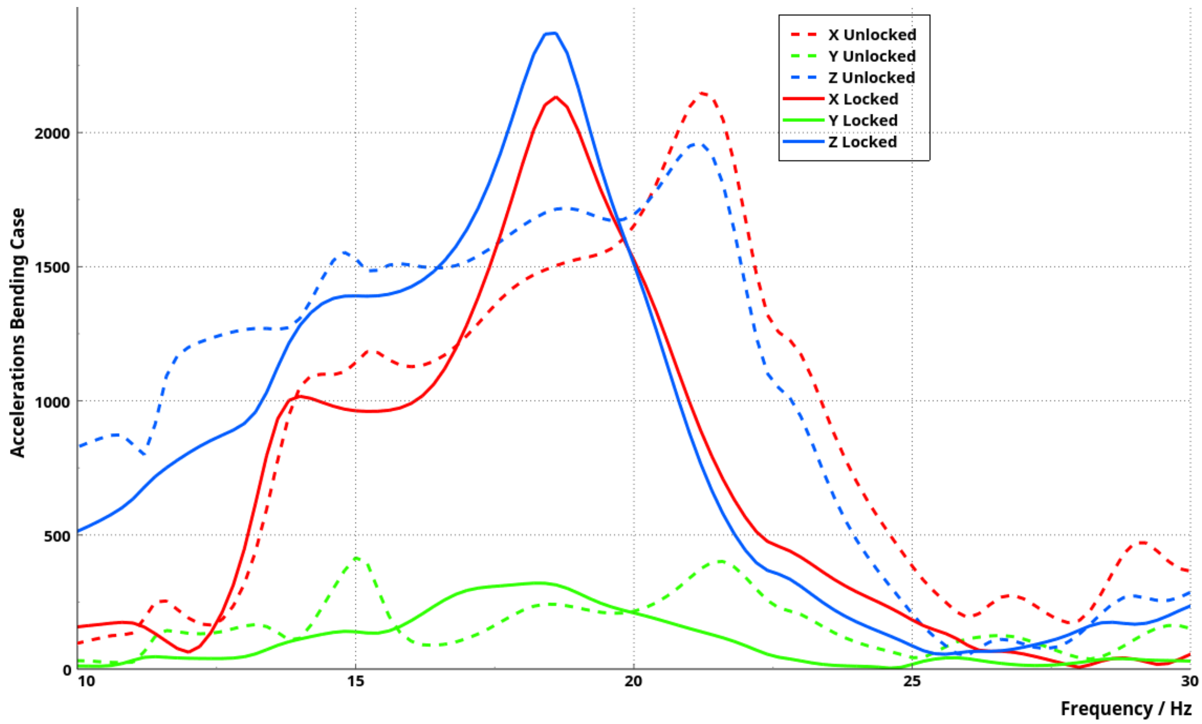

Figure 91: Model Flexion Engine Locked-Unlocked A-Pillar

9.2.2 Difference between the Engine Locked and Unlocked in the Middle Point of the Wheel

Again as in the A-Pillar point, the torsion case in the middle point of the wheel had some characteristics features. However, the differences were more evident in this point. We can see, the curves in the Figure 92 and the Figure 93.

On the one hand, we could see that the shape of the curves were similar. There was just one peak corresponded more or less to the four curves. The amplitude began in low frequencies, it increased to the peak, then it reduced to lower frequencies and finally it maintained its value.

On the other hand, there were some difference. First, the curves were displaced to lower frequencies in our model. Thus, the peak was in 17-17,5Hz in the real case and in 15 in our model. Obviously, the amplitude was lower in our model, because we had this feature in all our curves and we tried to change it as we will see in next chapters. Also, the curves of the engine locked were higher in our model, but in the real one depended of the wheel selected.

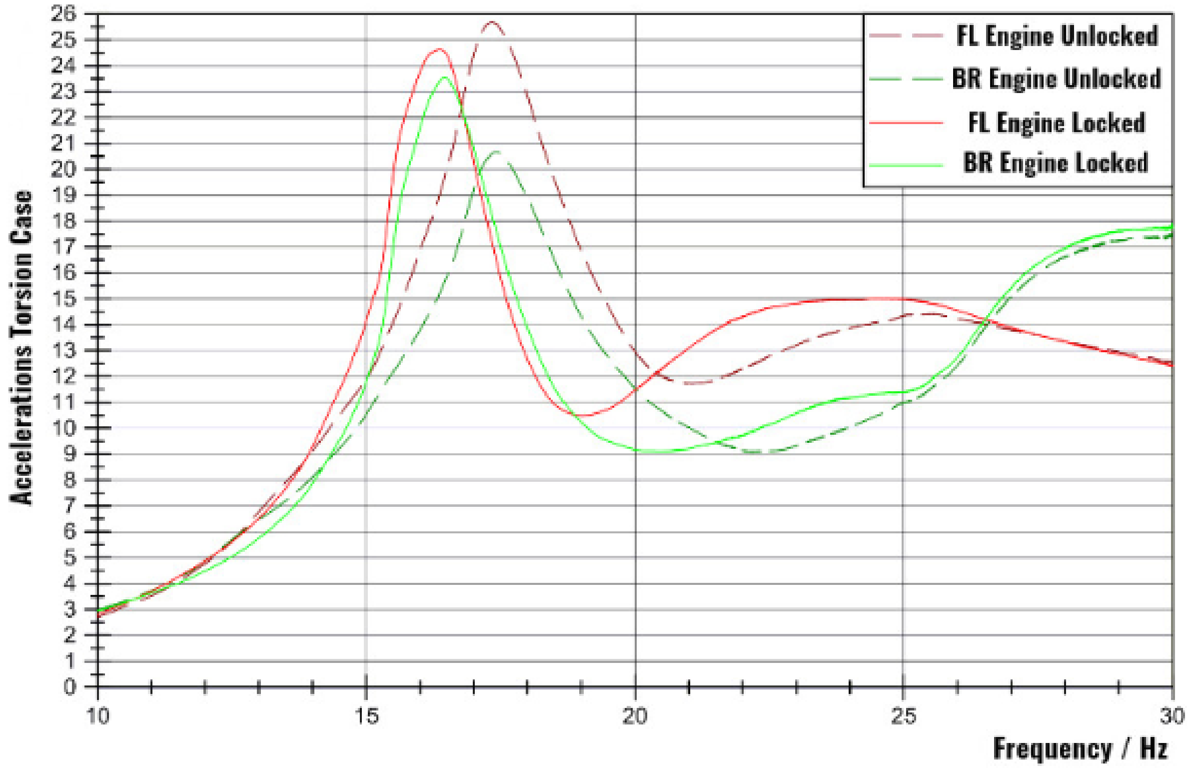

Figure 92: Real Torsion Engine Locked-Unlocked MP-Wheel

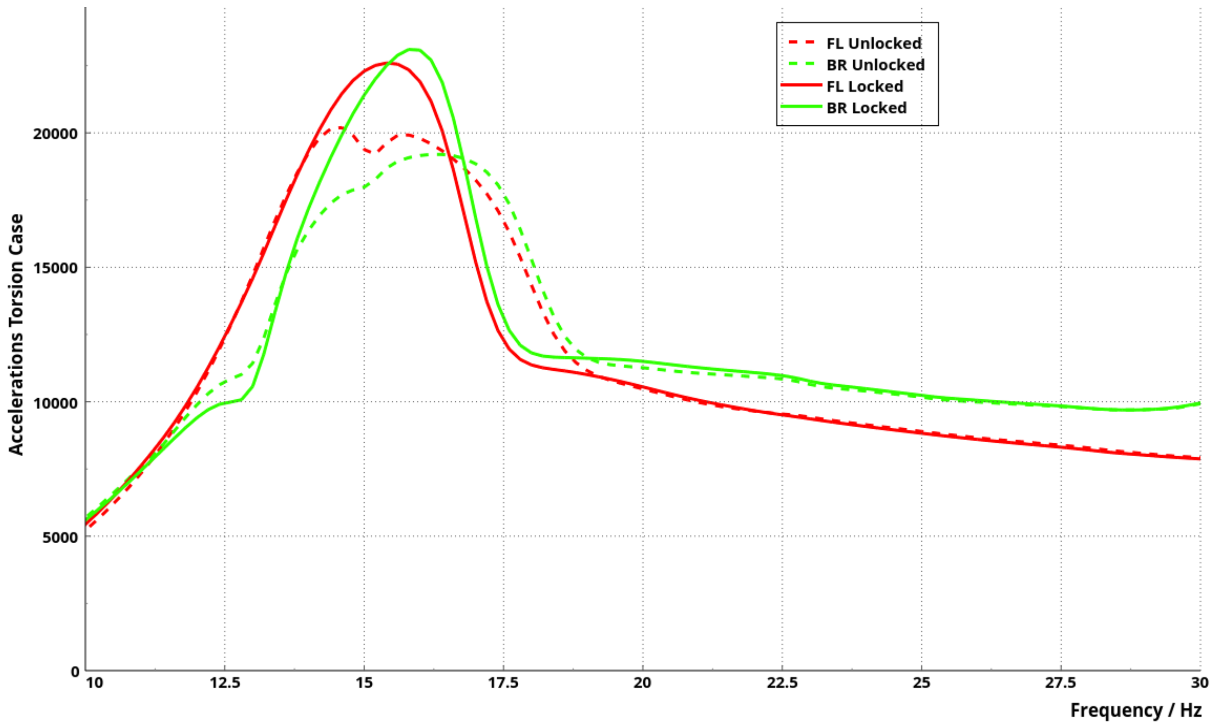

Figure 93: Model Torsion Engine Locked-Unlocked MP-Wheel

In the flexion case, if we studied the Figure 94 and the Figure 95 deeply, we can see that the curves that we obtained in the real case and in our model had some features in common. The difference just was that they were displaced more or less 8Hz.

In the real case, there was a peak in 21Hz. We could observe this peak clearly in the front left wheel. However, this peak was displaced to 13Hz in our model. Also, the amplitude was different as in the other curves that we have studied.

The shape of the curve was identical in both graphs. The front left wheel had a peak and after this, it was continuously decreasing. The back right wheel had a peak in the same frequency than the front left one but with a lower amplitude. Then decreased too, but after 3 Hz started to increase again to higher amplitudes.

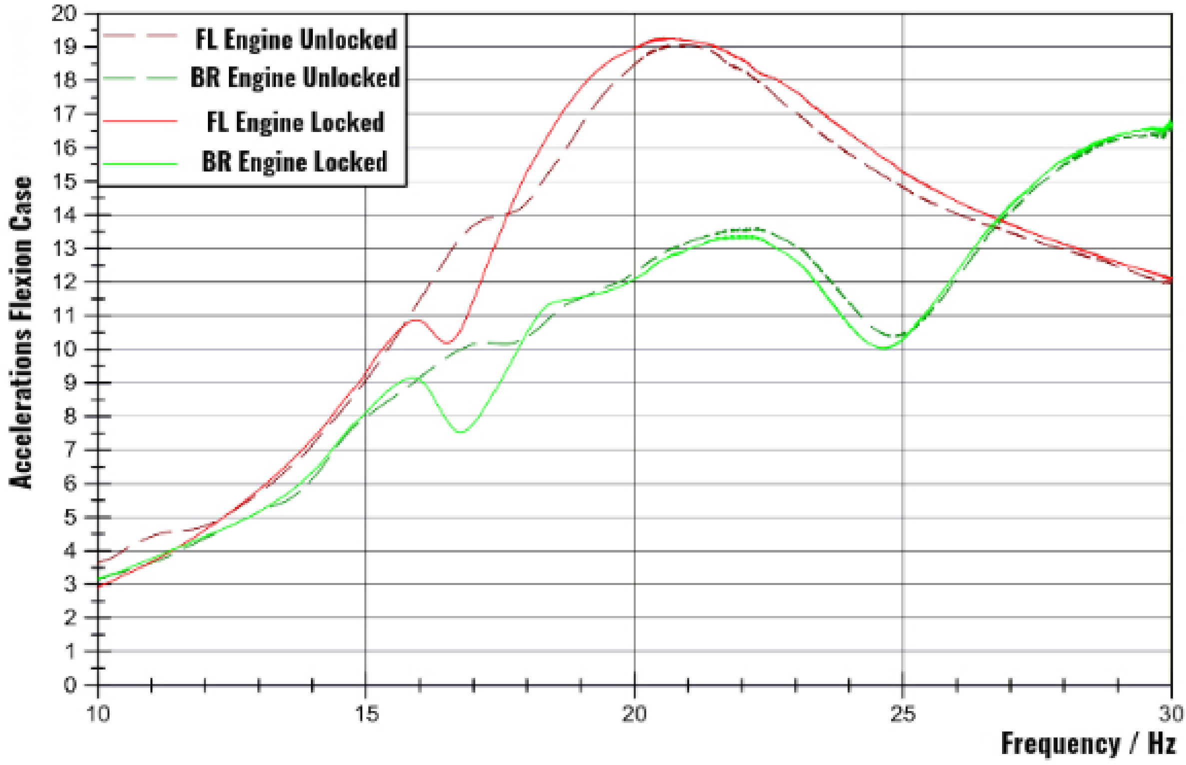

Figure 94: Real Flexion Engine Locked-Unlocked MP-Wheel

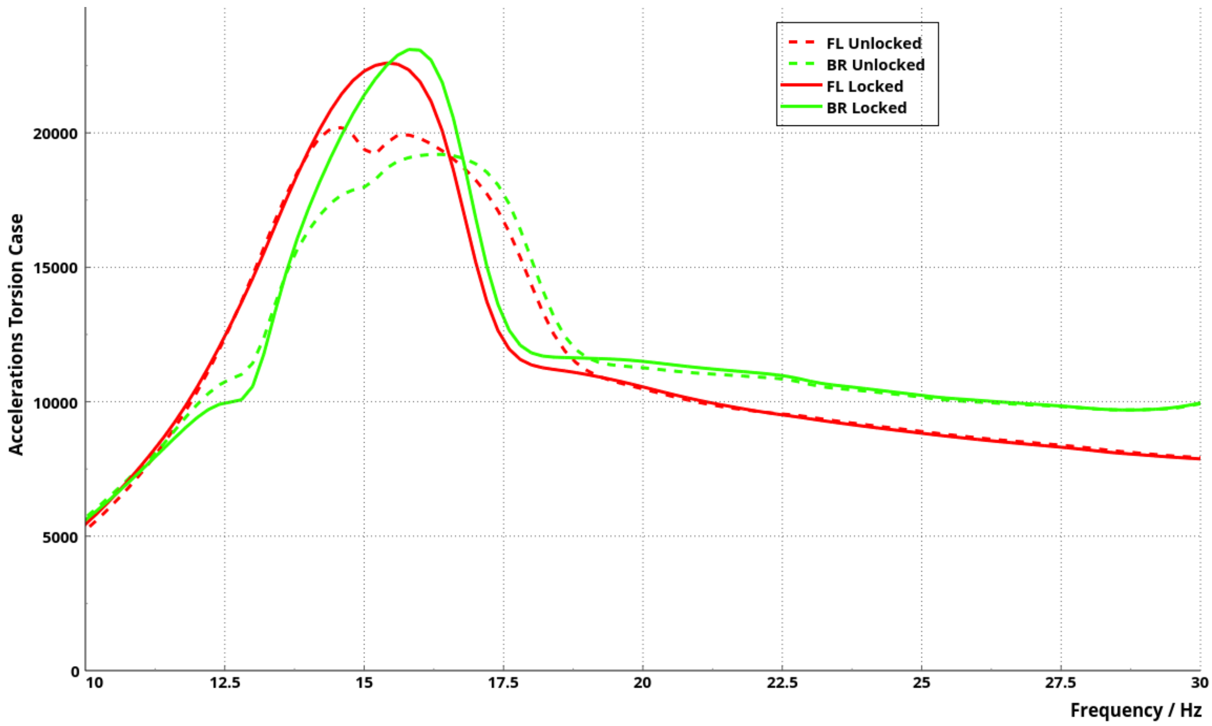

Figure 95: Model Flexion Engine Locked-Unlocked MP-Wheel

10 Roof Test

From the beginning, we were given two different NASTRAN files. One corresponded with the car model with the roof, and the other one is with the roof removed. As we were studying a convertible car body, we consider this point really interesting for the behavior of the car.

The process that we followed to make this comparison was simple: first we simulated the car body with the roof opened and then with the other NASTRAN file with the roof opened. We have to say that in this comparison, we only did the simulations with the unlocked engine, because our colleague Mr. Krampe had made his tests only with the engine unlocked.

Also we have to know that in this tests, the simulations are done with the next parameters changes:

- Dampers front axle changed from 2 (Nmm/s) to 1,2 (Nmm/s).
- Dampers rear axle changed from 1,75 (Nmm/s) to 1,15 (Nmm/s).
- Wheel stiffness front axle changed form 270 (N/mm) to 250 (N/mm).
- Wheel stiffness rear axle changed from 290 (N/mm) to 250 (N/mm).

10.1 Modal Analysis of the Car Body with the Roof

As we mentioned before, to start any modal analysis of any kind of structure, it is required the modal analysis, because thanks to it, we would be able to focus on the natural frequencies, and lately, we could understand better the behavior of our car for some particular frequencies.

The number of natural frequencies that we have in this case are only 35. We could have got more number of natural frequencies, but it is a number that we consider enough for our research. So in the next table 1, we could see the 35 mode shapes and its frequencies:

MODE	FREQUENCY	MODE	FREQUENCY
1	0,0087184	18	11,7718
2	0,739025	19	12,9299
3	0,818643	20	13,2133
4	0,884742	21	13,2379
5	0,973229	22	13,3834
6	2,55062	23	13,6049
7	5,0467	24	13,8515
8	5,11421	25	13,9335
9	7,4204	26	15,552
10	8,02035	27	14,563
11	8,56128	28	14,6454
12	9,19666	29	15,1004
13	10,0532	30	15,12
14	10,7009	31	15,2227
15	10,9623	32	15,7897
16	11,2231	33	16,1918
17	11,378	34	17,0592
		135	17,7768

Table 1: Mode Shapes and Natural Frequencies with the roof

10.2 Modal Analysis of the Car Body without the Roof

By the same way as we have with the car boy with the roof, we could obtain the same data from the modal analysis for the car body without the roof. In the table 2, we can see the mode shapes and the natural frequencies for each mode:

MODE	FREQUENCY	MODE	FREQUENCY
1	0,00870797	18	11,7538
2	0,745069	19	12,9367
3	0,871228	20	13,0556
4	1,16142	21	13,2088
5	1,26654	22	13,6399
6	2,73768	23	13,7125
7	5,04399	24	13,8698
8	5,11534	25	13,3301
9	7,43926	26	14,472
10	8,02235	27	14,5301
11	8,55403	28	15,0974
12	9,19711	29	15,2067
13	10,0536	30	17,0514
14	10,7828	31	18,1192
15	10,9549	32	18,686
16	11,2223	33	18,8297
17	11,376	34	20,5777
		35	21,317

Table 2: Mode Shapes and Natural Frequencies without the roof

10.3 Analysis of the Real Car with and without Roof with the Engine Unlocked

10.3.1 Bending Analysis

The point that we have to analyze here is the A-pillar, as we have made in many other analysis. In this analysis we have to consider that we do not have the modal analysis and the data that we can obtained from it, just because in the real car was not possible to get them. So we would like to explain these graphics by themselves.

If we begin looking to the Figure 96, that corresponds with the bending, the first thing that it comes to our mind is the peak made in the Z axis with the roof opened in the frequencies around 17 and 18 Hz. This peak was made arbitrarily, in the meaning of the amplitude, because if we look in the other peaks form the axes X and Y, the amplitude are slower than this one.

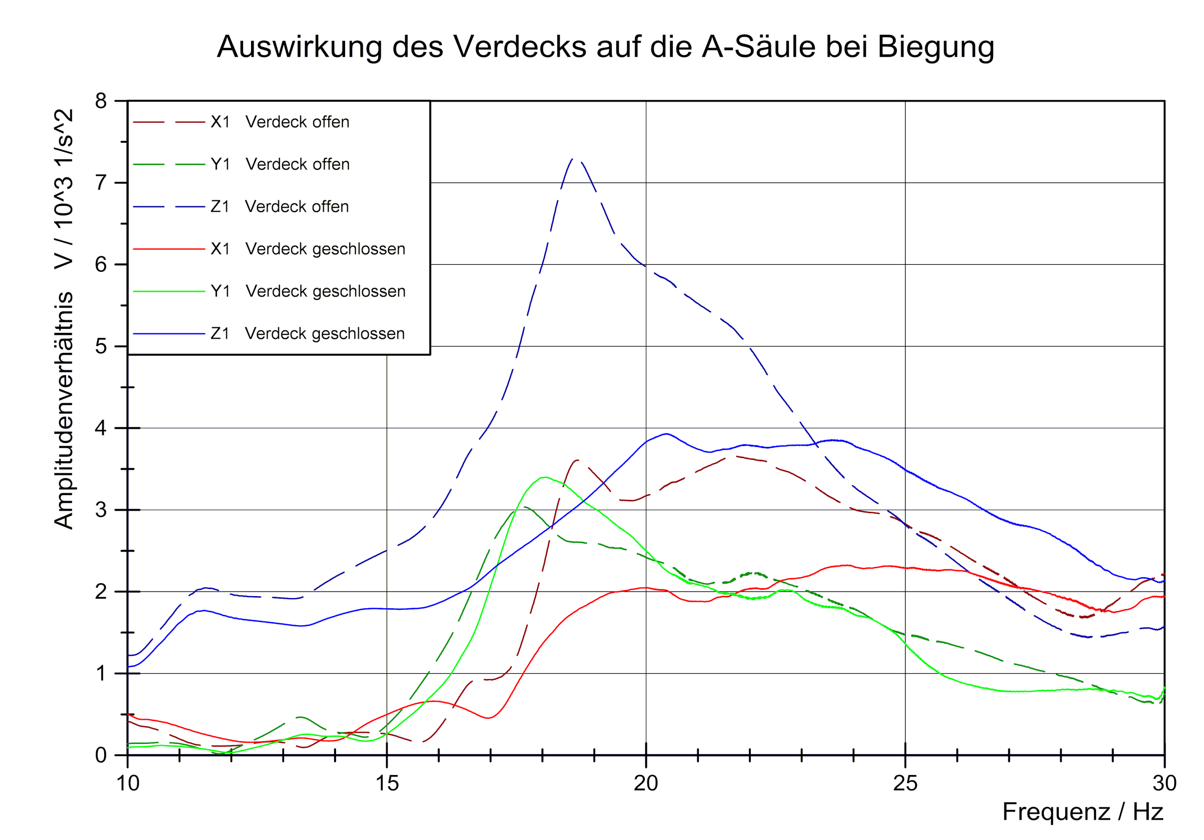

Figure 96: Bending Real Car Body with and without roof

We can appreciate a really big difference in shape and amplitude in the Z axis. So, we could guess that the results are completely different only for the influence of the roof. This an important theory that we should keep and see if later in the next tests will happen the same, because maybe in our models, the roof was the source of bad results.

For the X and Y axes, approximately the shapes are similar comparing to the Z axis results. Another remarkable point could be that in the bending case, the amplitude of Z and X axes are always bigger than the amplitude of the Y axis.

So to conclude with this bending analysis, the results are not very clear and we can not take any idea of the reason why the car behaves like this.

10.3.2 Torsion Analysis

If we continue with this analysis, we should to move on and see whats happens with the torsion. The torsion results are presented in the Figure 97:

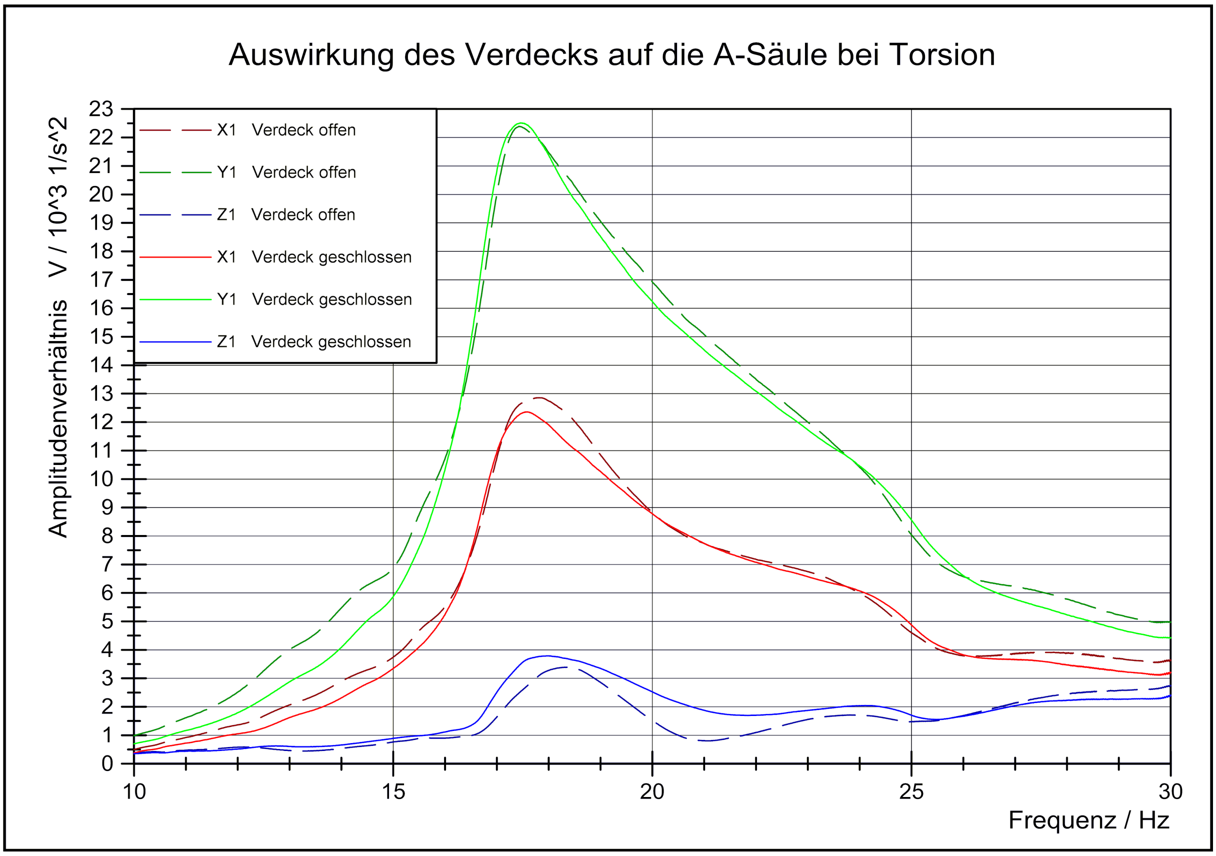

Figure 97: Torsion Real Car Body with and without Roof

In this case, the curves are cleaner than in the bending graphic. Moreover, we can clearly see that there is not a big difference in all the axes between the curves with and without the roof.

The biggest amplitude is made in the Y axis, that it is different than in the bending, where the Z axis was the biggest one, and here the Z axis is the smallest.

It is quite impressive the change of the behavior of the car under bending and torsion input signal, in the meaning of the shape and light of the curves.

10.4 Analysis of the Model Car with and without Roof with the Engine Unlocked

10.4.1 Bending Analysis

We start the analysis of our model studying the bending case. As we can appreciate in the Figure 98, there are four important stretch to consider when the car is closed:

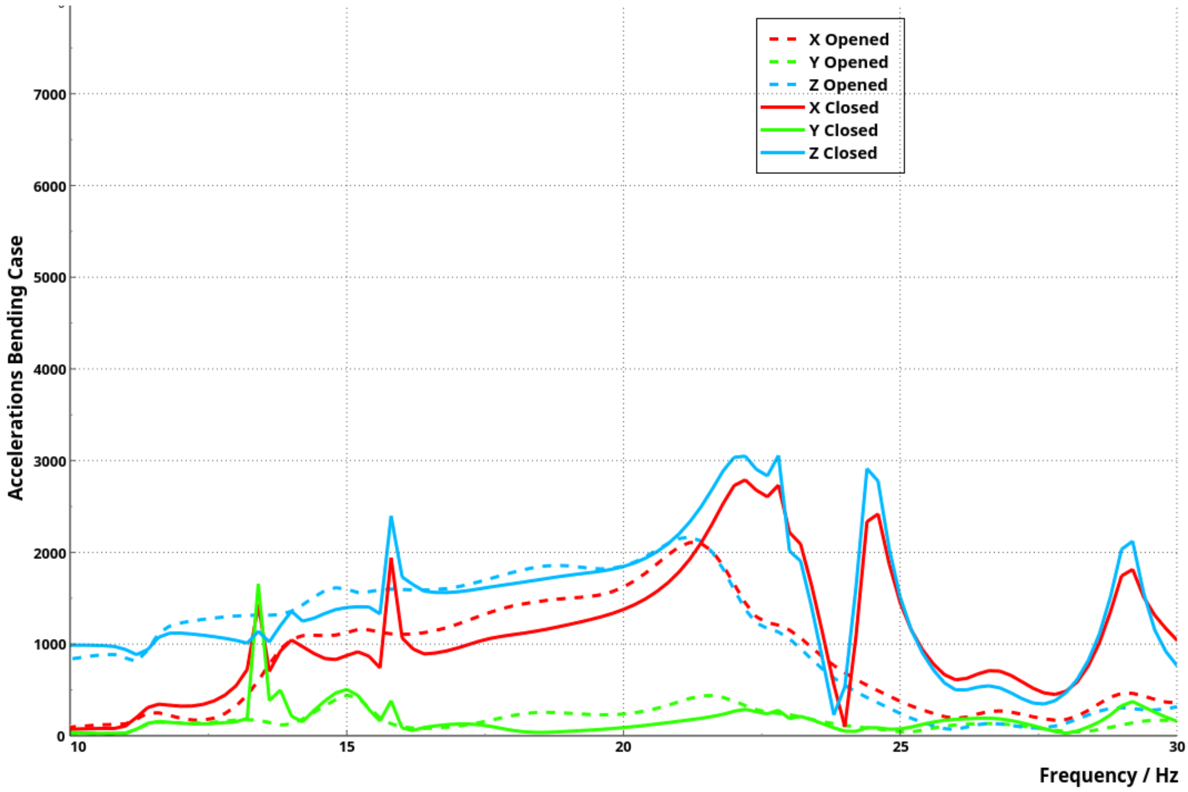

Figure 98: Bending Model with and without Roof

- The first corresponds with the frequencies around 13 and 14 Hz. The peak only appears in the Y axis, while in the other two axes does not happen anything. This is curious as we could see later, because every time that a peak appears in the Y axis, anything occurs in the other two axes. Unlike with the other two axes, X and Z, will happen the same, that every time that we have a peak in these two axes, nothing happens in the Y axis.
- The next peak that we find is the one around 15 and 16 Hz. In this point we can appreciate a very small signal from the Y axis, but nothing if we compare with the other axes.

- The highest peak from the X and Z axes are made between the 22 and 24 Hz. We can see that both axes has the same shape, but nothing happens in this point with the Y axis. It is important that the two highest peak from X and Z axes are not very clean, so maybe that is the reason to the next important point in our analysis.
- Suddenly, X and Z axes have a slow down in the frequencies around 24 and 25 Hz. The reason why this slow peak is produced is not very clear, just because in the left axis nothing happens. And it is quite significant that after this peak, both curves increase again to some similar big peaks.

In the other side, if we start to analyze what happens with the opened car, we can say that there are not significant peaks along all the frequencies. Moreover, the shapes of the curves are very calmed and there does not exist big steps between them.

So, in both cases, opened and closed, we can affirm that the Y axis is the one that does not match with the other two axes. This is probably related because we are in the bending moment, and the accelerations should be higher in the other two peaks.

We should make a deeply study to see what happens in the decrease of X and Z axes, but it depends on the time that we have to do the simulations further.

10.4.2 Torsion Analysis

Now it is the turn of starting with the analysis of the torsion. There are three remarkable aspects that we can appreciate in the Figure 99:

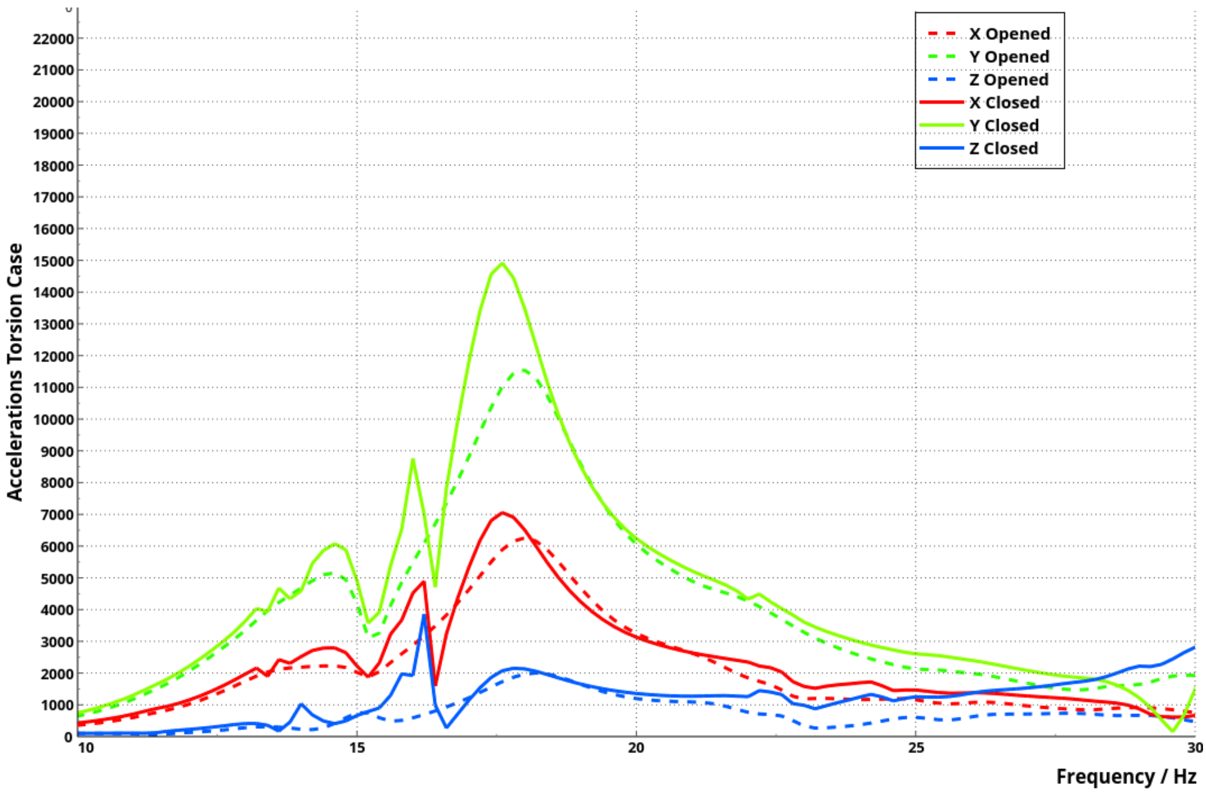

Figure 99: Torsion Model with and without Roof

The first important aspect is that in this case, there is one peak that is shared from all of the three axes. In this case, we can see perfectly the peak, while in the bending case was not so obvious. The peak is made around the frequency 17 and 18 Hz. If we remembered the modal analysis that we made for both cases of the model car with and without roof, we can easily look that in these frequencies there are some natural frequencies around. So for, here we have the reason why these peaks for all of the three axes occurred and moreover, with and without the roof, because in both cases there were natural frequencies.

Other important point if we focus in the model closed, we can perfectly see that around the frequencies 16 and 17 Hz, it is made a slow peak, so that maybe made because the disturbance of the roof in the A pillar, because with the model opened, we do not have this problem. For that reason, we will study now this range of frequencies deeply.

As we can see in the Figure 100 we decided to study the range of frequencies between

15 and 18Hz. In that range, we found 5 characteristic frequencies that we could study deeply. To do this, we change the command in the NASTRAN file when we were asking the output of our model. Now, we had the displacements and accelerations in ALL the point of the car instead of the SET of points. So, we could watch and record some videos with the movement of the car in the torsion case in this range of frequencies and after that we could study the causes of these curves.

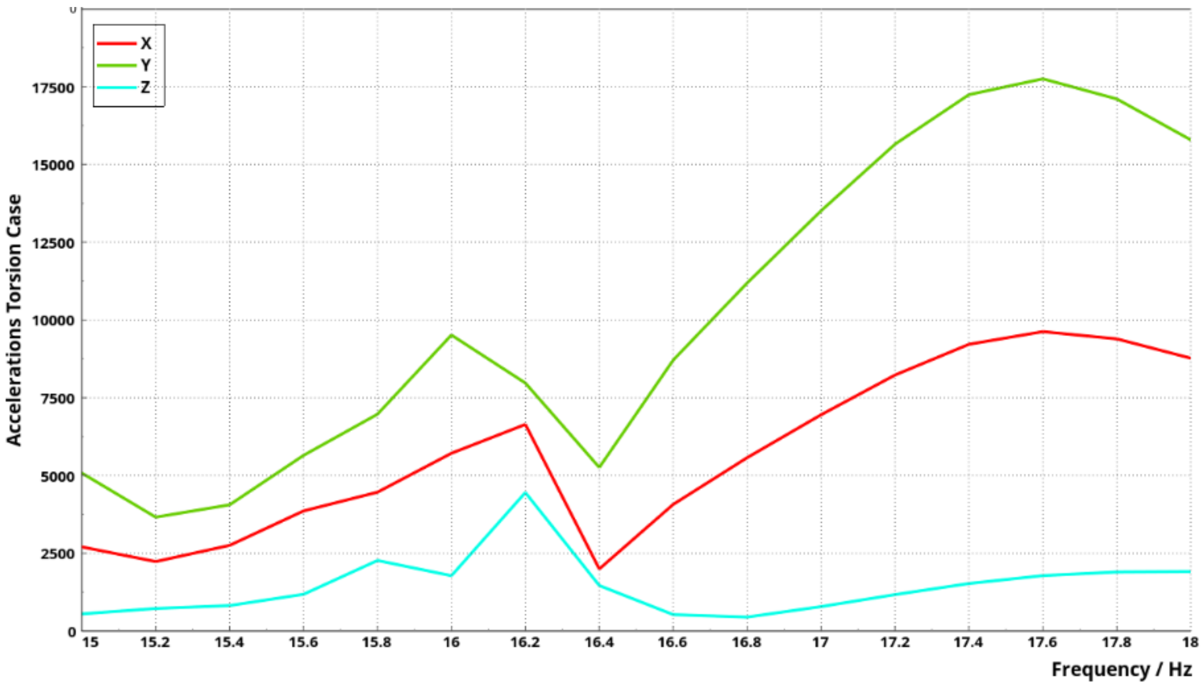

Figure 100: Study of the Range of Frequencies

In this document we can not add the videos, but we will try to explain what we could watch there. First of all, we had five videos about the movement of the car in the torsion case. We decided to watch the displacement in the Y axis in other color to check what parts were displaced more. Also, we put the scale multiplied per one hundred to watch better the results.

The first feature that we observed was that in the second frequency -16.2Hz- the movement of the back part of the roof was much higher than in the other cases. In this frequency, the the displacement in the A-Pillar point (where the graphs were referred) was not so high but there was a peak in the Z axis that could explain this. However, in the frequency of the peak in the Y axis in the A-Pillar -17.6Hz- the displacement of the roof in this axis was the lowest of all the frequencies studied. For that reason, we decided that the roof was not well modelled and we should change that to have better results in the model with the roof closed. This problem is normal because the roof combines a set of components that is really difficult to model.

10.5 Analysis of the Model Car with and without Roof with the Engine Locked

10.5.1 Bending Analysis

As we explain in the previous chapter, we tried to locked the engine to avoid the problems that it added to our study. Also, we wanted to know the influence of this component in the model. For that reason, we will study now the difference with and without roof when the engine was locked. In this case we did not have the real curves, so we will just study our model results.

In the flexion case, as we can see in the Figure 101, we could not appreciate many features in its curves. One important characteristic when the engine was locked is that the shape of the curves was the same that when the engine was unlocked. So, we are not going to study again this because is explained before in the unlocked case.

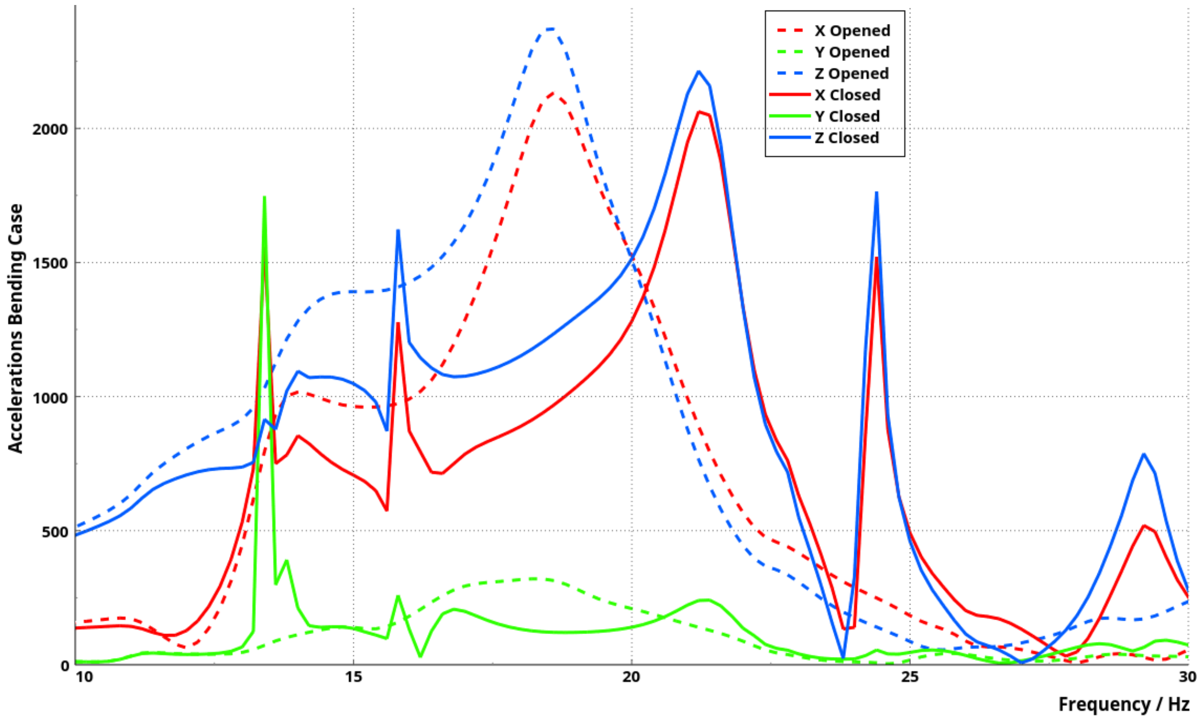

Figure 101: Model Flexion Engine Locked in the A-Pillar Point

10.5.2 Torsion Analysis

The torsion case, that we can see in the Figure 102, we had one remarkable feature that was not in the other cases. Around the frequencies between 15 and 17.5Hz, whe the roof was closed, the curves had two characteristic peaks that were cutting the original shape of the curve in two parts. We had this problem in each simulation that we tried to do when the engine locked and the roof closed, so we decided to study this range of frequencies as in the engine unlocked case.

The result of this study was the same that in the previous one. When the displacement in the A-Pillar point in the Y axis was the highest, the movement of the back part of the roof was the lowest. However, when the displacement in the A-Pillar point in the Y axis was the lowest, the movement of the back car of the roof in that axis was the highest. Curiously, in that last case, there was a peak in the Z axis amplitude.

For that reason, we decided again that the roof was not well modeled and we should try to change the elements and parameters to approach the model to the real one.

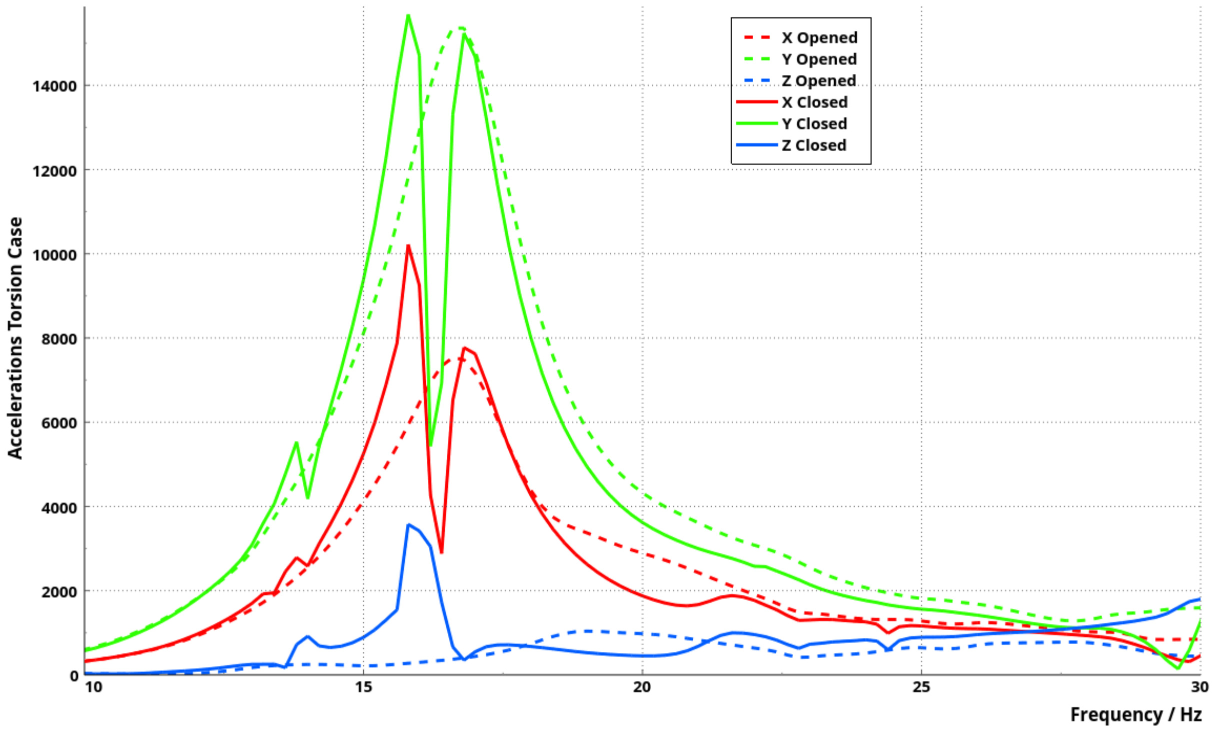

Figure 102: Model Torsion Case in the A-Pillar with the Engine Locked

11 Trying to Approach the Real Results

11.1 Changing Properties of the Roof

As we have observed, in the model with roof the curves were less clean and had more harmful frequencies in a certain range than the real curves or simply than the curves without including the roof. So, we decided that it might be one of the problem that the model had and we wanted to solve it. As with the other components of the car, we tried to modify the parameters of the roof to approach to the real model. In this case we wanted to reduce the number of peaks to have a curves cleaner.

The roof was composed by CQUAD4 and CTRIA3 elements. These elements defined a plane strain which joined three or four grid points. So, the roof was made by all these components joined in the points they had in common. Both the CQUAD4 and CTRIA3 elements of the roof were related with the same PSHELL command. This one defined different properties that we were going to modify.

Because we did not have to much time, and also we studied the model without the roof, we just had time to try once to change the properties. In that occasion, we decided to change two parameters that we considered important, the thickness and the modulus of elasticity. If we increase the thickness, we are probably increasing the longitudinal stiffness and also the rotational one. We say probably because, as we commented before, the behavior of the car depended of all components and their relationship. The modulus of elasticity relates the stress with the deformation so, if we change the parameters, the behavior will be different.

Then, we tried to increase the thickness from 1,5 to 3,0mm and the elastic modulus from 13 to 26. The results are shown in the Figure 103 and the Figure 104.

As we can observe in the torsion graph, the new curves had the same main shape than the initial model. However, there were a lot of differences between them. First, we got the goal that we wanted. The new curves were cleaner because the number of peaks had been reduced and they were not as pronounced. Also, the amplitude of the curves was slightly higher than before, less than 1 per cent. Besides, the peak frequencies were displaced to the left. It means that the frequencies with the higher amplitude were a little smaller with the new parameters than with the initial ones.

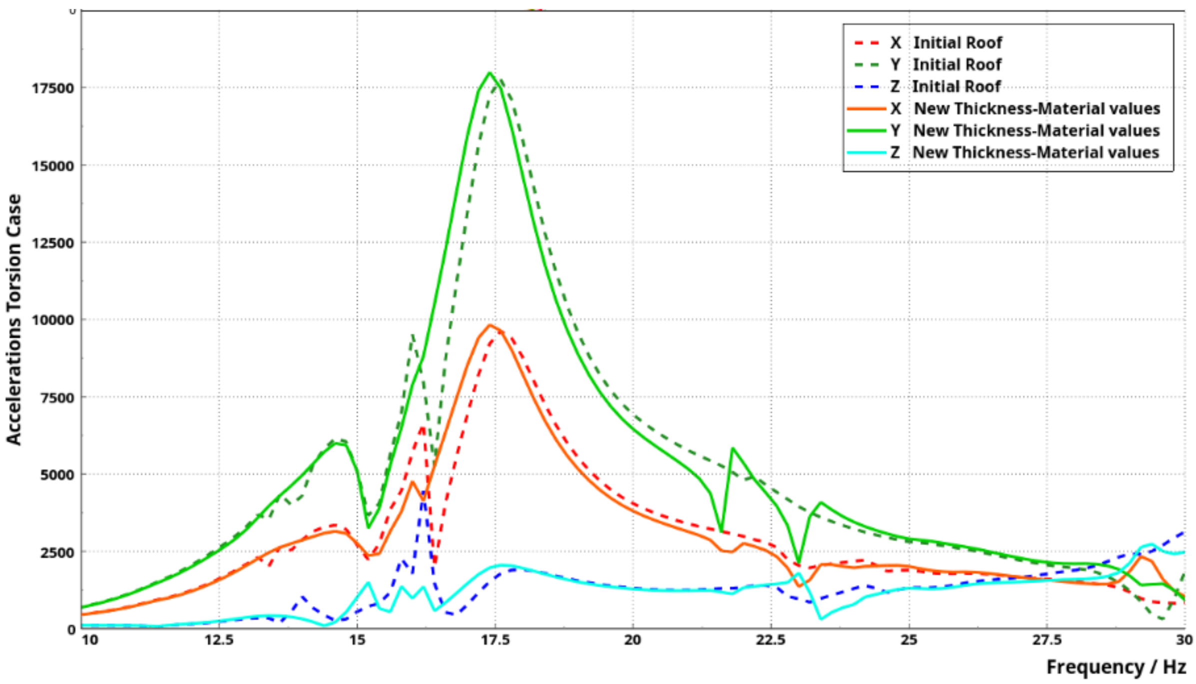

Figure 103: Torsion Case Changing the Properties of the Roof

The flexion case was worse represented than the torsion case. The results were not so similar to the real ones in this case, so we could not study the changes in this graphs with the same certainty. However, again we could observe that our goal had been achieved. The number of peaks were less than before and also the curves were cleaner. For example, the X and Z axes had now a remarkable frequency in 23,5Hz where the highest peak was. We could not assure anything about the frequency shift because in the flexion case we did not have a frequency as remarkable as the highest peak in the torsion case.

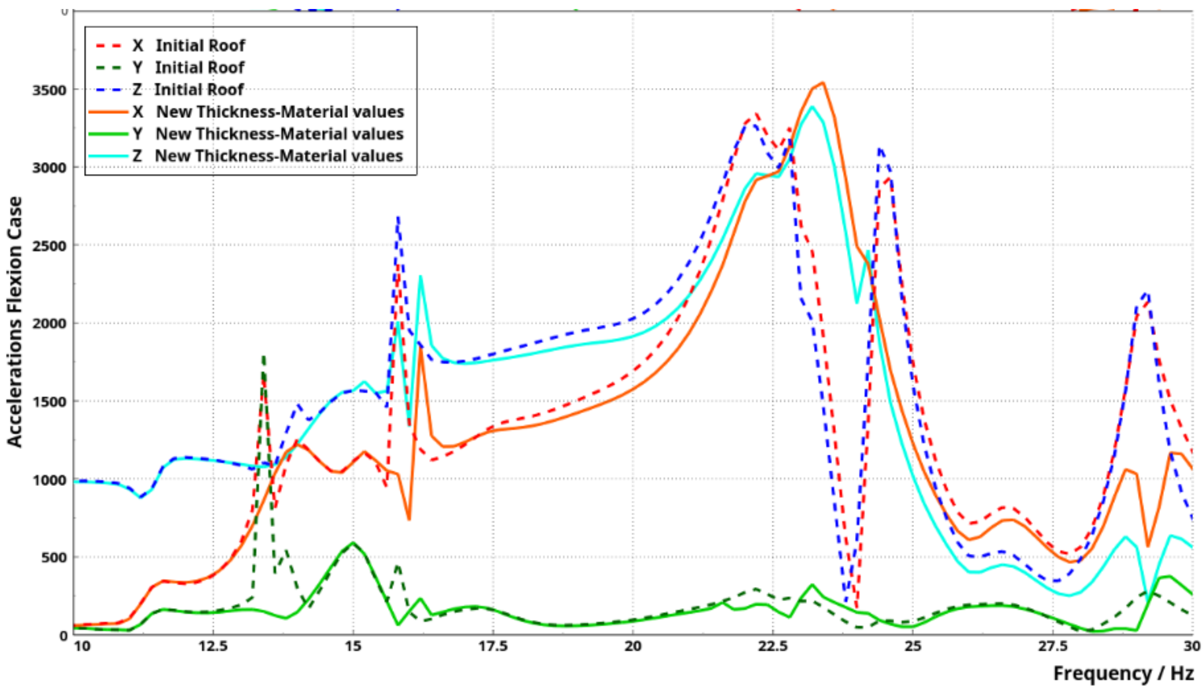

Figure 104: Flexion Case Changing the Properties of the Roof

Then, we can assure that the roof was at least partly responsible of the shape of the initial output. Probably if the roof were simulated better, the results would approach more to the real ones. This part of the car was really difficult to simulate because was in the highest part of it and the displacements and accelerations were larger there. Regrettably, probably if we had had more time to try with other parameters and other values, we could have approached more the results to the real ones.

11.2 Changing Dampers Values

The main goal of our project is to get the same results to get a validation of our model. To do so, as we have explained in other occasion, we changed the values of the dampers of our car, trying to get the same peak values for all of our axes.

The changes that we made the first time with the model opened were the following shown here in the table 3

Element	Units	Initial Value	Final Value
Dampers front axle	Nmm/s	1,2	4
Dampers rear axle	Nmm/s	1,15	4,15

Table 3: Dampers Changes

11.2.1 Analysis of the A-Pillar Point

We considerably changed the values from the dampers, so the results that we expected were a big jump in the peak values. To present better the information that we got, we made the comparison between both cases: the new values against the original ones.

We had the advantage that we knew before thanks to other tests, that if we increased the value of the dampers, we would have got a bigger peak amplitude than before. This increment is perfectly shown in the Figure 105 and the Figure 106:

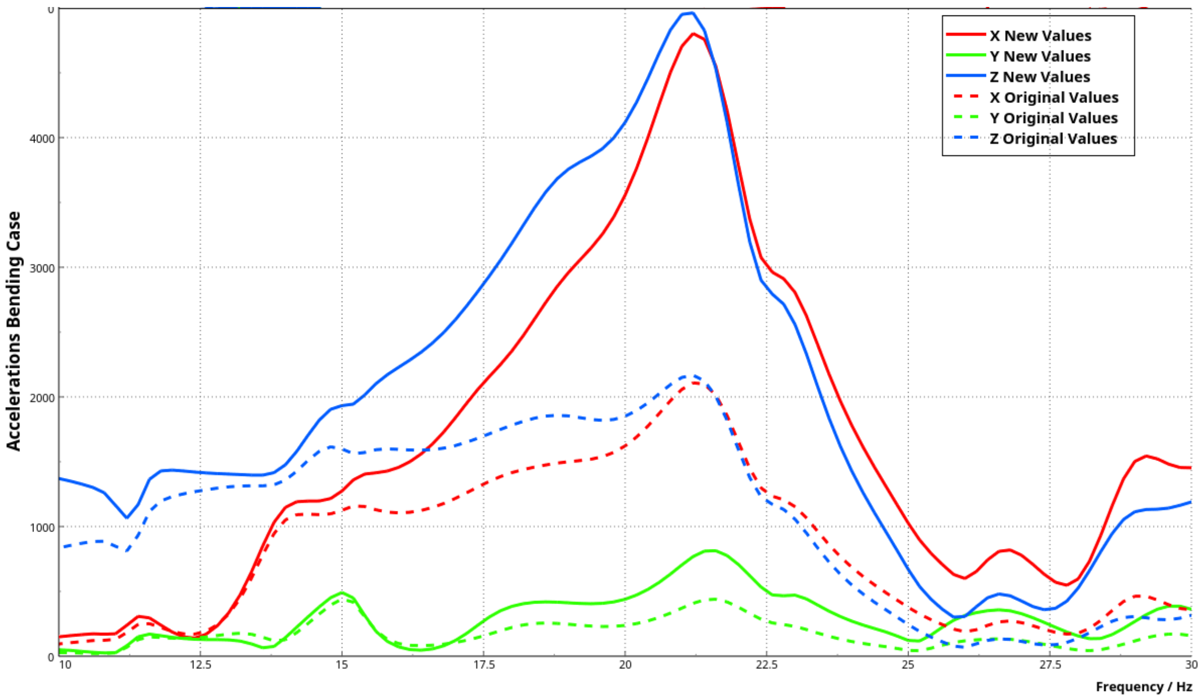

Figure 105: Bending Damper Changed A Pillar

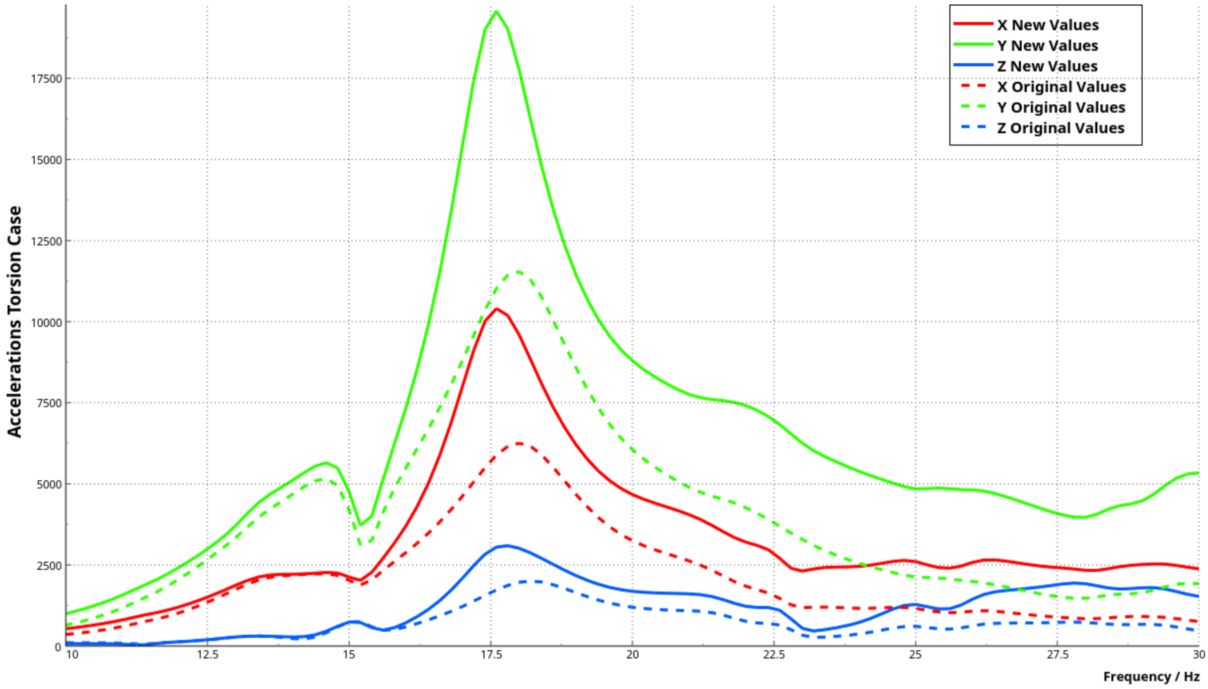

Figure 106: Torsion Damper Changed A Pillar

In both cases we can see that with the new values, the amplitude of the peaks were higher than with the original data. So, we are able to say that we are closer to find the correct parameter value of the dampers of our model car to get the proper validation.

The next simulation that we would like to introduce is the same results but increasing the damper values to reach the same peak amplitude as our colleague got with his real car body.

11.2.2 Analysis of the Middle Point of the Wheel

Another important point of analysis is the middle point of the wheel, because we need more than two points to approach to the real results, just because as we are going to see now, changing some parameters, they have another different influence depending which point we are studying.

If we start looking to the bending case, we can not obtain too much information about the influence of the dampers in these points. The peak frequencies in both wheels are out of phase, and the amplitude is so similar as we can appreciate in the Figure 107:

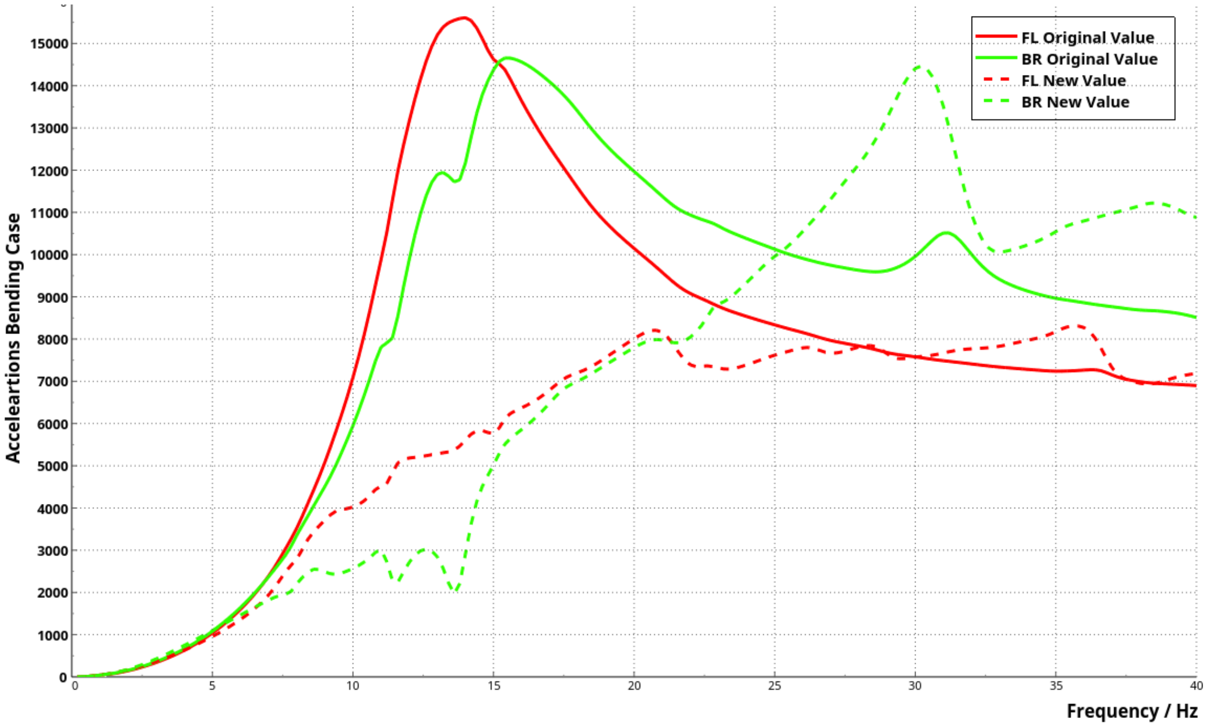

Figure 107: Bending Damper Changed in the MP

However, in the torsion case, we have an important data, that it will be very useful for us. If we see in the Figure 108, there is a decrease in the peak frequency amplitude when we raise the dampers values.

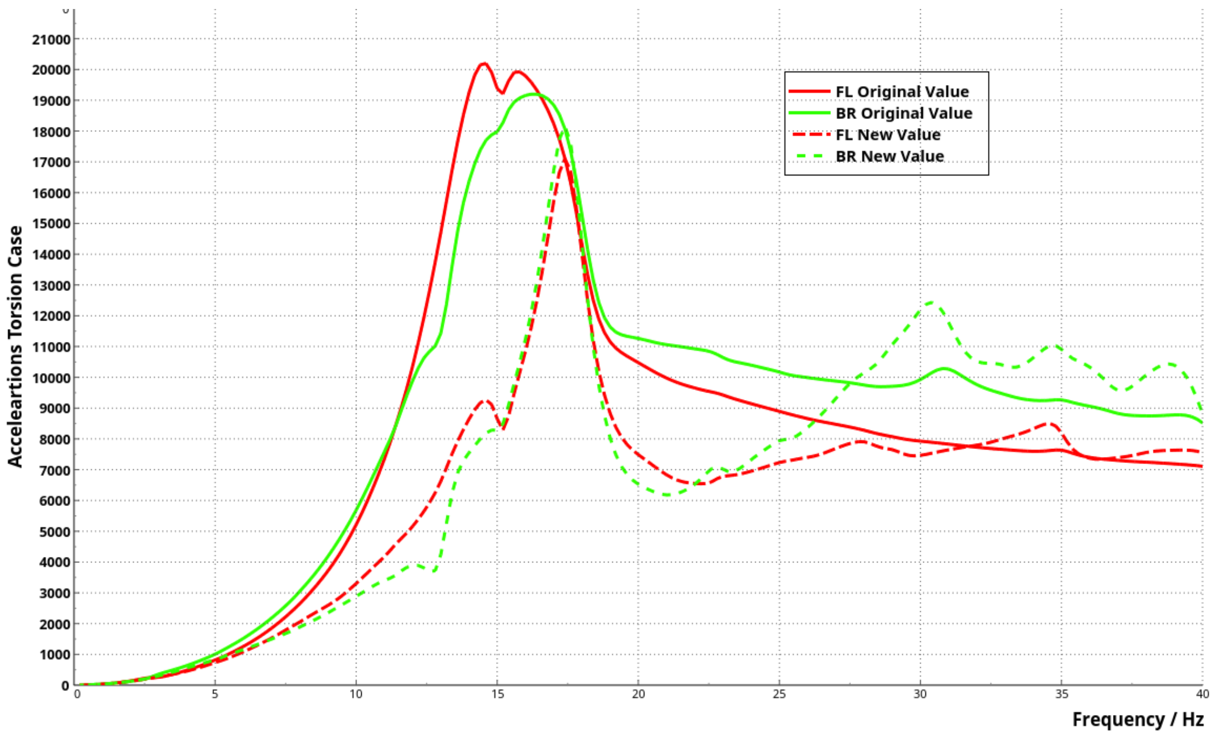

Figure 108: Torsion Damper Changed in the MP

We can see the decrease with the both new values in our graphic, so that means that we have an important new way of understanding the behavior of our car body, respect to changing the damper value referring to the middle point of the wheels.

As we have along all our project, the bending results were not so good, so for that reason, we should focus more in the torsion results.

11.3 Changing Wheel Stiffness

In this simulation, instead of changing the damper parameters, we decided that it would be useful try to know the influence of changing other parameters that could have a big influence in the response of the car. We are talking about the wheel stiffness.

So this simulation consisted on changing the wheel stiffness of our car body, but maintaining the same values for the dampers, not as in our last point presented. The main goal of this simulation is understand better the influence of the different elements that could have had an important meaning in the behavior of our car response.

At the end, changing the both parameters that we are referring, we should have been able of obtaining approximately the same values as the real car has, and at the end, it would have meant that we would have got the proper validation.

In this case, the changes made in the wheel stiffness were exaggerated, because we did not how it would have changed, so as we are going to appreciate in the results presented, the values are so far from the real ones.

The changes made are the followings shown in the table 4:

Element	Units	Initial Value	Final Value
Wheel Stiffness front axle	N/mm	250	500
Wheel Stiffness rear axle	N/mm	250	500

Table 4: Wheel Stiffness Changes

We increased the double the initial value that we had in our first simulations. We did this big increment because we would really wanted to appreciate the influence of changing the stiffness of the wheels with its elastic properties.

11.3.1 Analysis of the A-Pillar Point

We start the comparison looking to the Figure 109, where we can see the bending case. The peak amplitude is more than 6000 units bigger than the original one, only increasing the Elastic Modulus.

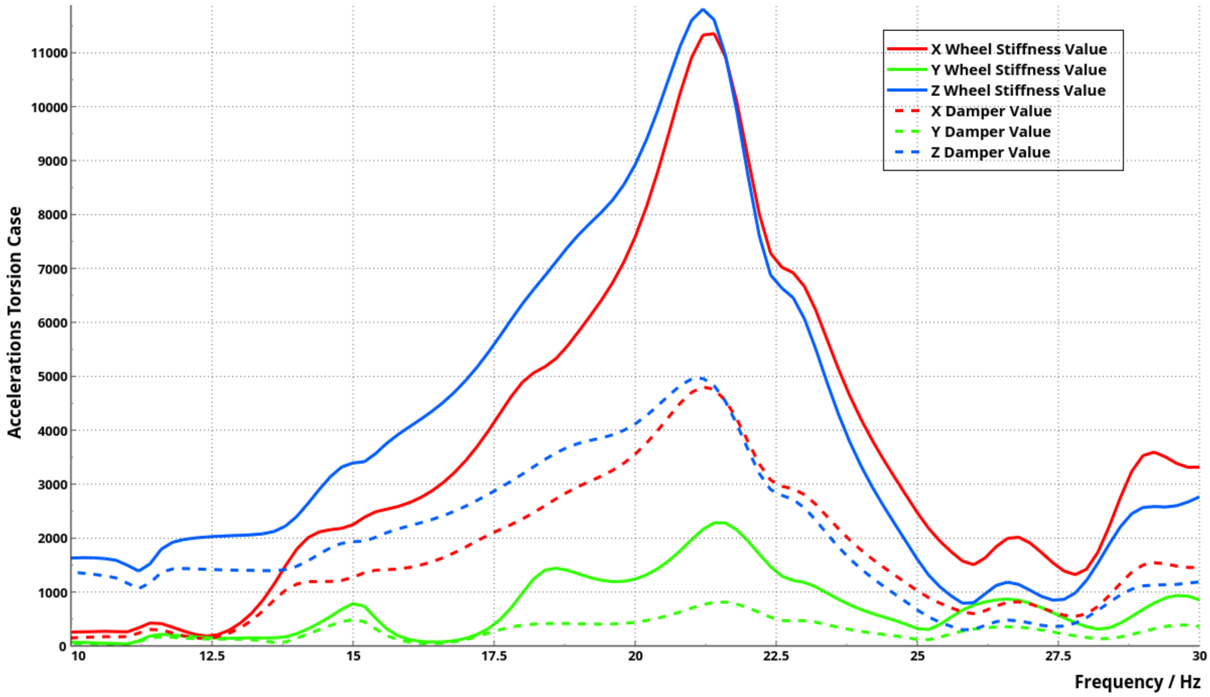

Figure 109: Bending Analysis of Wheel Stiffness Changes in the A Pillar

In the bending case we have the same shape for all the curves, just the change in the peak amplitude. If we would like to be more precise, we could say that maybe the peak frequency is quite different for the same axis, but it is not a big change.

If we concern now about the torsion case, in the Figure 110 we can see the considerably differences between this both cases.

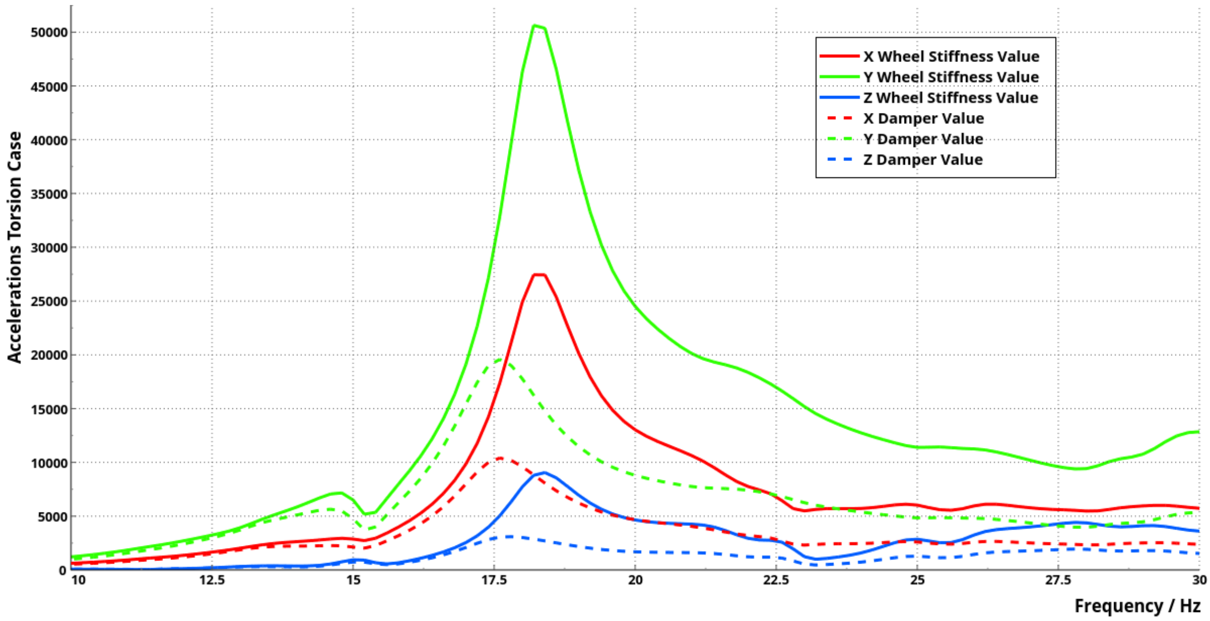

Figure 110: Torsion Analysis of Wheel Stiffness Changes in the A Pillar

The amplitude is so different. If we look for example in the Y axis, there is a step of more than 30000 units. This means that the Elastic Modulus is not accurate and we did a mistake changing for 500 (N/mm), it is a value disproportionate for what we were looking for.

Moreover, in the torsion case we can appreciate that it has been made a movement to the right of the peak frequency for all the axes. This could mean that as we increase the value of the Elastic Modulus, the natural frequencies of the car body could also increase. But we have to take a look that the value of the wheel stiffness is not at all accurate, just because it is no common have this big value for the wheel stiffness.

11.3.2 Analysis of the Middle Point of the Wheel

Beginning with the bending case, we have a curious case, because we have a coincidence with the behavior of the curves in the torsion case. We have that if we double the wheel stiffness, the amplitude also raises, as we can check in the Figure 111:

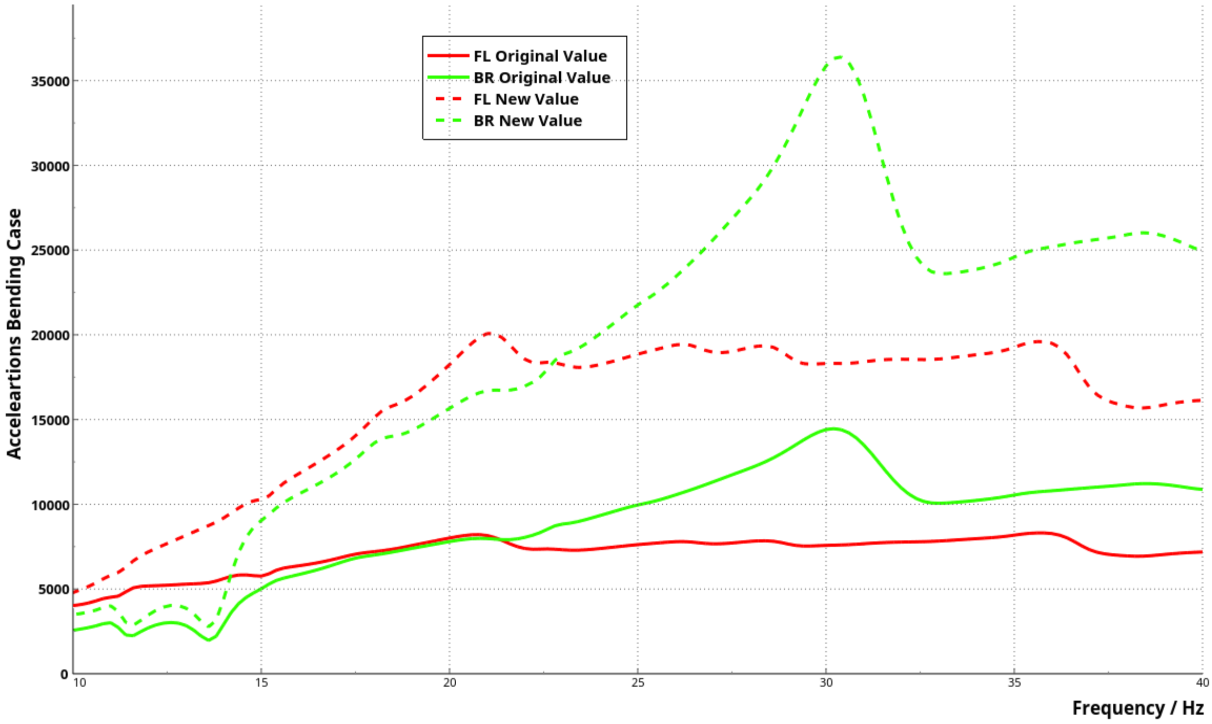

Figure 111: Bending Analysis of Wheel Stiffness Changes in the MP

This behavior is clearly seen in the torsion case, as we show below in the Figure 112:

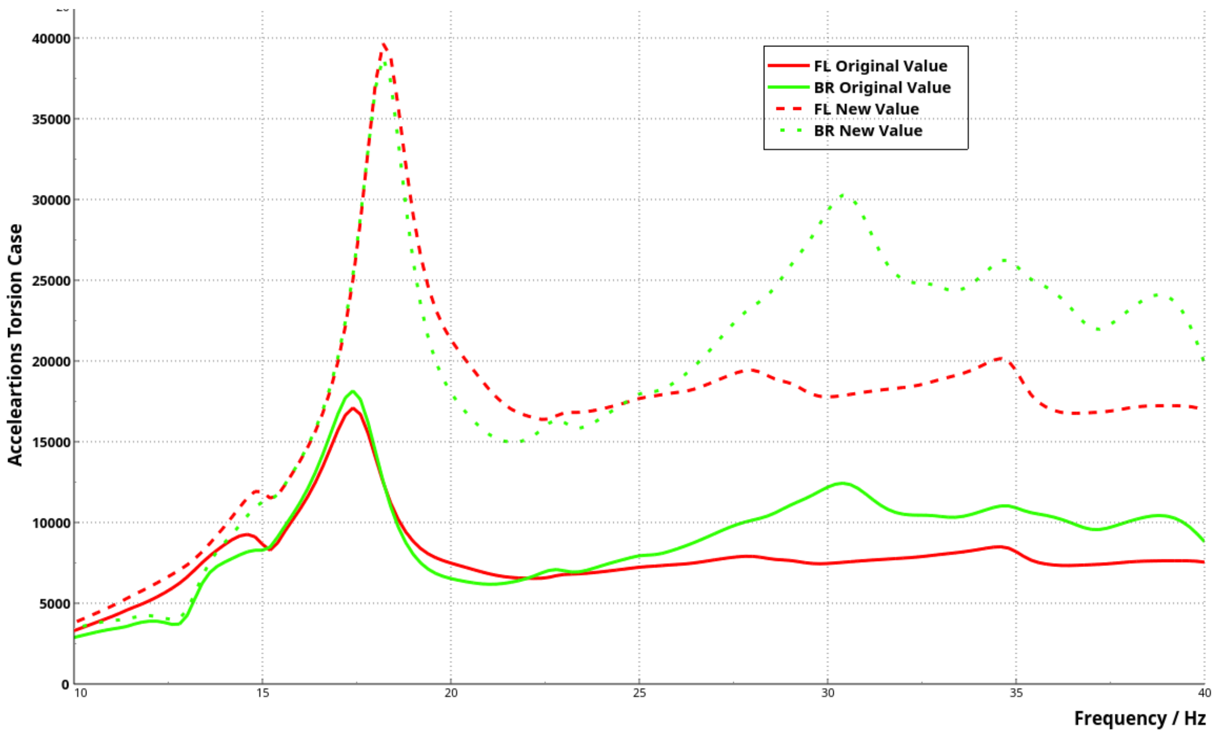

Figure 112: Torsion Analysis of Wheel Stiffness Changes in the MP

The new value from both wheel points is not accurate, as wu explained before with the A Pillar, just because the new value is the double of the original one. So for, we can appreciate a little change in the peak frequency, but we consider that the most important here is that the amplitude in both points raise when when amplify the wheel stiffness.

11.4 Changing Damping Coefficient

In the elastic properties, we are able to change the stiffness and the damping coefficient. The next test that we did were related changing this parameter. As we have seen in the last point that the stiffness has a great influence in the behavior of both points of studying, we have to compare if this parameter has also a big influence.

The changes that we did in the damping coefficient were the followings shown in the table

5:

Element	Units	Initial Value	Final Value
Damping Coefficient front axle	Dimensionless	0,07	0,14
Damping Coefficient rear axle	Dimensionless	0,06	0,12

Table 5: Damping Coefficient Changes

As we did in the stiffness case, we doubled the initial value, to see the impact. We did not know before doing the change if it would have had a greta repercussion, but we had to start for something.

11.4.1 Analysis of the A-Pillar Point

In this concrete case, we did not get any important information, just as we could see in the graphics, the change made in the damping coefficient for torsion and bending had not a significant change in the amplitude. So now, we can appreciate in the Figure 113 the bending case:

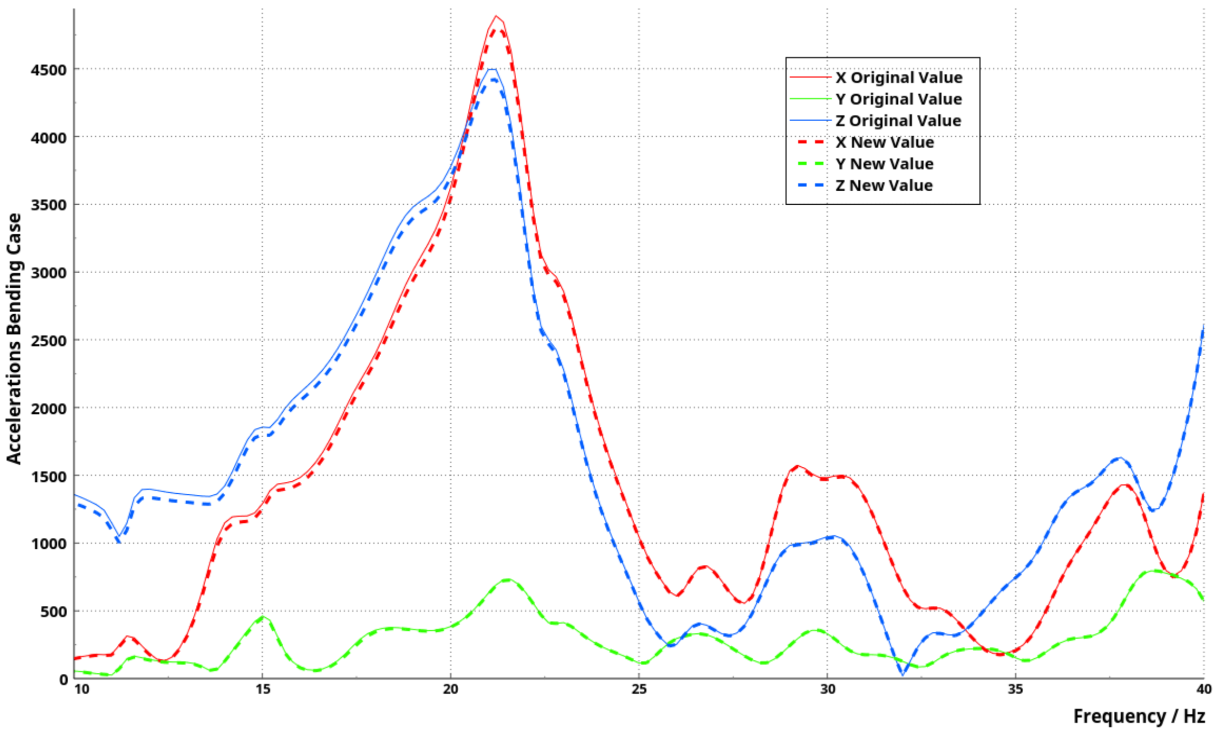

Figure 113: Bending Analysis of Damping Coefficient Changes in the A Pillar

In the Figure 114 shown here the torsion case:

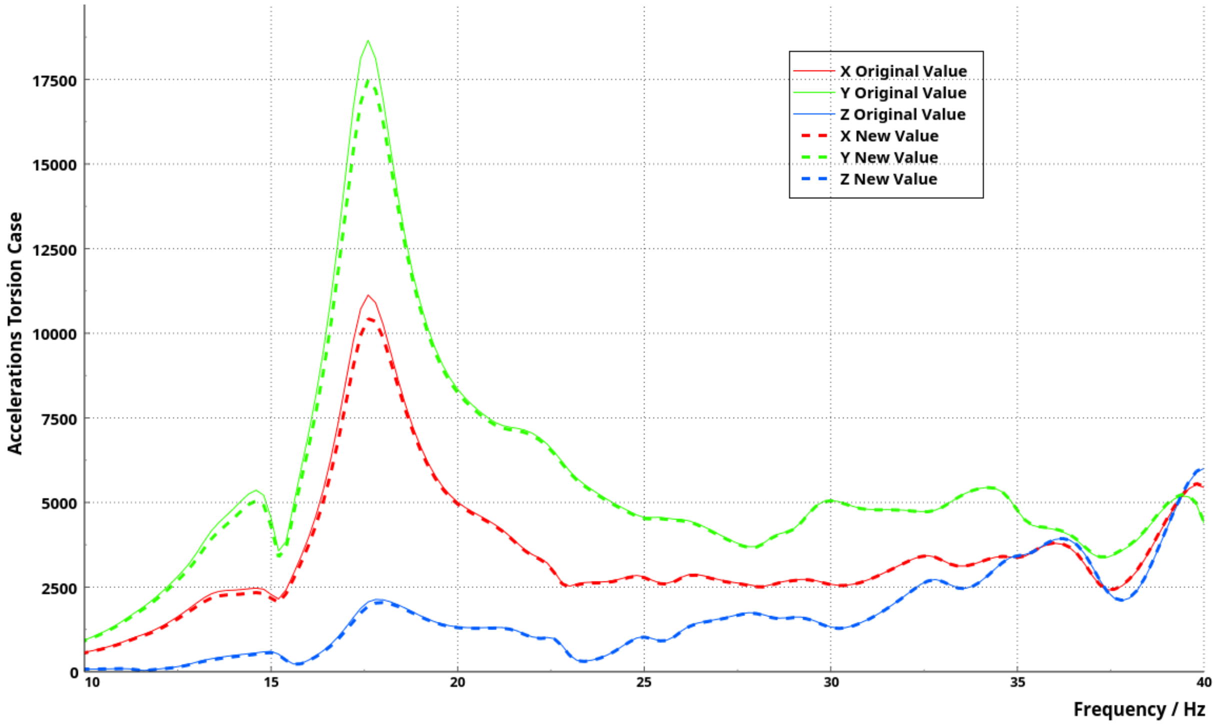

Figure 114: Torsion Analysis of Damping Coefficient Changes in the A Pillar

Both continuos and dashed curves have the same shape, so if we change the parameters of the damping coefficient, we are no going to obtain a great change in the amplitude.

11.4.2 Analysis of the Middle Point of the Wheel

In the same way as studying the A-pillar point, we have that we were not able of obtaining any value information from this results, just that the damping coefficient had not a big influence neither in the middle point of the both wheels.

We are going to show the graphics just to certificate what we are talking about, in the

Figure 115 and in the Figure 116:

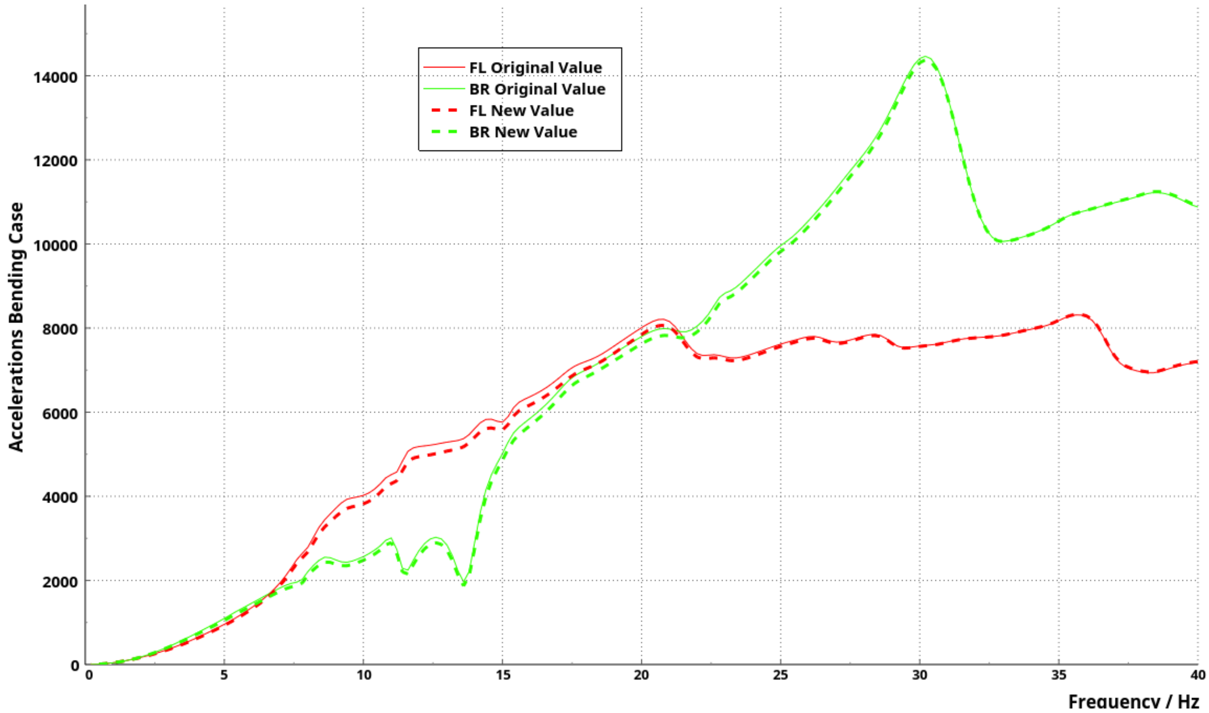

Figure 115: Bending Analysis of Damping Coefficient Changes in the MP

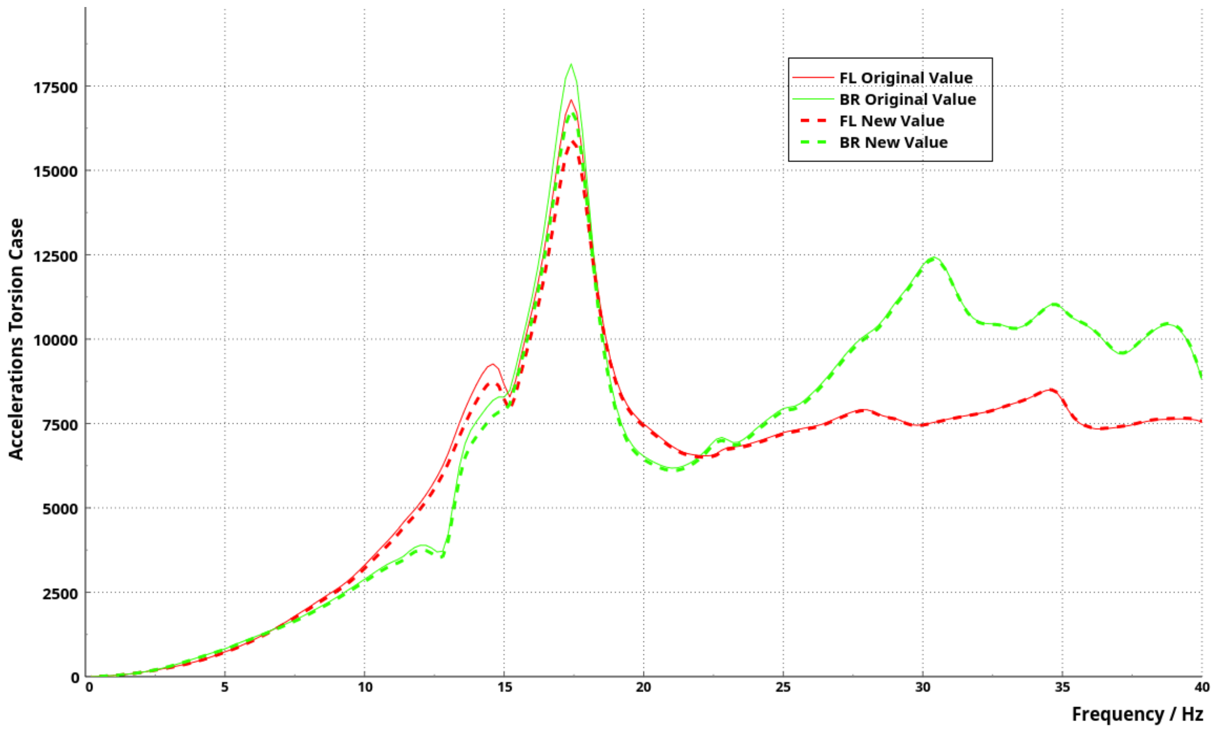

Figure 116: Torsion Analysis of Damping Coefficient Changes in the MP

11.5 Final Attempt

When we were on the part of the work in which we advanced more, we had to finish it because the semester was finishing. For that reason, we decided to approach the amplitude of the curves to the real ones. The shape was always the same in all our simulations so we just wanted to approach the amplitude. Because the goal of the project was to get a new model closer to the reality, we decided to check the curves in two points to do the results more reliable. So, we studied the curves in the A-Pillar point and in the Middle point of the wheels.

Each curve in these points had a significant peak. In the A-Pillar the peak had an amplitude of 22000 (mm/s^2) and in the Middle point of the wheel the peak increased to 23500. So, we wanted to obtain these amplitudes in the curves in our model.

To do this, and as we explained before, we could change 2 parameters, the force per velocity in the dampers and the stiffness in the springs of the wheels. Changing these two parameters the amplitude changed radically and we could approach to that one that we wanted. So, the previous day to finish the project we had done these five simulations that we can observe in the Figure 117 and Figure 118

Force per velocity	Wheel Stiffness $\frac{N}{mm}$	Amplitude $\frac{mm}{s^2}$
1, 175	250	11500
1, 875	280	17000
3,000	350	31000
4, 075	250	19500
4, 075	500	51000

Figure 117: Results of the Simulations in the Attempt to get the Real Amplitude in the A-Pillar

Force per velocity	Wheel Stiffness $\frac{N}{mm}$	Amplitude $\frac{mm}{s^2}$
1, 175	250	19600
1, 875	280	21150
3,000	350	30000
4, 075	250	17500
4, 075	500	38500

Figure 118: Results of the Simulations in the Attempt to get the Real Amplitude in the Middle point of the wheel

With these five attempts, we tried to guess the optimal values of the force per velocity and stiffness which that would lead us to the amplitudes of the acceleration that we wanted. That day we did not have any program to optimize the results so we used the solver of Microsoft Excel to get the results. These results were the following: Force per velocity of the damper: 2,387 (Nmm/s^2) Stiffness of the springs of the wheels: 303,58 (N/mm)

Finally, when we solved the NASTRAN file with these parameters, we got an amplitude of 22000 to the A-Pillar point and 24000 to the Middle point of the wheel. The result in the A-Pillar was perfect and the other case was increased just 500, so we decided that the results were close enough.

12 Conclusions

Arrived to the last part of our Master Thesis, we would like to sum up with some conclusions that we consider really important and significant about the all project.

The first thing that we would like to highlight is that from the beginning we find ourselves in a very autodidact way of learning. We are talking about that we had never worked before with the three software that we used. So since the first mont, we were afraid of not being able to learn in a good way this software, but we personally believe that it was a good practice of maybe what we are going to find in our future careers as engineers. So we learnt three software in three months by ourselves and that is a really good point and we are proud of that. And also, we learnt how to interpret the results with these new tools, and that is a full knowledge.

Talking more about the main project, we think that the simulations were too long, and it was a real problem not being able of doing more tests, just to have more data and it would have been really useful. So having more time, we would have been able of trying new changes in other parameters and parts of the car. We saw that there were multiple options.

The results that we got were acceptable comparing with the real ones. There was only two big inconvenience: the first one was related with the bending case. We were not able to get some good results compared with our colleague Mr. Krampe. It is a point that we will talk later in the future lines section. But it was complicated because with more good result from this case, we would have had more points and we could have been more effective trying to approach the real results.

The other great problem, as it was shown in the videos, was the roof. The results were always disturbed when we tested the closed model car. So at the end, we arrived to the conclusion that the roof was not perfectly modeled. We have to consider that it was made by many other components and the parts closed to the A-Pillar point did not represent very good the real car.

About the theoretical part of our project, we would to underline that in a dynamic analysis, it is really fundamental the modal analysis, just because after doing it, we understand better the behavior of our car, and later we can give some arguments to the results appeared in the graphics. And in a global meaning, FEM is also fundamental in the engineering world. There are many industries and applications that use normally this tool to test many parts or components.

To sum up with the conclusion, we personally believe that we learnt a lot of things about vibration, dynamic analysis and also about the different components in a car, because we have studied before some concrete parts of the car but not in this way, like a hole body and we think that it will be really useful for our future.

13 Future lines of our Project

In this section we would like to mention some future lines from our project that we consider interesting. We believe that after the period working with this software, we left the project in the moment when we were more able to understand better everything and when we had more ideas to improve our results.

The first future line is very clear. We would have liked to change more parameters in many other parts of our car body. Th reason was obvious, because after trying with only a few simulations, we were appreciating that the results were more or less acceptable. If we could have more time to do more simulations, we would have obtained better results and also a better validation of the real car.

Related with this line, we would have been interested in changing another parts from the car body, not just the dampers or the suspension system. It would have been a good option working more with the engine, because as we could have seen, it had an enormous influence in the vibrations of the car.

Referring more specifically to some parts of our car body, we would like to have done a better analysis of the roof. As we have checked along all the project, the roof had some problems that made us having some peaks in our results. To study better this points, we would have wished to have more time to do videos of this concrete peaks to watch better what happened there. But the problem was that this videos were costly time to do, because we had to print all the displacement of all the points from the car body, so it took a long time.

Another important point that we would like to improve is the bending case. we left the project without understanding very good why we were not able to get the same results as in the real car. We think that it was related maybe with the boundary conditions or with the input signal, so for, it would be an interesting point to solve.

In conclusion, we think that the most important future line that it should be taken into consideration is the time per each simulation, because as we have explained in these lines, if we would have had more time, we would have been able of doing more simulations, we would have got more test, more results and we could have approached better to our main goal in this project: getting the validation from our FEM model.

14 References

[1] ADAMS.VINCE/ASKENAZI.ABRAHAM *Building Better Products with Finite Element Analysis.OnWord Press.First Edition.1999*

[2] NEVILLE F. RIEGER *The Relationship Between Finite Element Analysis and Modal Analysis.Stress Technology Incorporated.Rochester.New York*

[3] HERMAN VAN DER AUWERAER *Structural Dynamics Modeling using Modal Analysis: Applications, Trends and Challenges.Technology Conference.LMS International.2001*

[4] MSC.SOFTWARE CORPORATION *Dynamic Analysis User's Guide.MSC Nastran 2013.1.2013*

[5] MSC.SOFTWARE CORPORATION *Getting Started with MSC Nastran User's Guide.MSC Nastran 2012.2012*

[6] MSC.SOFTWARE CORPORATION *Reference Manual.MSC Nastran 2004.2004*

[7] MSC.SOFTWARE CORPORATION *Quick Reference Guide.MSC Nastran 2013.1.1.2013*

Statement

We hereby certify that we have used our thesis written itself constantly and no other than the specified sources and aids.

Date: ....

(Sign)