# **POLITECNICO DI TORINO**

## Faculty of Engineering LM in Ingegneria Meccanica (Mechanical Design Engineering)

# **Master Thesis**

Numerical-experimental modal analysis of a steering wheel system



**Tutors:** 

Prof. Elvio Bonisoli Dott. Simone Venturini Dott. Domenico Lisitano Ing. Stefano Averame

> Candidate: *Ali Hijazi*

Politecnico di Torino, Department of Mechanical and Aerospace Engineering Corso Duca degli Abruzzi, 24 – 10129 Torino, Italy

2021-2022

# Ringraziamenti

Thanks God for everything he is giving to me.

Thanks for the Vehicle Mechanics research group for their work and efforts to fulfil this thesis.

I would like to thank my family for supporting my efforts and for their unlimited back up and extreme trust during all these years, thanks for their transmitted positive energy that is obviously not less important.

# Index

Ringraziamenti2
Index
Abstract5
Introduction6
1. Modal analysis7
2. Finite element method software
2.1 Hypermesh
2.1.1 Hypermesh model architecture9
2.1.2 Solver settings
2.1.3 Post processing of results
2.2 LUPOS
2.2.1 Post processing
2.3 LUPOS vs Hypermesh
2.4. Modal assurance criterion (MAC)
3. Steering wheel
3.1 Model without rubber
3.1.1 Optistruct results
3.1.2 Lupos results
3.1.3 Lupos vs Optistruct
3.2 Model with concentrated rubber mass
3.2.1 Optistruct results
3.2.2 Lupos results
3.2.3 Optistruct vs Lupos
3.3 Model with rubber
3.3.1 Optistruct results
3.3.2 LUPOS results
3.3.3 Results comparison Optistruct vs Lupos42
3.3.4 Model with rubber vs without rubber
4. Experimental modal analysis
4.1 Clamped steering wheel
4.1.1 Experimental – numerical post processing
4.1.2 Experiment Setup and sensor positioning
4.1.3 Geometrical aspects and acquisition parameters
4.1.4 Experimental data analysis
4.1.5 Post-processing
4.2 Free-free steering wheel
4.2.1 Experimental – numerical post processing81
4.2.2 Setup and sensor positioning
4.2.3 Experimental data analysis
4.2.4 Post-processing
4.3 Clamped Steering Wheel Second experiment103
4.3.1 Experimental – numerical post processing107
4.3.2 Experiment Setup and sensor positioning108
4.3.3 Geometrical aspects and acquisition parameters112
4.3.4 Experimental data analysis114
4.3.5 Post-processing
4.4 Steering Wheel models comparison123
4.5 Steering Plate free-free condition

4.5.1 Experimental – numerical post processing	
4.5.2 Experiment Setup and sensor positioning	
4.5.3 Geometrical aspects and acquisition parameters	
4.5.4 Experimental data analysis	
4.5.5 Post-processing	
4.6 Clamped plate	
4.6.1 Experimental – numerical post processing	
4.6.2 Experiment Setup and sensor positioning	
4.6.3 Experimental data analysis	
4.6.4 Post-processing	
5. Whole steering assembly	
5.1 Model configuration	
5.1.1 Bench	
5.1.2 Connectors	
5.1.3 Steering wheelbase	
5.1.4 Beams	
Conclusions	
Reference	

## Abstract

The creation of a suitable and accurate finite element model of a steering system in NVH and its experimental validation was understudy in this work. The steering in a car is critical. Steering influences handling and safety of car in operating and dynamic conditions that promotes it to be an important system. Hence in this thesis, further evaluations and applications of the experimental tests of steering-wheel and steering-column with car firewall were achieved pointing out a deep and comprehensive description of each phase, including the results of numerical modal analysis ran by different software of various steering-wheel models and other related components which they were under experiment in an experimental modal analysis (EMA). The final steps regarding the experimental tests correlating the FE models with the real component were performed, including the bench building in order to verify the performed analysis and to characterize the components behaviour.

### Introduction

The behaviour of steering wheel in automobile application is still a fair unframed issue in vehicle applications. One of the ways of detecting and clarifying the dynamic behaviour of a steering system is a modal analysis. Moreover, identifying a certain behaviour of any mechanical system using Finite Element Method (FEM) is an efficient way that has been performed and developed by means of numerical models and commercial software that are capable, if used wisely, of predicting the structure's future response for a certain input excitation. However, to validate and verify the obtained results experiment is always needed. The degree of compatibility can be evaluated by certain methods like MAC, that confirms the convergence of the theoretical solution towards the real one. As requested from IVECO, a NVH and modal analysis studies were carried out using the mentioned tools for a new top steering column, definitely to fulfil a certain level of comfort and control to avoid critical scenarios and vibrations of structure. Hence, finite element models were constructed performing a modal analysis that yields to a convenient solution and a summary of preliminary steps of experimental analysis were reported [1]. On the other hand, modifying a previous configuration in order to meet the new requirements is not an easy mission specially when the starting point is experimental. This research highlights the modal experimental analysis of the real behaviour of the entire system of a steering wheel, and firewall bench that verifies the numerical modal analysis and interprets the results to conclude future improvement and solutions for its structural design.

#### 1. Modal analysis

The steering system FEM model that was provided by the constructor was then analysed, manipulated, and developed by previous research done at PoliTO, nevertheless it is convenient to highlight certain software aspects that can be considered as a preliminary guide for a modal analysis. Modal analysis is an accurate prediction of a system behaviour identifying its response where a new modal coordinate system can be built and found leading to decoupling the ordinary differential equations that describe the system.

The following dynamic linear equation of the problem is considered in the study:

$$\mathbf{M}\ddot{\mathbf{x}} + \mathbf{C}\dot{\mathbf{x}} + \mathbf{K}\mathbf{x} = \mathbf{f}(t) \tag{1.1}$$

where **M** is the mass matrix and **K** is the stiffness matrix, the damping however was neglected since the damping ratio of steel is relatively low approaching by that the real case, but damping is heavily taken into account when dealing with the complete model where rubber layer is influencing.  $\mathbf{f}(t)$  is the vector of external forces that can be applied on the structure, and x is the displacement vector. When  $\mathbf{f}(t)$  is 0 we obtain a homogenous equation, without damping, that fairly describes the free behaviour of the system. Differently, by considering the rubber this equation is no more accurate because this non-structural mass has an evident impact on the behaviour of the structure due to its mass and the damping characteristics that it intensifies. The Automotive steering wheel system that has been studied in both configurations performing experimental modal analysis EMA showed the effect of the important rubber layer on damping the vibrations of the wheel. In the following research the experiment details are highlighted and analysed.

$$\mathbf{x} = \mathbf{T} \ \mathbf{x}_{BCs} = \mathbf{T} \ \mathbf{\Phi}_{BCs} \mathbf{\eta} = \mathbf{\Phi} \ \mathbf{\eta} \tag{1.2}$$

**x** represents the physical coordinates and dofs,  $\Phi$  is the modal transformation matrix that contains the modal vectors  $\eta$  however describes the principal modal coordinates with the modal dofs. This system shows the transformation matrix that is able to decouple the equations of motion giving opportunity to solve single dof systems. exploiting the mass and stiffness orthogonality property, then by solving the eigen value problem the natural frequencies are easily obtained

### 2. Finite element method software

Finite Element Method (FEM) is one of the most powerful tools that engineers adopt in order to simulate the physical phenomena in a numerical technique. It is used to analyse a prototype numerically reducing by that the usage of physical or real prototypes, consequently reducing the number of performed experiments. It helps in optimizing the studied component and developing its design efficiently, that reduces a lot of costs as well.

The system understudy is described by set of partial differential equations (PDE) that the computer is going to solve. Knowing that the real system has an infinite number of dof, the numerical model solved by FEM is stiffer than reality and the solution obtained is approximate and not exact since in numerical model there is a finite number of dof. In order to increase the level of accuracy certain skills of the user play a main role other than the mathematical knowledge the user must be able to apply a good mesh, since models' discretization has a main influence on the final results. Many FEM software are available today in market, but we decided to work on Hypermesh for meshing and on Optistruct and Lupos for solving.

#### 2.1 Hypermesh

Hypermesh is one of the tools exploited in order to perform a well discretized mesh. In particular Optistruct solver integrated in Hypermesh was adopted to solve and get the solution and the main steps are here reported.



Figure 2.1.1 – Hypermesh interface.

Once the geometry is imported, many aspects can be assigned to the model as seen from the interface in Figure 2.1.1 like material properties, element type, applied constraints, the wanted output and the type of analysis to be run after completely defining the model.

Modal analysis will give us a clear knowledge about the natural frequencies of the system and the modes of vibration, hence an efficient clue that helps in improving the design of the structure. But solvers require a deep knowledge by users especially about the engineering approach used by the computers in order to reach the desired results, hence it is important to shed light on the fact that FEM is an approximate method and not an exact one. Therefore, studying the elements available in the software and the way of selecting them is not less important since elements have also their influence on results.

Furthermore, an essential and important aspect is defining coherently the unit of measurement utilized in the analysis. For example, here the SI units can be implemented according to the specific application of the project. In this research length was considered in [m] mass in [kg] time in [s] and force in [N]. This is applicable by selecting the tools option then control cards DTI\_UNITS, these

units can be naturally modified according to a compatible unit system, and it is possible to manipulate these units by scaling them up or down according to the application in a coherent way.



Figure 2.1.2 – Unit Measurement selection in Hypermesh.

This can be done by going to the analysis option where you can select control cards where the image in Figure 2.1.2 shows the content.

#### 2.1.1 Hypermesh model architecture

In any model, rods and beams can be modelled as follows ROD element section type with a BEAM property type where "ROD doesn't say anything about 1D element type" [2], beams unlike rods that sustain only axial behaviour, allow for bending since they are characterized by more dofs although they can be assumed as mono-dimensional, and the connectors that grant a kinematic connection between nodes were modelled as tubular beam elements and this can be easily performed by the Hypermesh selecting the type and defining the corresponding parameters (such as thickness, density....). In our case we had steel and aluminium materials. To create any of the mentioned data it is sufficient to press a right click on the model browser as in Figure 2.1.1.1 and select the data of interest. Since in our model that, we are going to discuss about in details in chapter 4, we have a suitable geometry such as steering wheel frame, to use mostly beam elements that can be identified as PBEAML in Hypermesh or the PSOLID for the central part.



Figure 2.1.1.1 – Selecting card image and element section type from model browser.

Then a mesh must be created assigning the mesh type and size of elements. In critical zones where we can have stress concentration or an abrupt change of geometry near holes the mesh can be refined in order to get more correct and accurate results. From to the lower control panel selecting the 2D option where control mesh is possible.

planes	ruled	connectors	automesh	edit element	Geom
cones	spline	HyperLaminate	shrink wrap	split	] C 1D
spheres	skin	composites	smooth	replace	@ 2D
torus	drag		qualityindex	detach	0 3D
	spin		elem cleanup	order change	🔿 Analysis
	line drag		mesh edit	config edit	C Tool
	elem offset	midmesh	rebuild mesh	elem types	C Post

Figure 2.1.1.2 – Meshing in 2D option from control panel.

The mesh type and mesh elements can be defined after entering to automesh section the quad or tria elements as desired the selection is done by selecting the surfaces or elements to be meshed as

indicated in Figure 2.1.1.3. Furthermore, a mesh refinement can be performed also by mesh edit where the refinement is needed.



Figure 2.1.1.3 – Meshing in 2D option from control panel.

#### 2.1.2 Solver settings

Other than applying constraints and forces, In order to allow Optistruct to perform the analysis and obtain the results an EIGRL Real Eigenvalue Extraction Data can be applied, it is Lanczos method [3] that defines the data required to perform real eigenvalue analysis (vibration or buckling). This can be done by defining a new load collector selected from card images selecting EIGRL. V1

is the first value of frequency range, V2 is the last value of frequency range and ND is the number of modes that user wants to be visualized.

Name	ID 😵	nclude	
📲 😪 Beam Section Co	llectors (5)		
🖓 Cards (2)			
🗟 Components (14)			
📲 Load Collectors (	1)		
Denstrain	t 1	0	
- 🔂 Load Steps (1)			
Materials (3)			
Steel	1 🗖	0	
📲 Aluminium	2 📕	0	
Aluminium_ce	entral 3	0	
Parts (6)			
Properties (22)			
Titles (1)			
Name	Value		
Solver Keyword:	FICDI		
Name:	EIGRL		
	Constraint		
ID:	Constraint		
ID: Color:	Constraint		
ID: Color: Include:	Constraint 1 [Master Model		
ID: Color: Include: Card Image:	Constraint 1 [Master Model EIGRL		
ID: Color: Include: Card Image: Load Types:	Constraint 1 [Master Model EIGRL SPC(1)		
ID: Color: Include: Card Image: Load Types: User Comments:	EIGRL Constraint 1 [Master Model EIGRL SPC(1) Hide In Menu/E	¢port	
ID: Color: Include: Card Image: Load Types: User Comments: V1:	EIGRL Constraint 1 [Master Model EIGRL SPC(1) Hide In Menu/E 1.0	«port	
ID: Color: Include: Card Image: Load Types: User Comments: V1: V2:	EIGRL Constraint I [Master Model EIGRL SPC(1) Hide In Menu/E 1.0	¢port	
ID: Color: Include: Card Image: Load Types: User Comments: V1: V2: ND:	EIGRL Constraint 1 [Master Model EIGRL SPC(1) Hide In Menu/E 1.0	¢port	

Figure 2.1.2.1 – Load collector setup and EIGRL selection (25 imposed modes).

In case of a constrained analysis, it is possible to create another load collector as single point constraint SPC that represents a boundary condition as a bolt or adhesive joint where a kinematic equation is imposed in particular the SPC fixes a node without allowing for a displacement. These single point constraints have a distributed effect by using the rigid body elements (RBEs) created in components sector, and they practically represent the rigid motion of the structure or of two considered nodes with possible addition of the stiffness.

Then in Analysis create a load step to combine the load collectors' effects and create its name, after that in model browser at the left part of the screen the analysis type is defined as normal modes and the single point constraint are combined by the single point constraint load collector. Scrolling down, the output files can be selected and any other wanted output (displacement, ESE Element Strain Energy, ...).



Figure 2.1.2.2 – Load step setup.

#### 2.1.3 Post processing of results

After implementing all the needed entities correctly, it is possible to obtain the results by ordering the solver to do so, this is possible by selecting the analysis tab in the control panel and clicking on the solver name, in this case Optistruct, and if there are no errors results and output file can be visualized smoothly from the new window that appears after the software finish computation.

vectors	load types		interfaces	control cards	<ul> <li>Geom</li> </ul>
systems	constraints	accels		output block	C 1D
preserve node	equations	temperatures	entity sets	loadsteps	C 2D
	forces	flux	blocks		C 3D
	moments	load on geom	set segments	optimization	Analysis
	pressures		bodies		<ul> <li>Tool</li> </ul>
			nsm	OptiStruct	C Post

Figure 2.1.3.1 – Optistruct in bottom browser.

🛆 Whee	I.fem - Altair Compute Console Solver View				-		×
Solver:	optistruct_2021.2_win64.exe						
Input file:	Wheel.fem	Job com	pleted				
Run comm	nand:/hwsolver.tcl -solver OS -screen/Wheel.f	em					
Message I	og:		Optimi	zation summary:			Graph
Message	es for the job:	^	able	Grid/Elem	ID	Value	^
ANALYS	BIS COMPLETED.						
		~					
D			`				· ·
ANALYS	HANY: SIS COMPLETED.			Find:			^
==== E	and of solver screen output ====						
(	OptiStruct Job completed ====						
<							>
				Results	View	• C	lose

Figure 2.1.3.2 – Optistruct interface after completing analysis.

From the interface clicking on view button, it will be possible to view the input files that contain the nodes' number and coordinates and the type of elements and more details that were inserted previously, mode shapes and the output files that shows the details about each mode and about the elements as shown in Figure 2.1.3.3.

1	******	**	*****	**
	FINITE ELEMENT MODEL DATA INFORMATION :			
	matel 1 of cuide (Otwartwart)		2504	
	Total # of Grids (Structural)		3504	
	Total # of Bigid Blamants	2	2309	
	Total # of Rigid Elements	1	204	
	Total # of Rigid Element Constraints	2	1062	
	Total # of Degrees of Freedom	:	12036	
	(Structural)		0.400.00	
	Total # of Non-zero Stiffness Terms	:	348027	
	Element Type Information			
	CREAM Flements · 221			
	CHEVA Floments : 1000			
	CHEAR Elements . 1900			
	Load and Boundary, Information			
	Load and Boundary Information			
	SPC Sets 1			
	510 5005 . 1			
	Material and Property Information			
	naterial and rioperty information			
	PBEAM Cards : 21			
	PSOLID Cards : 1			
	MAT1 Cards : 3			

Figure 2.1.3.3 – Optistruct ouput file.

#### 2.2 LUPOS

LUPOS is a FEM code developed by DIMEAS department in POLITO and integrated in Matlab that follows the lumped parameter approach as it is the acronym of Lumped Parameter Open Source. It is able to integrate parametric analyses and non-linear process introducing reduction techniques and assembly approaches and many additional aspects.

"This software aims to emphasise these concepts from trivial examples to industrial cases starting from main concepts of dynamic modelling and simulations" [4]. Alternatively, the possibility of performing the FEM calculation of the same model requires the availability of the same input, to do so, it is possible to convert the input data from Hypermesh to LUPOS by a certain conversion from (.bdf) extension and this is achieved by a certain code as well.



Figure 2.2.1 – Test rig control panel interface.

The test rig control panel is a section in Lupos that permits the user to visualize and manipulate the component elements easily by selecting and unselecting as noticed in the figure. Once the model is selected and loaded this can be applied and the desired elements in different configurations can be visualized.

#### **2.2.1 Post processing**

Hence to visualize the model and see the elements type only, it's sufficient to use the test rig control panel that is a pre-processing of the model however, to apply the modal analysis graphical user interface of Lupos is used.

承 LUPOS Co	ontrol Panel			- 🗆 X	
	LUPOS				
Setup		Gui_SteeringWheel	hexa.m	Load	
Solution	Static	Dynamic	Dyn damped	Time	
	Prestressed	Buckling	Rotordyn	Time + MultiPhi	
Model	Model file	SteeringWheel_hexa.	Parametric	No parametric $\vee$	
	Param file		Parameter list	0	
Pre-proc			Load Model	TRCP	
Options	Parameter	Model repaired $\sim$	Value	$\checkmark$	
			Value	no use	
Solving	Graphics	On / Off	Close all	Run Simul	
Post-pro			Clear all	Save	
		XLSdata	PLOTdata	MCP	

Figure 2.2.1.1 – GUI interface of LUPOS.

The model then is selected then loaded on the setup section as noticed in figure 1.1.2.2.1 then solved by pressing on the Run simulation button. In order to solve the problem and be able to see the modes the Modal Control Panel (MCP) is run and open.

Once the MCP is open a control interface for visualizing and editing the modes and model elements appears.

承 Modal Control Pan	el		-	
		Modal Control Panel	ver. 202	21-06-16
Data	Sing $\vee$	Structure.m	Nodes	3504
0D 🗹 Str	ucture.geo	Structure.mss Struct Struct	0D	6
1D 🗹 Struc	t Struct	Struct Struct Struct Struct	1D	1905
2D 🗹 Struc	t Struct	Struct Struct	2D	none
3D 🗹 Struc	t Struct	Struct Struct Struct Struct	3D	1988
Solution 🗹 Eiger	nvectors	Phi.dat	DoF	21024
Real V Tal	. DoF	NodesPhi.dat	Modes	20
Close all Eige	nvalues	W2.dat	L	.oad
Graph xyz v	Fig	1000 0.75 0.4 0.3 Mode	Model	~
Hex contour Hex undef.	Scale	Colormap	Element o	olors ~
Reactions ✓ < >	Frame	4 • 1 GIF	•• ►	▶

Figure 2.2.1.2 – MCP interface of LUPOS.

In the bottom part of the panel there are the control actions by which the model visualization can be modified and the wanted mode can be selected. The following modes are visualized by MCP imposing the nodal displacement option with a deformed transparency.

#### 2.3 LUPOS vs Hypermesh

The most important aspect in LUPOS is that it's able to represent the same model with the name nodes and same elements, giving the possibility to have a well approximated calculation and reasonable results where elements can be visualized and modified easily and efficiently with respect to Hypermesh. In the figure below it is evident how much are they compatible where same entities, nodes and elements are demonstrated.



Figure 2.3.1 – Model nodes without rubber in Hypermesh and in Lupos respectively

In the previous figure it is noticed how much are comparable the two models in different software having identical geometry and very similar discretization. Later on, it is possible to perform a Modal Assurance Criterion (MAC) to check the compatibility of both software and the reliability of the model.

#### 2.4. Modal assurance criterion (MAC)

This numerical and experimental coupling is very fruitful combination where a clearer understanding about the behaviour can be fished. This was also done through the implemented tools presented in Lupos

The modal assurance criterion (MAC) is an index that indicates the compatibility consistency between two series of n eigen vectors [3]. It useful to perform a comparison between experimental modes and numerical modes or between numerical and numerical.

$$MAC_{j,k} = \frac{\left[\boldsymbol{\Phi}_{j}^{T}\boldsymbol{\Phi}_{k}\right]^{2}}{\left[\boldsymbol{\Phi}_{j}^{T}\boldsymbol{\Phi}_{j}\right]\left[\boldsymbol{\Phi}_{k}^{T}\boldsymbol{\Phi}_{k}\right]} = \cos^{2}\alpha_{j,k}$$
(2.4.1)

The square based vector helps to evidence the higher differences without being sensitive to small ones. The output of MAC is a *nxn* matrix having a value between 0 that indicates no consistency and 1 that means a complete consistency and compatibility, moreover  $\alpha_{j,k}$  is the angle between the inner product of the considered eigenvectors.

$$MACW2_{j,k} = \frac{\left[\mathbf{\Phi}_{j}^{T}\mathbf{\Phi}_{k}\right]^{2}}{\left[\mathbf{\Phi}_{j}^{T}\mathbf{\Phi}_{j}\right]\left[\mathbf{\Phi}_{k}^{T}\mathbf{\Phi}_{k}\right]} e^{-k\sqrt{2}\left|\frac{\omega_{j}^{2}-\omega_{k}^{2}}{\omega_{j}^{2}+\omega_{k}^{2}}\right|}$$
(2.4.2)

Equation (2.4.2) takes into consideration the eigenvalue contribution in order to avoid an illusion of having the same eigenvectors with different eigenvalues.

### 3. Steering wheel

One of the cases that will be understudy in this thesis is the Steering Wheel, the study is going to analyse experimentally a steering wheel system. A Steering Wheel is an essential component in vehicles that affects the entire dynamic behaviour of a car, so it is important to guarantee a good level of comfort, stability, and safety of it. The Seering Wheel was provided by Iveco group and was exposed to numerical and experimental modal analysis.



Figure 3.1 – Steering Wheel.

The Steering Wheel design was improved by a chain of manipulations in the numerical phase. Firstly, the system was analysed without the rubber layer that is a non-structural mass (NSM) and some unnecessary components were removed as the central plate because it has no structural influence leaving the internal part of the Steering Wheel that is made up of steel and aluminium.

#### 3.1 Model without rubber

Initially the model was analysed without considering any contribution of the rubber that surrounds the wheel frame. Hence it is important to emphasize the fact that no inertial characteristics of the rubber were considered at all, and this leads to expect higher frequencies of this model with respect to the other model that has rubber.



Figure 3.1.1 – Wheel model in Hypermesh.

The central part in red shown in Figure 3.1.1 is the aluminium component in the wheel, and the grey elements compose the steel frame. where the total mass of the correspondent structure is 2.343 kg to be verified with the real object when the rubber layer is released out from the wheel.

#### **3.1.1 Optistruct results**

In this subchapter the results obtained by Optistruct of the model with no considered rubber or its inertial contribution.

Mode #	Frequency [Hz]
1	56.32
2	56.88
3	84.42
4	87.07
5	135.05
6	144.66
7	187.44
8	344.35
9	351.58
10	371.20

Table 3.1.1.1 – Model without rubber, natural frequencies in Optistruct.



Figure  $3.1.1.1 - \text{Optistruct } 1^{\text{st}} (56.32 \text{ Hz}) \text{ and } 2^{\text{nd}} (56.88 \text{ Hz}) \text{ modes respectively.}$ 



Figure 3.1.1.2 – Optistruct  $3^{rd}$  (84.42 Hz) and  $4^{th}$  (87.07 Hz) modes respectively.



Figure 3.1.1.2 – Optistruct 5<sup>th</sup> (135.05 Hz) and 6<sup>th</sup> (144.6 Hz) modes respectively.



Figure 3.1.1.2 – Optistruct 7<sup>th</sup> (187.44 Hz) and 8<sup>th</sup> (344.3 Hz) modes respectively.



Figure 3.1.1.2 – Optistruct 9<sup>th</sup> (351.1 Hz) and 10<sup>th</sup> (371.2 Hz) modes respectively.

#### 3.1.2 Lupos results

Lupos frequencies are very similar to those in Optistruct according to an analogous model without rubber that was analysed.

Mode #	Frequency [Hz]
1	54.23
2	63.31
3	85.61
4	90.53
5	139.3
6	146.2
7	187.7
8	348.6
9	356.1
10	374.6

Table 3.1.2.1 – Model	without rubber,	natural free	quencies in Lup	oos.



Figure  $3.1.2.1 - LUPOS \ 1^{st}$  and  $2^{nd}$  modes respectively.



Figure  $3.1.2.2 - LUPOS 3^{rd}$  and  $4^{th}$  modes respectively.



Figure  $3.1.2.3 - LUPOS 5^{th}$  and  $6^{th}$  modes respectively.



Figure 3.1.2.4 – LUPOS 7<sup>th</sup> and 8<sup>th</sup> modes respectively.



Figure 3.1.2.5– LUPOS 9<sup>th</sup> and 10<sup>th</sup> modes respectively.

It can be noticed that the most displaced nodes are those that are more distant from the bars connected to the central part. Moreover, modes are identical in both numerical software this indicates the reliability and accuracy of them.

#### **3.1.3 Lupos vs Optistruct**

Mode #	<b>Optistruct</b> [Hz]	Lupos [Hz]
1	56.32	54.23
2	56.88	63.31
3	84.42	85.61
4	87.07	90.53
5	135.05	139.26
6	144.66	146.15
7	187.44	187.65
8	344.35	348.59
9	351.58	356.13
10	371.20	374.21

Table 3.1.3.1 – Model without rubber, Optistruct and Lupos natural Frequencies.

Despite the slight difference in the frequencies but again, same modes are noticed with different number according to each software. For example, the 9<sup>th</sup> mode (344.35 Hz) in Optistruct, as in Figure 3.1.1.5, is coincident with the 8<sup>th</sup> mode (348.6 Hz) in Lupos as Figure 3.1.2.4 shows. This slight difference is devoted to the different approaches adopted by both software.

#### 3.2 Model with concentrated rubber mass

The steering wheel was modelled benefiting from different elements provided by the software. Mainly the elements used were beam and rod, for the external and internal part of wheel. On the other hand, the central part was particularly modelled by solid elements as noticed the difference in The rubber layer in the first model was not considered in order to get a light model, only its inertial contribution have been taken into account, as the low stiffness properties of the rubber should not be so relevant. Hypermesh allows to add non-structural masses, expressed as mass for unit of length Figure 3.1.1. Hence the rubber was lumped and distributed on among structure to reserve the inertial properties where lower frequencies are expected.



Figure 3.2.1 – Wheel Model without rubber in Hypermesh and in Lupos respectively.

In particular in the left photo the blue elements represent the external wheel frame, the yellow represents the internal wheel frame, the orange represents the connectors, and the red shows the central solid meshed part, and small red elements represent the T-connectors. In order to get a more precise result, the meshing of particular components requires more or different elements or both so that the shape functions that will be interpolated by the software will be of higher order, hence a more convergent solution toward the exact one. Table 3.2.1 resumes the details of element properties.

Component	Property	Material			
-	Туре	Туре	Density [kg/m <sup>3</sup> ]	Young module [GPa]	Poisson coefficient [-]
Wheel external frame	PBEAML	MAT1	7800	210	0.3
Wheel internal frame	PBEAML	MAT1	7800	210	0.3
T-connectors	PBEAML	MAT1	2790	75	0.3

external side					
T-connectors internal side	PBEAML	MAT1	2790	75	0.3
Supports	PBEAML	MAT1	2790	75	0.3
Central part	PSOLID	MAT1	2790	75	0.3
Shaft seating	PBEAML	MAT1	7800	210	0.3

Table 3.2.2 – Geometrical properties of model elements.

Component	Section			
	Tumo	External diameter [m]	Internal diameter [m]	
-	Type	(1 <sup>st</sup> BAR Dimension)	(2 <sup>nd</sup> BAR dimension)	
Wheel external frame	ROD	0.011	-	
Wheel internal frame	ROD	0.008	-	
	ROD	0.010	-	
T-connectors external side	TUBE	0.017	0.011	
T-connectors internal side	TUBE	0.017	0.010	
	TUBE	0.014	0.008	
Supports	TUBE	0.010	0.004	
	TUBE	0.014	0.008	
	TUBE	0.017	0.004	
	ROD	0.017	-	
	TUBE	0.016	0.010	
	TUBE	0.012	0.004	
	TUBE	0.010	0.006	
	BAR	0.012	0.005	
	BAR	0.005	0.005	
Shaft seating	TUBE	0.031	0.026	
	TUBE	0.031	0.024	
	TUBE	0.031	0.022	
	TUBE	0.034	0.022	
	TUBE	0.029	0.022	

The model boundary condition at the beginning was imposed considering a sealing connection at the center of the wheel and this is shown in the following figure



Figure 3.2.1 – Wheel constraint in front and top views respectively (Hypermesh).

It can be noticed that the sealing (black component) was imposed as the constraint with the ground. In particular, the sealing was modelled as tube beam section with different diameters where the minimum radius was 11 mm, and the maximum radius was 17 mm leading to a high stiffness this tube element is connected to the central part of the wheel by rigid body elements (RBEs). Moreover, the considered dimensions are not geometrically convenient with the wheel. The analysis was run any way and the results are explained below. This constraint was initially adopted in both models with and without rubber including that with NSM.

#### **3.2.1 Optistruct results**

With the material properties mentioned previously and following the same procedure explained in chapter 3.1.1 the analysis was run and the following results for the first 10 modes were obtained.

Mode #	Frequency [Hz]
1	45.80
2	46.22
3	68.30
4	69.98
5	109.4
6	116.4
7	150.3
8	278.2
9	280.0
10	296.7

Table 3.2.1.1 – Model with NSM, natural frequencies in Optistruct.



Figure  $3.2.1.1 - 1^{st}$  (45.804 Hz) and  $2^{nd}$  (46.219 Hz) modes respectively.



Figure  $3.2.1.2 - 3^{rd}$  (68.3 Hz) and  $4^{th}$  (69.98 Hz) modes respectively.



Figure  $3.2.1.3 - 5^{\text{th}}$  (109.38 Hz) and  $6^{\text{th}}$  (116.42 Hz) modes respectively.



Figure  $3.2.1.4 - 7^{\text{th}}$  (150.3 Hz) and  $8^{\text{th}}$  (278.2 Hz) modes respectively.



Figure  $3.2.1.5 - 9^{\text{th}}$  (280.04 Hz) and  $10^{\text{th}}$  (296.73 Hz) modes respectively.

#### 3.2.2 Lupos results

Mode #	Frequency [Hz]
1	43.57
2	50.25
3	67.32
4	70.1
5	109.4
6	114.8
7	148.9
8	266.2
9	278.4
10	296.8

Table 3.2.2.1 – Model with NSM, natural frequencies in Lupos.



Figure  $3.2.2.1 - 1^{st}$  and  $2^{nd}$  modes respectively.



Figure  $3.2.2.2 - 3^{rd}$  and  $4^{th}$  modes respectively.



Figure  $3.2.2.3 - 5^{\text{th}}$  and  $6^{\text{th}}$  modes respectively.



Figure  $3.2.2.4 - 7^{\text{th}}$  and  $8^{\text{th}}$  modes respectively.



Figure  $3.2.2.4 - 9^{th}$  and  $10^{th}$  modes respectively.

#### **3.2.3 Optistruct vs Lupos**

Mode #	Optistruct	Lupos
1	45.80	43.56
2	46.22	50.24
3	68.30	67.31
4	69.98	70.08
5	109.4	109.4
6	116.4	114.7
7	150.3	148.9
8	278.2	266.1
9	280.0	278.3
10	296.7	296.8
11	403.71	397.37
12	407.07	402.52

Table 3.2.3.1 – Model with NSM, Optistruct and Lupos natural frequencies.

The compatibility of both software is relatively high even if some modes number in one software may correspond to different mode number in the other software. For example, in Table 3.1.3.1 the 9<sup>th</sup> mode of Optistruct corresponds to the 8<sup>th</sup> mode of Lupos. and vice versa.



Figure 3.2.3.1 – Model with NSM, MAC between Optistruct and Lupos.

#### 3.3 Model with rubber

Adding non-structural mass can cause considerably a relevant increase in the inertia of the system and a damping influence on vibrations. Moreover, as noticed in the following sections, a certain increase in the natural frequency amplitude is noticed.

Table 3.3.1 – Rubb	er physical	properties.
--------------------	-------------	-------------

ρ, Density [kg/m <sup>3</sup> ]	434
E, Young module [MPa]	5

v, Poisson coefficient [-]	0.49

The estimated values for the real system in the table will be verified in the experimental analysis.



Figure 3.3.1 – Wheel model with rubber layer.

The figure above shows the construction of the model with rubber layer modelled by hexahedron elements adopting the Minecraft project that aims to simplify the solid mesh by the simpler solid element, which is the cube, and the influence of this idea is to reduce the nodes number and elements number as well and to facilitate the calculation process and its time. However, adopting this procedure causes an important increase in the numerical calculated mass of structure so a change of density was necessary to compensate.

	Table 3.3.2 – Comp	parison between	new and old	wheel	configuration.
--	--------------------	-----------------	-------------	-------	----------------

-	Old central part	New central part	Old rubber layer	New rubber layer
Number of nodes [-]	12064	3168	17538	13727
Number of elements [-]	53271	1988	76568	8915
$\rho_r$ , Real density [kg/m <sup>3</sup> ]	2790	2790	434	434
<b>M</b> <sub>r</sub> , Real mass [kg]	0.460	0.693	1.614	1.981
$\rho_n$ , Adjusted density [kg/m <sup>3</sup> ]	-	1853	-	353.5
M <sub>n</sub> , New mass [kg]	-	0.460	-	1.614



Figure 3.3.2 – Modelled and real steering wheel mass comparison.

In the previous figure an approximate equal mass was obtained, and this again proves the reliability of the model and the calculations done with the explained comprised simplification and procedure that were followed.

#### **3.3.1 Optistruct results**

In the following section results obtained by Optistruct solver, where the first 10 structural modes are reported in a way similar to the same procedure explained before in performing a modal analysis. View the modes is possible by selecting the .h3d file after resolving the model by Optistruct then by choosing the wanted mode from the browser panel at the left part of the screen with the addition of other control methods available in the control panel at the bottom.

Table 3.3.1.1 – Model with rubber, natural frequencies in Optistruct.

Mode #	Frequency [Hz]
1	45.45
2	47.11
3	69.96
4	73.30
5	110.0
6	115.3
-	142.9
7	149.7
-	214.1

-	226.7
-	231.2
-	249.0
-	252.4
8	272.9
9	280.4
-	281.0
-	285.5
10	289.3
	300.54
	310.61



Figure  $3.3.1.1 - 1^{st}$  (45.45 Hz) and  $2^{nd}$  (47.11 Hz) modes respectively.




Figure  $3.3.1.2 - 3^{rd}$  (69.96 Hz) and  $4^{th}$  (73.3 Hz) modes respectively.

Figure  $3.3.1.3 - 5^{\text{th}}$  (110.0 Hz) and  $6^{\text{th}}$  (115.3 Hz) modes respectively.



Figure  $3.3.1.4 - 7^{\text{th}}$  (149.7 Hz) and  $8^{\text{th}}$  (272.9 Hz) modes respectively.



Figure  $3.3.1.5 - 9^{\text{th}}$  (280.4 Hz) and  $10^{\text{th}}$  (289.3 Hz) modes respectively.

#### **3.3.2 LUPOS results**

The rubber layer as Figure 3.3.2.1 shows that the layer is attached to the wheel by means of RBEs that can physically be the adhesive material. It is more important to emphasize the fact that this layer is affecting fairly the behaviour of system since this non-structural mass can add an important inertial contribution but also a damping one.



Figure 3.3.2.1 – Wheel with and without the RBEs respectively.

Lupos					
Mode #	Frequency [Hz]				
1	49.60				
2	56.59				
3	72.65				
4	80.99				
5	117.6				
6	122.1				
-	142.3				
7	151.8				
-	219.6				
-	229.3				
-	233.0				
-	256.5				
-	259.7				
-	277.0				
8	277.5				
-	285.3				
-	290.3				
9	292.4				

Table 3.3.2.1 – Model with rubber, natural frequencies in Lupos.

The modes presented by the model in Lupos are presented as follows:



Figure  $3.3.2.2 - 1^{st}$  and  $2^{nd}$  modes respectively.



Figure  $3.3.2.3 - 3^{rd}$  and  $4^{th}$  modes respectively.



Figure  $3.3.2.4 - 5^{\text{th}}$  and  $6^{\text{th}}$  modes respectively.



Figure 3.3.2.5 – 7<sup>th</sup> and 8<sup>th</sup> modes respectively.



Figure  $3.3.2.6 - 9^{\text{th}}$  mode.

Despite the difference between the two evolved software the between some mode shapes it is noticed also the compatibility between them. In particular, the 10<sup>th</sup> (298.3 Hz) structural mode represented by Optistruct Figure 3.3.1.5 corresponds to the 9<sup>th</sup> (292.4 Hz) structural mode evaluated by Lupos Figure 3.3.2.6 similarly the 9<sup>th</sup> (280.4 Hz) mode corresponds to the 8<sup>th</sup> (177.5 Hz) respectively this implies that there is an accepted convergence between the two results and shows an excellent reliability of Lupos with the addition the same mode shapes and wheel parts are involved in each relevant mode. Where some small differences may be devoted to the different computational methods adopted by the two software.

### 3.3.3 Results comparison Optistruct vs Lupos

(	Optistruct	Lupos			
Mode #	Frequency [Hz]	Mode #	Frequency [Hz]		
1	45.45	1	49.60		
2	47.11	2	56.59		
3	69.96	3	72.65		
4	73.30	4	80.99		
5	110.0	5	117.6		
6	115.3	6	122.1		
-	142.9	-	142.3		
7	149.7	7	151.8		
-	214.1	-	219.6		
-	226.7	-	229.3		
-	231.2	-	233.0		
-	249.0	-	256.5		
-	252.4	-	259.7		
8	272.9	-	277.0		
9	280.4	8	277.5		
-	281.0	-	285.3		
-	285.5	-	290.3		
10	289.3	9	292.4		
_	300.54	-	303.3		
-	310.61	-	322.3		
-	312.70	-	323		

Table 3.3.3.1 – Model with rubber, Optistruct and Lupos natural frequencies.

# 3.3.4 Model with rubber vs without rubber

Table 3.3.4.1 – Models with and without rubber respectively, natural frequencies in Optistruct.

W	Vith rubber	Without rubber			
Mode #	Frequency [Hz]	Mode #	Frequency [Hz]		
1	45.45	1	56.32		
2	47.11	2	56.88		
3	69.96	3	84.42		
4	73.30	4	87.07		
5	110.0	5	135.05		
6	115.3	6	144.66		
-	142.9	-	-		
7	149.7	7	187.44		
-	214.1	-	-		
-	226.7	-	-		
-	231.2	-	-		
-	249.0	_	_		
-	252.4	-	-		
8	272.9	8	344.35		



Figuree 3.3.4.1 – Models with and without rubber MAC and MAC\_distW respectively.

It is clear that the models have the mode shapes but the frequencies are not, this is due to the nonstructural mass of rubber, the has an inertial contribution but not a structural one, thus frequencies will decrease.

	Frequency [Hz]							
Mode #	Tetra4 no rubber	Hexa8 no rubber	Tetra4 with rubber*	Hexa8 with rubber*	Hexa8 rubber NSM*	Tetra4 rubber NSM		
1	55.39	56.32	44.55	45.45	45.80	44.87		
2	58.06	56.88	46.89	47.11	46.22	46.22		
3	85.35	84.42	68.67	69.96	68.30	67.57		
4	87.17	87.07	70.72	73.30	69.98	68.02		
5	138.24	135.05	109.72	110.0	109.4	109.0		
6	144.24	144.66	113.13	115.3	116.4	114.9		
7	188.87	187.44	149.43	149.7	150.3	150.4		
8	349.46	344.3	272.33	272.9	278.2	277.0		
9	361.07	351.1	279.78	280.4	280.0	281.2		
10	372.83	371.2	291.08	289.3	296.7	296.8		

Table 3.4.3.2 – Various models, natural frequencies in Optistruct.

In Lupos however, the following results were obtained:

	Frequency [Hz]						
Mode #	Tet no rubber	Hex no rubber	Tet with rubber	Hex with rubber*	Hex rubber CONM*		
1	53.61	54.23	56.18	49.60	43.57		
2	62.84	63.31	61.78	56.59	50.25		
3	84.72	85.61	74.81	72.65	67.32		
4	89.23	90.53	84.69	80.99	70.1		
5	138.3	139.26	126.4	117.6	109.4		
6	144.3	146.15	128	122.1	114.8		
7	186.9	187.65	156.3	151.8	148.9		
8	347.6	348.59	281.7	277.5	266.2		
9	350.4	356.13	304	292.4	278.4		
10	-	374.21	-	-	296.8		

Table 3.4.3.3 – Various models, natural frequencies in Lupos.

# 4. Experimental modal analysis

In order to validate the results of natural frequencies and mode shapes carried out by the FE model and Experimental Modal Analysis (EMA) is performed. EMA is a very efficient tool to check the accuracy of the studied model and to comprehend the dynamic behaviour of the structure.

Figure 4.1.1 shows the machine on which the steering wheel will be mounted for the first experimental analysis. This machine is characterized by high inertia where its mass is about 2 tons equipped by soft springs that guarantee a low frequency vibration that are far away from the frequencies of steering wheel and unreadable by the accelerometer in this way the correct and pure acquisition is assured avoiding any external influence and uncoupling of the test rig machine from the internal dynamic of the structure.

### 4.1 Clamped steering wheel

Moreover, this massive machine having this high inertia is used for fatigue experiments, where it guarantees a good clamping boundary condition that near to ideal one as in the case below.



Figure 4.1.1 – Test rig machine and experiment environment.



Figure 4.1.2 – Clamped SW on test rig machine.

## 4.1.1 Experimental – numerical post processing



Figure 4.1.1.1 – Experimental FRF Sum diagram.

The real frequencies, eigenvectors and eigen values of the experimental test were extracted from the testlab acquired data and by navigating the cursor into the peaks of the sum of the frequency response functions (FRFs) that were obtained. The selection was done also taking into consideration the mode shapes since this model has, due to the rubber layer, presents plenty of local modes that are not considered to be structural, then the following results reported below.

	Testlab	Lupos		
Mode #	Experimental	Mode #	Numerical	
	frequency [Hz]		frequency [Hz]	
1	43.411	1	49.60	
2	45.283	2	56.6	
3	68.757	3	72.66	
-	-	-	81.01	
4	104.730	4	117.6	
5	110.508	5	122.1	
-	-	-	142.3	
6	157.448	6	151.8	
-	-	-	219.6	
-	-	_	229.3	
-	-	_	233.0	
-	-	-	256.5	
-	-	-	259.7	
-	-	-	277.0	
-	-	-	277.5	
-	-	-	285.3	
-	-	-	290.3	
7	291.566	7	292.4	
_	-	-	303.3	
-	-	-	322.3	
-	-	-	323	
-	-	-	354.5	
_	-	-	357	
-	-	-	360.3	
_	-	-	365	
-	-	-	369.9	
-	-	-	374.2	
-	-	-	376.1	
-	-	-	381.2	
-	-	-	384.5	
-	-	-	394.4	
8	410	8	411	
-		-	415.3	
-	-	-	423.2	
-	-	-	430.9	
-	-	-	438.4	
-	-	-	442	
-	-	-	443.9	
-	-	-	456.6	
9	425.1	9	466.1	
-	-	-	468.5	
-	-	-	473.7	

Table 4.1.1.2 - Comparison between experimental and numerical Frequency results

Although, the obtained MAC graph has some discrepancy that leads to think about optimizing the numerical model. In particular the numerical model will be modified by modifying the stiffness since the inertial contribution is already at a very good point.



Figure 4.1.1.2 – MAC representation between experimental and numerical data.

It is clear that 1<sup>st</sup> and 5<sup>th</sup> modes are compatible, the 7<sup>th</sup> experimental mode is coherent with the 8<sup>th</sup> numerical mode. The discrepancy in modes 3, 4 and 5 may be due to the rubber local modes or because the boundary condition that was imposed in the numerical model was extremely high or due to an error in the identification of the experimental results while extracting it from Testlab. To get rid of these doubts several steps were done.



Figure 4.1.1.3 – MAC\_dist representation between experimental and numerical data.

The MAC\_dist representation shows the correlation between the experimental and the numerical sets of eigenvectors with proportional distance of the related natural frequencies as shown in the previous figure. It seems from the neighbourhood between isoline and the MAC indication that the  $5^{th}$  numerical mode refers to the  $5^{th}$  experimental mode since it is nearest to the isoline. Unfortunately, the doubts regarding the  $2^{nd}$ ,  $3^{rd}$  and  $4^{th}$  mode are still storming in mind.



Figure 4.1.1.4 – Auto MAC representation of experimental data.

In Figure 4.2.3.3 clears out the doubt that there was double selected mode during the identification since in the FRF figure only one peak is noticed, hence, the same mode was chosen two times. To clear up any doubts a re-identification process was performed, and the results were re-analysed.



Figure 4.1.1.5 – MAC between FEA and EMA after re-identification.

Firstly, it can be noticed that the dispersion in modes 2, 3 and 4 is demolished but it is still needed to detect the real relation between 4<sup>th</sup> EMA mode and the 7<sup>th</sup> FEA mode, looking again to 7<sup>th</sup> numerical mode shape, it is important to emphasize the fact that this mode is local and it is related exclusively to the rubber. This kind of dispersion that may be called also spatial aliasing, may be due to the low number of nodes taken during the experiment, so the isoline is again needed to verify this. It is also important to note that the 4<sup>th</sup> numerical mode does not have any correspondence, and this is because this mode is torsional acting around the wheels axis as obtained numerically, while the experiment was done by hammering only radially the structure this justifies the fact this mode is not detected experimentally.



Figure  $4.1.1.6 - MAC_{dist}$  of the second identification.

Furthermore, it is convenient to confirm this check by plotting the MACW2 that distinguishes not only the eigenvectors but also compares eigenvalues, and this is great advantage in case there are similar eigenvectors while eigenvalues are not.



Figure 4.1.1.7 – MACW2 after re-identification.

And this confirms that what was concluded before that  $4^{th}$  EMA mode corresponds exclusively to the  $5^{th}$  FEA mode.

Finally, to avoid any ambiguities the auto-MAC is replotted



Figure 4.1.1.8 – Auto MAC after re-identification.

Here again, the 2x2 sub matrix noticed up right the figure, is probably a spatial aliasing in these high frequency modes since there were not enough experimental nodes, to measure and uniquely identify the concerned mode, hence nodes number was lower than what was needed but this is not a big issue since it was detected and highlighted here in figure indicating also that a good identification was performed.

#### 4.1.2 Experiment Setup and sensor positioning

The analysis is performed to exercise the experimental methodologies i.e. definition of experimental setup, construction of the same, setting of the LMS.Testlab environment for impact testing. In the second part, the identification of mode shapes is performed focussing on stabilisation diagram, stable point recognition and extraction of data from LMS.Testlab environment for data manipulation in common programming languages such as Matlab®.

In conclusion, post-processing was performed: CompleX Modal Assurance Criterion (MACX) for checking goodness of experimental activity and comparison with a preliminary Finite Element (FE) model was understudy as well.

Interestingly the experimental and numerical models show a good compatibility, confirming though the model validation. This steering wheel is used for Iveco truck vehicles was modelled with different elements in different software and there was a great convergence to the exact solution, and this was assured by the EMA.

The experimental modal analysis is performed adopting free-free conditions, exciting the structure with roving hammer and evaluating the response through accelerometers, either mono-axial or triaxial. The steering wheel is fixed on a huge test rig machine supplied by soft springs that do not affect the component natural frequencies the excitations were done on 19 nodes along all the possible directions allowed by its geometry, obtaining as an overall 33 excitations to the system. The responses of 3 nodes are measured: 1, 8, and 11. The response for each node is averaged on a set of 5 responses acquired to give more repeatability and stability to each measurement. For each response:

• the spatial coordinates are continuous for each instant of time.

- the time/frequency data and the modal content are acquired and identified with Siemens Test.Lab v.17.
- it evinces those modal coordinates enable approximate solutions, according to the number of modal coordinates used, thus in nonlinear problems this property can improve computational efforts in the best way.



Figure 4.1.2.1 – Experimental setup scenario.

In order to have a clear idea about sensor positioning, the suitable place where to allocate sensors can be performed also by Modal and Geometrical Selection Criterion, MoGeSeC [6], that is an efficient tool based on both geometry and modal properties of the system, obtained by a numerical modal analysis, for choosing the best representative nodes. The concept behind this technique is that the modal behaviour of a model can be represented by a list of nodes, whose eigenvectors resume the modal properties of the whole system. The progressive optimal location is based on both modal independence information and geometrical location to distribute accelerometers on the whole structure.

The Selection Criterion is based on the evaluation of the maximum value of the vector w:

$$\mathbf{w} = \mathbf{diag}(\mathbf{w}_{g}\mathbf{w}_{m}^{T}) \tag{4.1.2.1}$$

that represents the combination of the geometrical vector and the modal vector weights, calculated for each node, as follows:

$$\mathbf{w}_{g,p} = \frac{1}{\max(\mathbf{w}_g)} \frac{1}{\sum_{i=1}^{r} \left[ \frac{1}{(x_p - x_i)^2 + (y_p - y_i)^2 + (z_p - z_i)^2} \right]^{k_1}}$$
(4.1.2.2)  
$$\mathbf{w}_{m,p} = \frac{1}{\max(\mathbf{w}_m)} \sum_{j=1}^{3(r+1)} \left[ \mathbf{\Phi}_{p,x}^{(j)^2} + \mathbf{\Phi}_{p,y}^{(j)^2} + \mathbf{\Phi}_{p,z}^{(j)^2} \right]^{k_2}$$
(4.1.2.3)

Coefficients  $k_1$  and  $k_2$  allow to weigh the influence of each component; start value are respectively 2 and 1.

The result of this selection is a list of nodes positioned as far as possible on the structure, with a homogeneous distribution, in order to have the best possible representation of the vibrational behaviour.



Figure 4.1.2.2 – Model with rubber, MoGeSec results in Lupos.

Both pictures of the entire system, with different points of view and details of the hardware implemented and of their location are requested information.

Importance has the reference system (x, y, z) used, according also to suitable models. In case of already developed models, consider the same reference system.



Figure 4.1.2.3 – Overview of the experimental setup.



Figure 4.1.2.4 – Details of the implemented hardware, from left to right in reading order: Tri-axial-node 1, Tri-axial-node 8, Tri-axial-node 11.

The acquisition is performed with a LMS SCADAS Mobile system. A roving hammer EMA is performed using an impact hammer PCB 086C03 using hard steel tip (Figure 1.1.3 and Figure 1.1.4) to reach a good frequency identification range of the structure. A series of 3 hammer repetitions is linearly averaged. The characteristics are listed in Table 1.1.1.



Figure 4.1.2.5 – Impact Hammer PCB Model 086C03.



Figure 4.1.2.6 – Hammer stiff steel tip.

Table 4.1.2.1 -	Impact hammer	characteristics.
-----------------	---------------	------------------

Component	Characteristics	Value
	Sensitivity [mV/N]	2.199
Imme at Hammar DCD	Measurement range pk [N]	±2224
Model 086C03	Hammer mass [kg]	0.16-0.235
	Resonant frequency [Hz]	>22000
	Tips number [-]	1

Table 4.1.2.2 – Accelerometer technical characteristics.

Component	Channel ID	Accelerometer ID	Node ID	Weight [kg]	Sensitivity [mV/g]	Acc. Direction [-]	
LW139287	1	Tri_1_x	1	$14.2 \cdot 10^{-3}$	96.1	-X	

							_
(Peroni)	2	Tri_1_y			97.3	-Y	l
	3	Tri_1_z			99.3	+Z	
	4	Tri_3_x			102	-X	
LW226918	5	Tri_3_y	11	$14.2 \cdot 10^{-3}$	102.1	+Z	
-	6	Tri_3_z			102.3	-Y	
	8	Tri_2_x			102.5	+X	
LW226919	9	Tri_2_y	8	$14.2 \cdot 10^{-3}$	97.5	+Y	
	11	Tri 2 z			98.6	+Z	



Figure 4.1.2.7 – Steering wheel weight.

#### 4.1.3 Geometrical aspects and acquisition parameters

To run the experimental modal analysis on the component it is necessary to create a model by identifying the most relevant nodes that allow to build a reference geometry as much close as the real one and at the same time could allow to give important information with regards to the response of the system.

The definition of the nodes map is carried out by obtaining the CAD geometry information of the chassis model developed in Solidworks then imported in Hypermesh and Lupos, where the coordinates of each node are measured according to the reference system set on the component, as shown in Figure 4.1.3.1.

As an overall, 19 nodes are identified and used to develop the model geometry in Siemens LMS Test.Lab, as shown in Figure 4.1.3.3





Figure 4.1.3.2 – Wheel CAD geometry.

According to reference system and to numerical model, definition of points and corresponding model nodes are listed in Table 1.3.1. 19 nodes are identified for roving hammer EMA. 3 of them are used for accelerometer positioning and hammered in the neighbourhood for auto-inertance. Node nomenclature is based on the following considerations:

- Wheel external frame starting from node 1 (12 o'clock; +Y) in clockwise direction until node 14.
- Wheel upper central part from left to right (15 to 16).
- Wheel lower central part from left to right (17 to 19).
- 20<sup>th</sup> node is the point of clamping.



Figure 4.1.3.3 – Experimental geometry: excited (left) and sensor nodes (right).



Figure 4.1.3.4 – Node nomenclature.

Details of the experimental geometry are supplied in Table 1.3.1 in which directions of actuation and reference system for hammer direction are listed. As a reference approach, the hammered directions are always orthogonal to nearest surface to defined nodes, some nodes however had a defined orientation with respect to the global reference frame hence Euler angles were also imposed precisely in order to be available to hammer the Steering Wheel in sufficient directions that can help in characterizing well the modes.

Identifier	Exp. point	Model node	<i>x</i> [m]	<i>y</i> [m]	<i>z</i> [m]	Notes
Acc. 1, <i>x</i> , <i>y</i> , <i>z</i>	1	28181	0	0.224	0.144	LW139287
-	2	28112	0.096	0.2	0144	-22° XY
-	3	27995	0.16	0.152	0.144	-33° XY
-	4	27945	0.184	0.12	0.144	-45° XY
-	5	27782	0.216	0.016	0.144	-85° XY
-	6	27613	0.208	-0.088	0.144	-117° XY
-	7	27432	0.104	-0.192	0.144	-160° XY
Acc. 2, <i>x</i> , <i>y</i> , <i>z</i>	8	27383	0	-0.2	0.144	LW226918; -180° XY
-	9	27443	-0.12	-0.184	0.144	-220° XY
-	10	27541	-0.176	-0.136	0.144	-240° XY
Acc. 3, <i>x</i> , <i>y</i> , <i>z</i>	11	27691	0.415	0.075	0.393	LW226919; -265° XY
-	12	27839	-0.216	0.064	0.144	-290° XY
-	13	27986	-0.176	0.152	0.144	-310° XY
-	14	28066	-0.128	0.192	0.144	-330° XY
-	15	18037	-0.072	0.04	0.04	-
-	16	18050	0.072	0.04	0.04	-
-	17	17667	-0.096	-0.08	0.04	-
	18	17679	0	-0.08	0.04	-
	19	17691	0.096	-0.08	0.04	
-	20	705	0	0	0	_

Table 4.1.3.1 – Geometry of I/O points.

During impact testing measurements, the parameters in Table 4.1.3.2 are selected:

Table 4.1.3.2 – Independent and dependent parameters of acquisition.

Quantity	Symbol	Relationship	Value	Units
Total acquisition time	Т	-	4	S
Sample frequency	$f_s$	-	8192	Hz
Total samples	N	$N = f_s T$	32768	-
Bandwidth (max frequency)	$f_b$	$f_b = \frac{f_s}{2}$	4096	Hz
FRF frequency resolution	Δf	$\Delta f = \frac{f_s}{N} = \frac{1}{T}$	0.25	Hz
Number of spectral lines	$N_{f}$	$N_f = \frac{N}{2}$	16384	-

The natural frequencies and damping ratios are extracted in the frequency range  $0\div600$  Hz and reported in the Table 4.1.3.3.

#### 4.1.4 Experimental data analysis



Figure 4.1.4.1 – Experimental model mesh in Testlab and Lupos respectively.

It can be noticed that the mesh is approximately identical which implies a good reliability. However, the experimental frequencies and the numerical frequencies are not exactly similar. The data was reorganized in order to get a coherent visualization that permits a smooth comparison and interpretation of results.

The natural frequencies and damping ratios are extracted in the frequency range  $0\div600$  Hz from Test.Lab v.17 Impact Testing environment where the values were extracted as shown in Figure 2.1 and values were reported relatively in Table 4.1.4.1. selecting the poles was limited on 520 Hz because after that frequency there were no identified poles.



Figure 4.1.4.2 – Stabilization FRF Sum diagram.



Mode	Freq. [Hz]	ζ, damping ratio [%]	Description
1	43.44	0.39	local bending YZ
2	45.28	0.69	1 <sup>st</sup> global bending
3	68.73	2.19	bending XZ
4	104.9	0.85	2 <sup>nd</sup> global bending
5	110.6	0.61	2 <sup>nd</sup> global bending
6	157.5	0.65	local bending XZ
7	291.7	1.11	local bending XZ
8	410	2.93	local bending XZ-YZ
9	425.1	3.97	bending YZ
10	434.4	2.55	3 <sup>rd</sup> global bending YZ-XZ
11	484.8	5.29	bending YZ
12	506.8	4.35	4 <sup>th</sup> global bending YZ-XZ







Mode 4 - 104.9 Hz, 0.84682 %





















Figure 4.1.4.3 – Steering wheel first 12 experimental mode shapes.

In order to carry out a comparison of the independency of the mode shapes. the MACX (compleX Modal Assurance Criterion) is adopted allowing to measure the correlation level between two complex mode shapes  $\Psi_j$  and  $\Psi_k$  respectively.

The MACX is defined as follow:

$$MACX_{j,k} = \frac{\left[\left|\boldsymbol{\Psi}_{j}^{H}\boldsymbol{\Psi}_{k}\right| + \left|\boldsymbol{\Psi}_{j}^{T}\boldsymbol{\Psi}_{k}\right|\right]^{2}}{\left[\boldsymbol{\Psi}_{j}^{H}\boldsymbol{\Psi}_{j} + \left|\boldsymbol{\Psi}_{j}^{T}\boldsymbol{\Psi}_{j}\right|\right]\left[\boldsymbol{\Psi}_{k}^{H}\boldsymbol{\Psi}_{k} + \left|\boldsymbol{\Psi}_{k}^{T}\boldsymbol{\Psi}_{k}\right|\right]}$$
(4.1.4.1)

where:

- the superscript *H* is the Hermitian, complex transposed;
- its value ranges from 0 to 1, always real, independently if the eigenvectors are real or complex;
- it is independent on the norm and the phase of vectors Ψ<sub>j</sub> and Ψ<sub>k</sub>, so the normalisation of eigenvectors is neglected;
- it is insensitive to conjugate operations on its arguments;
- $MACX(\Psi_j, \Psi_k) = 0$  is equivalent to  $\Psi_j \ \Psi_k = 0$  and  $\mu > 1 \ \Psi_k = 0$ ;
- if  $\Psi_k = z \Psi_j$  or  $\Psi_j = z \Psi_j$  for some complex number *z* then  $MACX(\Psi_j, \Psi_k) = 1$ . But it is worth noting that this is only a sufficient condition;

- if one of the vectors  $\Psi_j$  and  $\Psi_k$  is monophase, then the *MACX* and *MAC* criterions are identical;
- conversely, vectors  $\Psi_j$  and  $\Psi_k$  can be found such that  $MAC(\Psi_j, \Psi_k) = 0$  and  $MACX(\Psi_j, \Psi_k) = 1$ . But, in this case, both vectors are "full" complex since  $MPC(\Psi_j) = 0$  and  $MPC(\Psi_k) = 0$ .



Figure 4.1.4.4 – Experimental mode shapes Auto-MACX.

The definition of the Modal Phase Collinearity (MPC) index is defined as follows

$$MPC = MAC_{j,j^*} = \frac{\left|\boldsymbol{\Psi}_j^H \boldsymbol{\Psi}_j^*\right|^2}{\left[\boldsymbol{\Psi}_j^H \boldsymbol{\Psi}_j\right] \left[\boldsymbol{\Psi}_{j^*}^H \boldsymbol{\Psi}_{j^*}\right]} = \frac{\left|\boldsymbol{\Psi}_j^T \boldsymbol{\Psi}_j^*\right|^2}{\left[\boldsymbol{\Psi}_j^H \boldsymbol{\Psi}_j\right]^2} = \cos^2 \alpha_{j,j^*}$$
(4.1.4.2)

Which is a modal indicator that gives the idea whether a mode shape used in its computation is a real or complex-valued vector. Its estimate inherits the statistical properties of the corresponding mode shape estimate giving as a result an estimation of the phase of the dofs.

Table 4.1.4.2 – Modal Phase Collinearity values.

Mode	MPC
1	0,8712
2	0,956
3	0,8929
4	0,9393
5	0,8907
6	0,8971
7	0,7988
8	0,619
9	0,1666
10	0,2384
11	0,6128
12	0,2179



Figure 4.1.4.5 – MCP values for each mode.

Based on the obtained results, a real re-identification was performed for modes having relatively high MPC, in particular, it was performed for the first 7 modes.





Figure 4.1.4.6 – Steering Wheel first 7 re-identified real mode shapes.



Figure 4.1.4.7 – Experimental mode shapes Auto-MAC between real and complex modes.

## 4.1.5 Post-processing

The inertance is a frequency response function (FRF) that is defined in the frequency domain as the ratio between the acceleration and the external forces. To check the quality of the results and the linearity of the component several ways can be performed to do so as auto-inertance

Equation (4.1.5.1) or (4.1.5.2) and cross receptance following Maxwell approach Equation (4.1.5.3).

In the complex case of a *m*-dofs system the inertance formula can be defined as:

$$A_{j,k}(\omega) = \frac{\ddot{x}_{j}(\omega)}{F_{k}(\omega)} = -\omega^{2} \frac{x_{j}(\omega)}{F_{k}(\omega)} = \sum_{r=1}^{2m} \frac{-\omega^{2} \psi_{j,r} \psi_{k,r}}{a_{r}(i\omega + S_{r})}$$
(4.1.5.1)

where:

- $S_r$  is the complex pole of *r*-mode shape;
- $\omega$  is the excitation frequency;
- $\psi_{j,r}$  and  $\psi_{k,r}$  are the mode shapes related to the *j* and *k* nodes respectively.

However, in the real case of a *m*-dofs system the inertance formula can be defined as:

$$A_{j,k}(\omega) = \frac{\ddot{x}_{j}(\omega)}{F_{k}(\omega)} = -\omega^{2} \frac{x_{j}(\omega)}{F_{k}(\omega)} = \sum_{r=1}^{m} \frac{-\omega^{2} \Phi_{j,r} \Phi_{k,r}}{\omega_{r}^{2} + 2i\zeta_{r} \omega_{r} \omega - \omega^{2}}$$
(4.1.5.2)

where:

- $\omega_r$  is the real pole of *r*-mode shape;
- $\omega$  is the excitation frequency;
- $\Phi_{j,r}$  and  $\Phi_{k,r}$  are the mode shapes related to the *j* and *k* nodes respectively.

After performing the roving hammer test on the clamped wheel structure, the obtained modes and shapes with their relative vectors and frequencies were analysed by organizing them where each numerical node data is assigned to the relative experimental node by exploiting Matlab codes and the powerful functions that exist in Lupos.

The auto-inertance can be defined as the FRF evaluated in the node k due to the excitation on the node j. where j = k and along the same direction. Therefore, the node of excitation is the same as the node were the FRF is evaluated.

In the following figures the auto-inertances experimental (solid blue line), synthetised with lower and upper residuals (LR and UR) (dash-dot red line) and synthetised that is represented as well.



Figure 4.1.5.1 – Auto-inertance: E1y-R1y.



Figure 4.1.5.3 – Auto-inertance: E8y-R8y.



Figure 4.1.5.5 – Auto-inertance: E11y-R11y.



Figure 4.1.5.4 – Auto-inertance: E8z-R8z.



Figure 4.1.5.6 – Auto-inertance: E11z-R11z.

With real modal superposition of *m* modes or with complex modal superposition of 2m modes, a generic cross-receptance  $\alpha_{j,k}$  (similarly to mobility  $Y_{j,k}$  and inertance  $A_{j,k}$ ), the Maxwell reciprocity imposes that input and output can be switched:

$$\alpha_{j,k}(\omega) = \frac{x_j(\omega)}{F_k(\omega)} = \alpha_{k,j}(\omega) = \frac{x_j(\omega)}{F_j(\omega)} = \sum_{r=1}^m \frac{\Phi_{j,r} \Phi_{k,r}}{\omega_r^2 + 2i\zeta_r \omega_r \omega - \omega^2}$$
(4.1.5.3)

$$\alpha_{j,k}(\omega) = \frac{x_j(\omega)}{F_k(\omega)} = \sum_{r=1}^{2m} \frac{\Psi_{j,r} \Psi_{k,r}}{a_r(i\omega - S_r)}$$
(4.1.5.4)

An index of non-linearity regarding different dofs of the system is assumed as a consequence of the complex MAC index (scalar values between 0 and 1) applied to the modulus and phase of Equations. (4.1.5.5) or (4.1.5.6):

#### Figure 4.1.5.2 – Auto-inertance: E1z-R1z.

$$MAX_{j,k} = \frac{\left|\boldsymbol{\alpha}_{j,k}^{H}\boldsymbol{\alpha}_{k,j}\right|^{2}}{\left[\boldsymbol{\alpha}_{j,k}^{H}\boldsymbol{\alpha}_{j,k}\right]\left[\boldsymbol{\alpha}_{k,j}^{H}\boldsymbol{\alpha}_{k,j}\right]}$$
(4.1.5.5)

as a consequence, values far from 1 detect nonlinear behaviour, due to incoherence of similar receptances, according to Maxwell reciprocity. This index can be used for both numerical and experimental receptance transfer functions. Since it is not guaranteed the  $0\div1$  range, also a variation of MAX is defined similar to MACX definition.

$$MAXX_{j,k} = \frac{\left[ \left| \boldsymbol{\alpha}_{j,k}^{H} \boldsymbol{\alpha}_{k,j} \right| + \left| \boldsymbol{\alpha}_{j,k}^{T} \boldsymbol{\alpha}_{k,j} \right| \right]^{2}}{\left[ \left| \boldsymbol{\alpha}_{j,k}^{H} \boldsymbol{\alpha}_{j,k} \right| + \left| \boldsymbol{\alpha}_{j,k}^{T} \boldsymbol{\alpha}_{j,k} \right| \right] \left[ \left| \boldsymbol{\alpha}_{k,j}^{H} \boldsymbol{\alpha}_{k,j} \right| + \left| \boldsymbol{\alpha}_{k,j}^{T} \boldsymbol{\alpha}_{k,j} \right| \right]}$$
(4.1.5.6)



Figure 4.1.5.7 - MAX index in  $30 \div 520$  Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.1.5.8 - MAX index in  $30 \div 320$  Hz: on experimental receptances (left) and on synthetised receptances (right).



 $Figure \ 4.1.5.9 - MAX \ index \ in \ 30 \div 120 \ Hz:$  on experimental receptances (left) and on synthetised receptances (right).



Figure 4.1.5.10 – Reciprocity test: E8z-R8y: experimental inertance.



Figure 4.1.5.11 – Reciprocity test: E11y-R1y: experimental inertance.



Figure 4.1.5.12 – Reciprocity test: E11y-R8z: experimental inertance.



Figure 4.1.5.13 – Reciprocity test: E11y-R11z: experimental inertance.

The experimental modal analysis carried out on the chassis gives as a final result the identification of 12 mode shapes in the frequency range 0.600 Hz, where the global mode shapes identified are reported.

The identification of the mode shapes can be considered complete and satisfactory, as it can be noted in the previous figures looking at the good correspondence among the auto-inertances and the Maxwell reciprocity comparing the respective experimental and synthetized FRFs.

#### 4.2 Free-free steering wheel

The Free-free EMA has a similar scenario as indication Figure 4.2.2.2 where screw were mounted on the wheel in order to detect the torsional behaviour of the wheel and to be able to have more excited direction due to its particular geometry.



Figure 4.2.1 – Steering Wheel with added screws and accelerometers.

Since there are no ideal conditions for performing an ideal Free-Free experiment elastic bands were used to guarantee a fairly low natural frequency of the supposed rigid body motions, since theoretically they must be equal to 0 Hz. Also, in order to simulate the effect of these elastic bands certain type of elements were exploited in Lupos to do so. In particular the named elastic elements (els) were put in simulation with a stiffness value approximately equal to the real stiffness of the elastic bands. The procedure followed in order to calculate the required stiffness was as will be explained.

Firstly, the initial length of the elastic band was measured, then by trying different 3 masses the final different lengths were registered.



Figure 4.2.2 – Elastic band initial length measurement.



Figure 4.2.3 – Elastic band fixed end.




Figure 4.2.4 – Three different used components.

Table 4.2.1 – Used components mass properties
---

Component	Weight [kg]
Mass 1	1.068
Mass 2	1.8999
Mass 3	3.0259





Figure 4.2.5 – Elastic band initial length measurement.

Component	Elastic band length [m]
Mass 1	0.36
Mass 2	0.385
Mass 3	0.45

$$F_{r-1} = F(x - \Delta x), F_r = F(x), F_{r+1} = F(x + \Delta x)$$
(4.2.1)

$$K_{r}(x) \cong \frac{(F_{r} - F_{r-1}) + (F_{r+1} - F_{r})}{2\Delta x} = \frac{F_{r+1} - F_{r-1}}{2\Delta x}$$
(4.2.2)

clearvars clear all clc

```
m = [1.068 1.8899 3.0259]; %[kg]
x_i = [0.36 0.385 0.45]; %[m]
x_0 = 0.335; %[m]
F = 9.81.*m; %[N]
delta_F_3_1 = (F(1,3)-F(1,1));
delta_x = [x_i(2)-x_i(1) x_i(3)-x_i(2)];
K = 0.5*delta F 3 1/max(delta x); %[N/m]
```

So the final calculated values based on the adopted measurements were obtained but the minimum stiffness was considered in order to have the minimum effect of the bands and to preserve the physical properties of the model.

Table 4.2.3 – Elastic bands stiffness.





Figure 4.2.6 – Added elastic bands.

The goal of adding the effect of the bands is to improve the numerical model and to have more consistency with the experimental mode.



Figure 4.2.7 – First experimental global bending mode in extreme deformed frames.



Figure 4.2.8 – First numerical global bending mode.

By noticing the mode shape of this mode, it can be noticed that the numerical mode has a structural node at coordinate (0,0.13) where in the experimental mode it is not. Despite of adding these bands numerically the behaviour of the numerical model does not change importantly. Hence, the Optistruct model was considered mainly for this analysis.

The elastic bands (light green) are added in such a way that the symmetry of the effect on the Steering Wheel is guaranteed where they were added into two central symmetrical nodes as well. Moreover, the added masses can be simply neglected because they have a slight inertial affect, so the natural frequencies are approximately equal since the difference is between 0.5 and 0.7 Hz, thus they were not heavily considered.

Table 4.2.4 – Free-free model with rubber, Optistruct natural frequencies.

Mada #	Frequency [Hz]
Mode #	CHEXA8
1	8.55x10 <sup>-4</sup>
2	8.89x10 <sup>-4</sup>
3	9.96x10 <sup>-4</sup>
4	$1.57 \times 10^{-3}$

5	1.60x10 <sup>-3</sup>
6	1.65x10 <sup>-3</sup>
7	86.70
8	107.22
9	107.48
10	148.97
11	201.66
12	250.7





Figure 4.2.9 – First 3 rotational rigid modes, tetra and hexa models respectively.



Figure 4.2.10 – First 3 translational rigid modes, tetra and hexa models respectively.



Figure  $4.2.11 - 7^{\text{th}}$  mode (86.70 Hz) and  $8^{\text{th}}$  mode (107.22 Hz) respectively.



Figure  $4.2.12 - 9^{\text{th}}$  mode (107.48 Hz) and  $10^{\text{th}}$  mode (148.9 Hz) respectively.



Figure 4.2.13 – 11<sup>th</sup> mode (201.66 Hz) and 12<sup>th</sup> mode (250.7 Hz) respectively.

# 4.2.1 Experimental – numerical post processing

Testlab		Optistruct	
Mode #	Experimental frequency [Hz]	Mode #	Numerical frequency [Hz]
1	87.77	1	86.71
2	104.9	2	107.2
3	105.9	3	107.5
-	-	-	143.2
-	134.5	4	149
4	153.6	-	200.3
5	215	5	201.7
6	227.1	6	220.8
-	-	-	234.1
-	243.1	-	250.8
-	259.6	-	253.9
-	273.3	-	267.7
-	275.5	-	275.3
-	290.4	-	280.2

Table 4.2.1 – Experimental and numerical natural frequencies.



Figure 4.2.1.1 – Free-free Steering Wheel AutoMAC.



Figure 4.2.1.2 – Free-free Steering Wheel MAC and MACW2 respectively.



Figure 4.2.1.3 – Free-free Steering Wheel MAC\_distW.

## 4.2.2 Setup and sensor positioning

The free-free experiment was performed by adding the necessary inertial contribution of some additional components that were inserted in order to meet the real case condition.



Figure 4.2.2.1 – Additional screws mass.

Hence, the additional inertial contribution of the screws and masses was as follows:



### Figure 4.2.2.2 – A followed similar scenario of experiment: Screws (red) excitation direction (green)

The screws are going to be inserted in all the nodes of the handlebar, as example in such an order:

- Clockwise direction in particular 3, 5, 6, 7, 9 and 12 o'clock as shown in Figure 4.2.2.2

It was noticed that the screw on the upper position (12 o'clock) is practically a node in the 4<sup>th</sup> mode, however it is possible to clamp the screw there and excite it, since the energy will be transmitted consequently from the concerned node to the whole system. Moreover, this point is a node in mode 4 but it is not in other modes.

An important aspect that must be taken into consideration is the inertial contribution of these additional screws. Their inertia may decrease the natural frequency.

Additional inertial contribution	Exp. point	Model node
Acc. 1, $x$ , $y$ , $z$ , & screw masses	1	28181
Screw mass	2	28112
Screw mass	3	27995
Screw mass	4	27945
Screw mass	5	27782
Screw mass	6	27613
Screw mass	7	27432
Acc. 2, $x$ , $y$ , $z$ , & screw masses	8	27383
Screw mass	9	27443
Screw mass	10	27541
Acc. 3, $x$ , $y$ , $z$ , & screw masses	11	27691
Screw mass	12	27839
Screw mass	13	27986
Screw mass	14	28066
-	15	18037
-	16	18050
-	17	17667
-	18	17679
-	19	17691

Table 4.2.2.2 – Added inertial contribution of nodes.



### Figure 4.2.2.3 – Additional screws and sensors position.

The same data sheets of sensors and hammer in chapter 4.1 were exploited, same setup with same geometry same positions of sensors and same acquisition data. However, in this experiment since the screws were inserted in the first 14 nodes the Euler angles were not assumed, or they were considered null.

The experimental modal analysis is performed adopting free-free conditions, exciting the structure with roving hammer and evaluating the response through accelerometers, either mono-axial or triaxial. The steering wheel is fixed on a huge test rig machine supplied by soft springs that do not affect the component natural frequencies the excitations were done on 19 nodes along all the possible directions allowed by its geometry, obtaining as an overall 47 excitations to the system. The responses of 3 nodes are measured: 1, 8, and 11. The response for each node is averaged on a set of 5 responses acquired to give more repeatability and stability to each measurement. For each response:

- the spatial coordinates are continuous for each instant of time.
- the time/frequency data and the modal content are acquired and identified with Siemens Test.Lab v.17.
- it evinces those modal coordinates enable approximate solutions, according to the number of modal coordinates used, thus in nonlinear problems this property can improve computational efforts in the best way.





Figure 4.2.2.4 – Experimental setup scenario.

Both pictures of the entire system, with different points of view and details of the hardware implemented and of their location are requested information.

Importance has the reference system (x, y, z) used, according also to suitable models. In case of already developed models, consider the same reference system.





Figure 4.2.2.5 – Overview of the experimental setup.

Figure 4.2.2.6 – Details of the implemented hardware, from left to right in reading order: Tri-axial-node 1, Tri-axial-node 8, Tri-axial-node 11.

The acquisition is performed with a LMS SCADAS Mobile system. A roving hammer EMA is performed using an impact hammer PCB 086C03 using hard steel tip (Figure 4.1.2.4 and Figure 4.1.2.5) to reach a good frequency identification range of the structure. A series of 3 hammer repetitions is linearly averaged. The characteristics are listed in Table 4.1.2.1.

The definition of the nodes map is carried out by obtaining the CAD geometry information of the chassis model developed in Solidworks then imported in Hypermesh and Lupos, where the coordinates of each node are measured according to the reference system set on the component, as shown in Figure 4.2.2.9.

As an overall, 19 nodes are identified and used to develop the model geometry in Siemens LMS Test.Lab, as shown in Figure 4.2.2.5





Figure 4.2.2.8 – Wheel CAD geometry.

According to reference system and to numerical model, definition of points and corresponding model nodes are listed in Table 1.3.1. 19 nodes are identified for roving hammer EMA. 3 of them are used for accelerometer positioning and hammered in the neighbourhood for auto-inertance. Node nomenclature is based on the following considerations:

- Wheel external frame starting from node 1 (12 o'clock; +Y) in clockwise direction until node 14.
- Wheel upper central part from left to right (15 to 16).
- Wheel lower central part from left to right (17 to 19).



• Figure 4.2.2.9 – Experimental geometry: excited (left) and sensor nodes (right).

Details of the experimental geometry are supplied in Table 1.3.1 in which directions of actuation and reference system for hammer direction are listed. As a reference approach, the hammered directions are always orthogonal to nearest surface to defined nodes, some nodes however had a defined orientation with respect to the global reference frame hence Euler angles were also imposed precisely.

Identifier	Exp. point	Model node	<i>x</i> [m]	<i>y</i> [m]	<i>z</i> [m]	Notes
Acc. 1, <i>x</i> , <i>y</i> , <i>z</i>	1	28181	0	0.224	0.144	LW139287
-	2	28112	0.096	0.2	0144	-
-	3	27995	0.16	0.152	0.144	-
-	4	27945	0.184	0.12	0.144	-
-	5	27782	0.216	0.016	0.144	-
-	6	27613	0.208	-0.088	0.144	-
-	7	27432	0.104	-0.192	0.144	-
	8	27282	0	-0.2	0.144	LW226918, screw fixed
Acc. $2, x, y, z$		27383				radially
-	9	27443	-0.12	-0.184	0.144	-
-	10	27541	-0.176	-0.136	0.144	-
Acc. 3, <i>x</i> , <i>y</i> , <i>z</i>	11	27691	0.415	0.075	0.393	LW226919
-	12	27839	-0.216	0.064	0.144	-
-	13	27986	-0.176	0.152	0.144	-
-	14	28066	-0.128	0.192	0.144	-
-	15	18037	-0.072	0.04	0.04	-
-	16	18050	0.072	0.04	0.04	-
-	17	17667	-0.096	-0.08	0.04	-
-	18	17679	0	-0.08	0.04	-
-	19	17691	0.096	-0.08	0.04	-
-	20	705	0	0	0	

Table 4.2.2.3 – Geometry of I/O points.

# 4.2.3 Experimental data analysis

During impact testing measurements, the parameters in Table 4.2.3.1 are selected:

Table 4.2.3.1 – Independent and dependent parameters of acquisition.

Quantity	Symbol	Relationship	Value	Units
Total acquisition time	Т	-	4	S
Sample frequency	$f_s$	-	8192	Hz
Total samples	N	$N = f_s T$	32768	-
Bandwidth (max frequency)	$f_b$	$f_b = \frac{f_s}{2}$	4096	Hz
FRF frequency resolution	Δf	$\Delta f = \frac{f_s}{N} = \frac{1}{T}$	0.25	Hz
Number of spectral lines	$N_{f}$	$N_f = \frac{N}{2}$	16384	-



Figure 4.2.3.1 – Experimental model mesh in Testlab and Lupos respectively.

The natural frequencies and damping ratios are extracted in the frequency range  $0\div300$  Hz from Test.Lab v.17 Impact Testing environment where the values were extracted as shown in Figure 4.2.3.2 the model size was 350 and values were reported relatively in Table 4.2.3.2.



Figure 4.2.3.2 – Stabilization FRF diagram.

Table 4.2.3.2 – Ex	perimental natura	l frequencies a	nd damping rati	os of the Steering Wheel
I were manena and				

Mode	Freq. [Hz]	ζ, damping ratio [%]	Description
1	43.94	1.18	Bending XZ
2	87.77	0.35	1 <sup>st</sup> local bending YZ
3	104.9	0.458	2 <sup>nd</sup> global bending YZ
4	105.9	0.392	3 <sup>rd</sup> bending XZ-YZ
5	134.5	0.951	local bending XZ
6	153.6	0.585	4 <sup>th</sup> global bending XZ-YZ
7	215.1	2.25	5 <sup>th</sup> global bending XZ-YZ
8	227.1	2.97	6 <sup>th</sup> global bending XZ-YZ
9	243.1	5.01	local bending XZ
10	259.6	0.0044	-

11	273.3	0.029	-
12	275.5	1.042	7 <sup>th</sup> global bending XZ-YZ
13	290.4	0.216	-

































Mode 10 - 259.6 Hz, 0.0044744 %

-0.2

-0.1

0

Axis x [m]

0.1

Mode 12 - 275.5 Hz, 1.0423 %





Figure 4.2.3.3 – Steering wheel first 13 experimental mode shapes.

In order to carry out a comparison of the independency of the mode shapes. the MACX (compleX Modal Assurance Criterion) is adopted allowing to measure the correlation level between two complex mode shapes  $\Psi_i$  and  $\Psi_k$  respectively.

The MACX defined in Equation (4.1.4.1) was considered.



Figure 4.2.3.4 – Experimental mode shapes Auto-MACX.

The definition of the Modal Phase Collinearity (MPC) index is defined as in Equation (4.1.4.2)

Mode	MPC
1	0,9656
2	0,8747
3	0,9296
4	0,5945
5	0,5392
6	0,4647
7	0,4382
8	0,0745
9	0,6311
10	0,9918



Figure 4.2.3.5 – MCP values for each mode.

Based on the obtained results, a real re-identification was performed for the modes having relatively high MPC, in particular, it was performed specially for modes 1, 2, 3, 10, 11, 13.















Mode 8 - 227.1 Hz



Mode 10 - 259.6 Hz





Figure 4.2.3.6 – Steering Wheel re-identified real mode shapes.



Figure 4.2.3.7 – Experimental mode shapes Auto-MAC between complex and real modes.

### 4.2.4 Post-processing

The inertance is a frequency response function (FRF) that is defined in the frequency domain as the ratio between the acceleration and the external forces. To check the quality of the results and the linearity of the component several ways can be performed to do so as auto-inertance

Equation (4.1.5.1) or (4.1.5.2) and cross receptance following Maxwell approach Equation (4.1.5.3).

In the complex case of a m-dofs system the inertance formula can be defined as in Equation (4.1.5.1)

After performing the roving hammer test on the clamped wheel structure, the obtained modes and shapes with their relative vectors and frequencies were analysed by organizing them where each numerical node data is assigned to the relative experimental node by exploiting matlab codes and the powerful functions that exist in Lupos.

The auto-inertance can be defined as the FRF evaluated in the node k due to the excitation on the node j. where j = k and along the same direction. Therefore, the node of excitation is the same as the node were the FRF is evaluated.

In the following figures the auto-inertances experimental (solid blue line), synthetised with lower and upper residuals (LR and UR) (dash-dot red line) and synthetised with modal superposition that are presented as well.



Figure 4.2.4.1 – Auto-inertance: E1x-R1x.

Figure 4.2.4.2 – Auto-inertance: E1y-R1y.



Figure 4.2.4.3 – Auto-inertance: E1z-R1z.







Figure 4.2.4.4 – Auto-inertance: E8x-R8x.

Figure 4.3.4.5 – Auto-inertance: E8y-R8y.



Figure 4.2.4.6 – Auto-inertance: E8z-R8z.



Figure 4.2.4.7 – Auto-inertance: E11x-R11x.



Figure 4.2.4.8 – Auto-inertance: E11y-R11y.



Figure 4.2.4.8 – Auto-inertance: E11z-R11z.

However, in the real case of a m-dofs system the inertance formula can be defined as in Equation (4.1.5.2):



Figure 4.2.4.9 – Auto-inertance: E1x-R1x.



Figure 4.2.4.10 – Auto-inertance: E1y-R1y.



Figure 4.2.4.11 – Auto-inertance: E1z-R1z.



Figure 4.2.4.12 – Auto-inertance: E8y-R8y.



150

Exp E8y R8y Syn E8y R8y MS

300





0

Figure 4.2.4.14 – Auto-inertance: E8z-R8z.



Figure 4.2.4.15 – Auto-inertance: E11x-R11x.



Figure 4.2.4.16 – Auto-inertance: E11y-R11y.



Figure 4.2.4.17 – Auto-inertance: E11z-R11z.

With real modal superposition of *m* modes or with complex modal superposition of 2m modes, a generic cross-receptance  $\alpha_{j,k}$  (similarly to mobility  $Y_{j,k}$  and inertance  $A_{j,k}$ ), the Maxwell reciprocity imposes that input and output can be switched as in Equation (4.1.5.3) or (4.1.5.4).

An index of non-linearity regarding different dofs of the system is assumed as a consequence of the complex MAC index (scalar values between 0 and 1) applied to the modulus and phase of Equations. (4.1.5.5) or (4.1.5.6).



Figure 4.2.4.18– MAX index in 30÷300 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.2.4.19 – MAX index in 30÷200 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.2.4.20 – MAX index in  $0\div120$  Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.2.4.21 – Reciprocity test experimental inertance: E1x-R1y (left) E1x-R11z (right).



Figure 4.2.4.22 – Reciprocity test experimental inertance: E8y-R1x (left) E8x-R11x (right).



Figure 4.2.4.23 – Reciprocity test experimental inertance: E11z-R8z (left) E11x-R11y (right).

The experimental modal analysis carried out on the chassis gives as a final result the identification of 13 mode shapes in the frequency range 0÷300 Hz, where the global mode shapes identified are reported.

The identification of the mode shapes can be considered complete and satisfactory, as it can be noted in the previous figures looking at the good correspondence among the auto-inertances and the Maxwell reciprocity comparing the respective experimental and synthetized FRFs.

### 4.3 Clamped Steering Wheel Second experiment

During the process of identification, it was noticed that the 4<sup>th</sup> numerical mode is not existing experimentally. After checking what was behind, a mismatch was discovered where the 4<sup>th</sup> numerical mode is a torsional mode while from the experiment no torsional mode was excited, hence was not even discovered. So, it was decided to repeat that experiment to identify the torsional mode or other torsional modes if they exist. Simultaneously, a numerical optimization procedure was performed in order to reach the most optimal numerical solution that maybe very suitable to the real exact solution. To bring it clear, the new experiment will be performed exciting the radial and

tangential directions as well so torsional behaviour can be detected the problem was that the structure has a circular geometry that makes difficult the excitation in the tangential direction. But there is a way to bypass the problem, to do so, 2 screws will be mounted to the structure in such a way that the tangential direction is excited through these screws, and this must provoke the torsional mode leading to their detection.

A new constraint was suggested to the Steering Wheel, the change is desired in order to get a more accurate modelling for the real object where the experimental test will be performed by mounting the wheel on the test rig machine by a screw. Hence an optimization process was needed.



Figure 4.3.1 – Front section and Top view of new constraint (Hypermesh).

However, to have a broad and better optimization affect, more parameters were iterated as shown in the tables below.



Figure 4.3.2 – 4<sup>th</sup> numerical, experimentally unidentified mode in Optistruct (73.3 Hz) and Lupos (81.01 Hz) respectively.

The objective function in the first optimization was to reach the maximum compatibility of both the MAC and MACW2 iterating on inertial parameters. The optimization attempt was performed iterating 3 main parameters that aimed to improve the numerical mode to approach the 1<sup>st</sup> experimental test, which will not be very different from the second experiment test results. The values indicated in Table 4.2.6.1 were obtained after 81 iterations.

Iterated parameters	<b>Optimization final results</b>
<i>t</i> [-], Clamping rod diameter multiplier	5.0016
<i>z</i> [m], Clamping rod height	-0.0099783
$E_{R}$ [Pa], Rubber young Modulus	5261354.099





Figure 4.3.3 – Iterated parameters values of 1<sup>st</sup> attempt, unidimensional and dimensional respectively.



Figure 4.3.4 – Objective functions of 1<sup>st</sup> attempt.

From the obtained results, it can be noticed that the model is improving slightly but insufficiently, this indicates that the selected parameters were not influencing importantly, hence a very low improvement would occur. This low margin of variability that was set to the algorithm did not allow for obtaining better results. For this reason, higher margin of variability was given in the second optimization, in particular the inertial and structural parameters are going to be iterated in 200 iterations.

Table 4.3.2 – Second evaluated optimization results.

Iterated parameters Optimization final results
--

<i>t</i> [-], Clamping rod diameter multiplier	5.5023
$E_{\rm R}$ [Pa], Rubber young Modulus	4235478.77
E steel [Pa], Steel young Modulus	168118215781.53
E AI [Pa], Aluminuium young Modulus	67000145634.87
z [m], Clamping rod height	-7.3509e-05
$\rho_{\rm R}$ , Rubber density [kg/m <sup>3</sup> ]	357.43
$\rho_{steel}$ , Steel density [kg/m <sup>3</sup> ]	7927.84
$\rho_{Al}$ , Aluminium density [kg/m <sup>3</sup> ]	2742.36
$v_R$ , Rubber poisson ratio [-]	0.43513
<i>vsteel</i> , Steel poisson ratio [-]	0.29514
$v_{Al}$ , Aluminium poisson ratio [-]	0.26725



Figure 4.3.5 – Iterated parameters values of 2<sup>nd</sup> attempt, unidimensional and dimensional respectively.



Figure 4.3.6 – Objective functions of 2<sup>nd</sup> attempt.

Similarly in the second attempt the objective function is not approaching an optimal value and it is noticed that the function remained constant even after a high number of iterations hence another approach must be adopted by varying the previously iterated parameters.

# 4.3.1 Experimental – numerical post processing

Testlab		Lupos	
Mode #	Experimental	Mode #	Numerical
	frequency [Hz]		frequency [Hz]
1	43.642	1	48.46
2	46.0557	2	55.72
3	69.556	3	71.45
4	70.234	4	80.001
5	105.46	5	114.55
6	111.54	6	120.43
-	-	-	142.3
7	158.05	7	150.17
-	-	-	219.6
-	-	-	229.3
-	-	-	233.0
-	-	-	256.5
-	-	-	259.7
-	-	-	277.0
-	-	-	277.5
-	-	-	285.3
-	-	-	290.3
8	292.2	8	297.25
-	-	-	303.3
-	-	-	322.3
-	-	-	323
-	-	-	354.5
-	-	-	357

Table 4.3.1 – Comparison between experimental and numerical Frequency results.

In order to confirm the compatibility between the numerical and the real models the MAC as indicated in Equation (4.1.1.1) was again performed.



Figure 4.3.1.1 – AutoMAC of experimental modes.

To assure that there are no double selected poles and that the spatial aliasing is negligible the AutoMAC results, despite of a low spatial aliasing signs, are sufficiently convenient.



Figure 4.3.1.2 – MAC of numerical and experimental modes.

The compatibility of the first 7 modes is pretty high, the 7<sup>th</sup> modes is around 160 Hz. This justifies as seen before in the 1<sup>st</sup> experiment that the Steering Wheel is linear until this frequency, where it tends to be strongly non-linear after. It is important to emphasize again the fact that the 7<sup>th</sup> numerical mode is local, and it is related exclusively to the rubber. The correlation that is noticed in between modes 11 and 12 with the 11<sup>th</sup> numerical mode, is probably due to spatial aliasing because the 11<sup>th</sup> numerical frequency is equal to 209.2 Hz while the 11<sup>th</sup> and 12<sup>th</sup> experimental modes are 425.8 and 507.8 Hz respectively.



Figure 4.3.1.3 – MAC\_distW and MACW2 of numerical and experimental modes.

To resolve all doubts the Figure 4.3.1.3 is assuring the hypothesis of spatial aliasing by demonstrating that there is no correlation between the concerned nodes.

### 4.3.2 Experiment Setup and sensor positioning

Following the same setup of the first experiment of Steering Wheel, the second experiment was performed with slight changes pf accelerometers orientation as will be shown in the following
chapters. The experimental test was performed in 2022-02-28 and postprocessed in 2022-03-01 on which the main activity was concluded.

The experimental modal analysis is performed adopting free-free conditions, exciting the structure with roving hammer and evaluating the response through accelerometers, either mono-axial or triaxial. The Steering Wheel is fixed on a huge test rig machine supplied by soft springs that do not affect the component natural frequencies the excitations were done on 19 nodes along all the possible directions allowed by its geometry, obtaining as an overall 47 excitations to the system. The responses of 3 nodes are measured: 1, 8, and 11. The response for each node is averaged on a set of 5 responses acquired to give more repeatability and stability to each measurement.

For each response:

- the spatial coordinates are continuous for each instant of time; •
- the time/frequency data and the modal content are acquired and identified with Siemens • Test.Lab v.17:
- it evinces those modal coordinates enable approximate solutions, according to the number of ٠ modal coordinates used, thus in nonlinear problems this property can improve computational efforts in the best way.



Figure 4.3.2.1 – Experimental setup scenario.

Both pictures of the entire system, with different points of view and details of the hardware implemented and of their location are requested information.

Importance has the reference system (x, y, z) used, according also to suitable models. In case of already developed models, consider the same reference system.



Figure 4.3.2.2 – Overview of the experimental setup.



Figure 4.3.2.3 – Details of the implemented hardware, from left to right in reading order: Tri-axial-node 1, Tri-axial-node 8, Tri-axial-node 11.

The acquisition is performed with a LMS SCADAS Mobile system. A roving hammer EMA is performed using an impact hammer PCB 086C03 using hard steel tip (Figure 4.1.2.4 and Figure 4.1.2.5) to reach a good frequency identification range of the structure. A series of 3 hammer repetitions is linearly averaged. The characteristics are listed in Table 4.1.2.1.

Component	Channel ID	Accelerometer ID	Node ID	Weight [kg]	Sensitivity [mV/g]	Acc. Direction [-]
	1	Tri_1_x		14.2.10-3	102	-X
LW226918	2	Tri_1_y	1		102.1	+Z
	3	Tri_1_z			102.3	+Y
1 W120297	4	Tri_3_x		14.2.10-3	96.1	+Y
(Peroni)	5	Tri_3_y	11		97.3	-Z
(Peroliii)	6	Tri_3_z			99.3	-X
LW226919	8	Tri_2_x			102.5	-X
	9	Tri_2_y	8	14.2.10-3	97.5	-Z
	11	Tri_2_z			98.6	-Y

Table 4.3.2.1 – Accelerometer technical characteristics.



 $Figure \ 4.3.2.4 - Steering \ Wheel \ weight \ with \ added \ screws \ and \ accelerometers.$ 



Figure 4.3.2.5 – Steering Wheel weight with added screws and accelerometers.

Component	Weight [kg]	
Steering Wheel	3.9811	
Screw	1.39 10 <sup>-3</sup>	

Table 4.3.2.2 – Steering	Wheel and scr	rews mass properties.
--------------------------	---------------	-----------------------

## 4.3.3 Geometrical aspects and acquisition parameters

To run the experimental modal analysis on the component it is necessary to create a model by identifying the most relevant nodes that allow to build a reference geometry as much close as the real one and at the same time could allow to give important information with regards to the response of the system.

The definition of the nodes map is carried out by obtaining the CAD geometry information of the chassis model developed in Solidworks then imported in Hypermesh and Lupos, where the coordinates of each node are measured according to the reference system set on the component, as shown in Figure 4.3.3.1.

As an overall, 19 nodes are identified and used to develop the model geometry in Siemens LMS Test.Lab, as shown in Figure 4.3.3.2.





Figure 4.3.3.2 – Wheel CAD geometry.

According to reference system and to numerical model, definition of points and corresponding model nodes are listed in Table 4.3.3.1. 19 nodes are identified for roving hammer EMA. 3 of them are used for accelerometer positioning and hammered in the neighbourhood for auto-inertance.

Details of the experimental geometry are supplied in Table 4.3.3.1 in which directions of actuation and reference system for hammer direction are listed. As a reference approach, the hammered directions are always orthogonal to nearest surface to defined nodes, some nodes differently from the 1<sup>st</sup> experiment had no defined orientation with respect to the global reference frame hence Euler angles were not imposed or considered.

Identifier	Exp. point	Model node	<i>x</i> [m]	<i>y</i> [m]	<i>z</i> [m]	Notes
Acc. 1, <i>x</i> , <i>y</i> , <i>z</i>	1	28181	0	0.224	0.144	LW139287
-	2	28112	0.096	0.2	0144	-
-	3	27995	0.16	0.152	0.144	-
-	4	27945	0.184	0.12	0.144	-
-	5	27782	0.216	0.016	0.144	-
-	6	27613	0.208	-0.088	0.144	-
-	7	27432	0.104	-0.192	0.144	-
	8	27282	0	-0.2	0.144	LW226918, screw fixed
Acc. $2, x, y, z$		27383				radially
-	9	27443	-0.12	-0.184	0.144	-
-	10	27541	-0.176	-0.136	0.144	-
Acc. 3, <i>x</i> , <i>y</i> , <i>z</i>	11	27691	0.415	0.075	0.393	LW226919
-	12	27839	-0.216	0.064	0.144	-
-	13	27986	-0.176	0.152	0.144	-
-	14	28066	-0.128	0.192	0.144	-
-	15	18037	-0.072	0.04	0.04	-
-	16	18050	0.072	0.04	0.04	-
_	17	17667	-0.096	-0.08	0.04	_

Table 4.3.3.1 – Ge	ometry of I/O	points.
--------------------	---------------	---------

-	18	17679	0	-0.08	0.04	-
-	19	17691	0.096	-0.08	0.04	-
-	20	705	0	0	0	-

During impact testing measurements, the parameters in Table 4.3.3.2 are selected:

Table 4.3.3.2 – Independent and dependent parameters of acquisition.

Quantity	Symbol	Relationship	Value	Units
Total acquisition time	Т	-	4	S
Sample frequency	$f_s$	-	8192	Hz
Total samples	N	$N = f_s T$	32768	-
Bandwidth (max frequency)	$f_{b}$	$f_b = \frac{f_s}{2}$	4096	Hz
FRF frequency resolution	$\Delta f$	$\Delta f = \frac{f_s}{N} = \frac{1}{T}$	0.25	Hz
Number of spectral lines	$N_{f}$	$N_f = \frac{N}{2}$	16384	-

## 4.3.4 Experimental data analysis

The natural frequencies and damping ratios are extracted in the frequency range 0.550 Hz from Test.Lab v.17 Impact Testing environment where the values were extracted as shown in Figure 4.3.4.1 for a model size of 500 and values were reported relatively in Table 4.3.4.1.



Figure 4.3.4.1 – Stabilization FRF-sum diagram.

Table 4.3.4.1 – Experimental natural frequencies and damping ratios of the Steering Wheel.

Mode Freq. [Hz]		ζ, damping ratio [%]	Description
1	43.64	0.674	local bending YZ
2	46.05	0.565	local bending XZ
3	69.55	0.799	_

4	70.23	1.259	1st torsional XY
5	105.46	0.329	1 <sup>st</sup> global bending XZ-YZ
6	111.55	0.326	2 <sup>nd</sup> global bending XZ-YZ
7	158.05	0.425	local bending XZ-YZ
8	274.89	0.727	3 <sup>rd</sup> global bending XZ-YZ
9	287.81	1.27	local bending YZ
10	292.2	0.923	local bending XZ-YZ
11	425.77	1.77	local bending XZ
12	507.83	2.025	4 <sup>th</sup> global bending XZ-YZ
13	521.8	3.694	5 <sup>th</sup> global bending XZ-YZ









Mode 6 - 111.5 Hz, 0.32645 %



Mode 1 - 43.64 Hz, 0.67427 %



Mode 3 - 69.56 Hz, 0.79903 %









Mode 11 - 425.9 Hz, 1.7705 %



Mode 12 - 507.9 Hz, 2.0257 %



116 / 168

-0.2

-0.3

-0.2



Figure 4.3.4.2 – Steering Wheel first 13 complex mode shapes.

According to equation (4.1.4.1) the following diagram is obtained for the second Steering Wheel experiment.



Figure 4.3.4.3 – Experimental mode shapes Auto-MACX.

The definition of the Modal Phase Collinearity (MPC) index as defined in equation (4.1.4.2)

Table 4.3.4.2 – Modal Phase Collinearity values.

Mode	MPC
1	0.0742
2	0.697
3	0.21
4	0.61
5	0.79
6	0.54
7	0.48
8	0.76
9	0.58
10	0.65
11	0.61



Figure 4.3.4.4 – MCP values for each mode.

Based on the obtained results, no real re-identification was performed since the modes are heavily complex.

# 4.3.5 Post-processing

The inertance is a frequency response function (FRF) that is defined in the frequency domain as the ratio between the acceleration and the external forces. To check the quality of the results and the linearity of the component several ways can be performed to do so as auto-inertance Equation (4.1.5.1) or (4.1.5.2) and cross receptance following Maxwell approach Equation (4.1.5.3).

In the complex case of a *m*-dofs system the inertance formula as defined in equation (4.1.5.1)

The auto-inertance can be defined as the FRF evaluated in the node k due to the excitation on the node j. where j = k and along the same direction. Therefore, the node of excitation is the same as the node were the FRF is evaluated.

In the following figures the auto-inertances experimental (solid blue line), synthetised with lower and upper residuals (LR and UR) (dash-dot red line) and synthetised that is represented as well. At low frequencies, the clamped structure is characterized by inertances that tends to 0 is noticed from the blue curve.



Figure 4.3.5.1 – Auto-inertance: E1x-R1x.



Frequency [Hz]

300

Exp E1y R1y Syn E1y R1y

450



10<sup>3</sup>

10<sup>2</sup>

10<sup>1</sup>

10<sup>0</sup>

10

10<sup>-2</sup>

10<sup>-3</sup>

0

150

Inertance [m/s<sup>2</sup> / N]

Figure 4.3.5.3 – Auto-inertance: E1z-R1z.



Figure 4.3.5.4 – Auto-inertance: E8x-R8x.



Figure 4.3.5.5 – Auto-inertance: E8y-R8y.



Figure 4.3.5.6 – Auto-inertance: E8z-R8z.



Figure 4.3.5.7 – Auto-inertance: E11x-R11x.

Figure 4.3.5.8 – Auto-inertance: E11y-R11y.

Frequency [Hz]

300

Exp E11y R11y Syn E11y R11y

450



Figure 4.3.5.9 – Auto-inertance: E11z-R11z.

With real modal superposition of *m* modes or with complex modal superposition of 2m modes, a generic cross-receptance  $\alpha_{j,k}$  (similarly to mobility  $Y_{j,k}$  and inertance  $A_{j,k}$ ), the Maxwell reciprocity imposes that input and output can be switched as in Equation (4.1.5.3) and (4.1.5.4).

Furthermore, an index of non-linearity regarding different dofs of the system is assumed as a consequence of the complex MAC index (scalar values between 0 and 1) applied to the modulus and phase of Equations. (4.1.5.5) or (4.1.5.6). Consequently, values far from 1 detect nonlinear behaviour, due to incoherence of similar receptances, according to Maxwell reciprocity. This index can be used for both numerical and experimental receptance transfer functions. Since it is not guaranteed the  $0\div1$  range, also a variation of MAX is defined similar to MACX definition.



Figure 4.3.5.10 – MAX index in 30÷550 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.3.5.11 - MAX index in  $30 \div 300$  Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.3.5.12 - MAX index in  $30 \div 150$  Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.3.5.13 – MAX index in 30÷120 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.3.5.14 – Reciprocity test experimental inertance: E1x-R1y (left) E1x-R11z (right).



Figure 4.3.5.15 – Reciprocity test experimental inertance: E8y-R1x (left) E8x-R11x (right).



Figure 4.3.5.16 – Reciprocity test experimental inertance: E11z-R8z (left) E11x-R11y (right).

The experimental modal analysis carried out on the Steering Wheel gives as a final result the identification of 13 mode shapes in the frequency range 0.550 Hz, where the global mode shapes identified are reported.

The identification of the mode shapes can be considered complete and satisfactory, as it can be noted in the previous figures looking at the good correspondence among the auto-inertances and the Maxwell reciprocity comparing the respective experimental and synthetized FRFs.

## 4.4 Steering Wheel models comparison

To check how would the behaviour of the wheel change in different boundary conditions. The freefree steering wheel has same similar structural mode shapes but at higher frequencies as can be noticed mainly from the first frequencies.



Figure 4.4.1 – Lupos clamped-free numerical models' comparison.

The difference between the two identification is that in the second EMA the torsional mode at around 70 Hz ( $4^{th}$  mode) that was not found in the  $1^{st}$  EMA.



Figure  $4.4.2 - 1^{st}$  and  $2^{nd}$  experimental steering wheel models' comparison.

# 4.5 Steering Plate free-free condition



Figure 4.5.1 – Steering Plate.

The Steering Plate was then tested variously in both free-free and clamped condition, the plate is made up of aluminium with a particular geometry. The numerical model that was constructed before is related to the Minecraft project [6]. However, an incompatible mass of the numerical model was detected, moreover the numerical natural frequencies were way higher than the experimental ones. To resolve this, several modifications were applied to the model to make it more consistent to real one. Firstly, the numerical frequencies were about the double of the real one, despite the correct young modulus imposed to the model. So, to not decrease drastically the young modulus the real mass was doubled, and the young modulus was reduced to half. These are not totally physical number it is the choice of a compromise to reach a good approximated numerical model.

Table 4.5.1 – Steering Plate, old and new numerical models' parameters.

Model	density [kg/m <sup>3</sup> ]	Youg Modulus, E [GPa]
Old Plate	1365.5	70
New Plate	2731	36



Figure 4.5.2 – Steering Plate numerical model, various views.

# 4.5.1 Experimental – numerical post processing

Table 4.5.1.1 – Steering Plate free-free, experimental and numerical natural frequencies.

	Testlab	Optistruct		
Mode #	Experimental frequency [Hz]	Mode #	Numerical frequency [Hz]	
1	305.1	1	308.8	
2	339.4	2	351	
3	493	3	569.05	
4	778.6	4	810	
5	862.3	5	886.4	
6	990.2	6	1026	
7	1175	7	1180	
-	1297	8	1322	
8	1366	-	1343	
9	1432	-	1431	
-	1556	-	1521	
-	1615	9	1582	
-	1701	-	1588.4	
-	1762	-	1688.3	



These modes were calculated in Optistruct then imported and visualized in Lupos.

Figure 4.5.1.1 – Steering Plate free-free, 1<sup>st</sup> and 2<sup>nd</sup> structural frequencies.



Figure 4.5.1.2 – Steering Plate free-free,  $3^{rd}$  and  $4^{th}$  structural frequencies.



Figure 4.5.1.3 – Steering Plate free-free, 5<sup>th</sup> and 6<sup>th</sup> structural frequencies.



Figure 4.5.1.4 – Steering Plate free-free, 7<sup>th</sup> and 8<sup>th</sup> structural frequencies.



Figure 4.5.1.5 – Steering Plate free-free,  $9^{th}$  and  $10^{th}$  structural frequencies.



Table 4.5.1.6– Steering Plate free-free model AutoMAC.



Table 4.5.1.7 – Steering Plate free-free model MAC and MACW2 respectively.



Table 4.5.1.8 – Steering Plate free-free model MAC\_distW.

#### 4.5.2 Experiment Setup and sensor positioning

The experimental modal analysis is performed adopting free-free conditions, exciting the structure with roving hammer and evaluating the response through accelerometers, either mono-axial or triaxial. The Plate is fixed on a huge test rig machine supplied by soft springs that do not affect the component natural frequencies the excitations were done on 27 nodes along all the possible directions allowed by its geometry, obtaining as an overall 61 excitations to the system. The responses of 3 nodes are measured: 1, 13, and 20. The response for each node is averaged on a set of 3 responses acquired to give more repeatability and stability to each measurement. The experimental test was performed in 2021-09-21 and postprocessed at the same day on which the main activity was concluded.





Figure 4.5.2.1 – Experimental setup scenario.

Both pictures of the entire system, with different points of view and details of the hardware implemented and of their location are requested information.

Importance has the reference system (x, y, z) used, according also to suitable models. In case of already developed models, consider the same reference system.



Figure 4.5.2.2 – Overview of the experimental setup.

In order to have a good sensor positioning MoGeSec was again exploited to define the sensors nodes as in Equation (4.1.2.1), (4.1.2.2), and (4.1.2.3).



Figure 4.5.2.3 – Steering Plate MoGeSec configuration.

As can be noticed various nodes were identified by MoGeSec, where these accelerometer we fixed by adhesive material in their exact positions, alternatively the positions and orientations will be selected according to the engineers experiment and knowledge, hence the following nodes were chosen accordingly with MoGeSec suggestions.



Figure 4.5.2.4 – Details of the implemented hardware, from left to right in reading order: Tri-axial-node 1, Tri-axial-node 13, Tri-axial-node 20.

The acquisition is performed with a LMS SCADAS Mobile system. A roving hammer EMA is performed using an impact hammer PCB 086C03 using hard steel tip (Figure 4.1.2.4 and

Figure 4.1.2.5) to reach a good frequency identification range of the structure. A series of 3 hammer repetitions is linearly averaged. The characteristics are listed in Table 4.1.2.1.

Component	Channel ID	Accelerometer ID	Node ID	Weight [kg]	Sensitivity [mV/g]	Acc. Direction [- ]
LW226918	8	Tri_3_x		14.2.10-3	102	-X
	9	Tri_3_y	1		102.1	-Z
	11	Tri_3_z			102.3	-Y
LW226919	1	Tri_1_x	13	14.2.10-3	102.5	-X
	2	Tri_1_y			97.5	-Y
	3	Tri_1_z			98.6	+Z
LW139287	4	Tri_2_x		14.2.10-3	96.1	-X
	5	Tri_2_y	20		97.3	-Y
	6	Tri_2_z			99.3	+Z

Table 4.5.2.1 – Accelerometer technical characteristics.



Figure 4.5.2.5 – Plate weight.

Table 4.5.2.2 – Plate mass properties.

Component	Weight [kg]	
Plate	4.923	

# 4.5.3 Geometrical aspects and acquisition parameters

To run the experimental modal analysis on the component it is necessary to create a model by identifying the most relevant nodes that allow to build a reference geometry as much close as the real one and at the same time could allow to give important information with regards to the response of the system.

The definition of the nodes map is carried out by obtaining the CAD geometry information of the chassis model developed in Solidworks then imported in Solidworks and Lupos, where the coordinates of each node are measured according to the reference system set on the component, as shown in Figure 4.5.3.1.

As an overall, 27 nodes are identified and used to develop the model geometry in Siemens LMS Test.Lab, as shown in Figure 4.5.3.2.



Figure 4.5.3.1 – Plate reference frame.

Figure 4.5.3.2 – Plate CAD geometry.

According to reference system and to numerical model, definition of points and corresponding model nodes are listed in Table 4.5.3.1. There were 27 nodes identified for roving hammer EMA. 3 of them are used for accelerometer positioning and hammered in the neighbourhood for auto-inertance. Node nomenclature is based on the following pattern:

- Plate external frame starting from node 1 in anti-clockwise direction until node 10;
- Plate large external wings from left to right (11 to 16);
- Plate small internal wings from left to right (17 to 27).



Figure 4.5.3.3 – Experimental geometry: excited (left) and sensor nodes (right).



Figure 4.5.3.4 – Node nomenclature.

Details of the experimental geometry are supplied in Table 4.5.3.1 in which directions of actuation and reference system for hammer direction are listed. As a reference approach, the hammered directions are always orthogonal to nearest surface to defined nodes, some nodes however had a defined orientation with respect to the global reference frame however, Euler angles were not imposed or considered.

Identifier	Exp. point	Model node	<i>x</i> [m]	<i>y</i> [m]	<i>z</i> [m]	Notes
Acc. 3, <i>x</i> , <i>y</i> , <i>z</i>	1	113118	0	0.224	0.144	LW226918
-	2	366196	0.096	0.2	0144	-
-	3	636483	0.16	0.152	0.144	-
-	4	661488	0.184	0.12	0.144	-
-	5	678408	0.216	0.016	0.144	-
-	6	619376	0.208	-0.088	0.144	-
-	7	366296	0.104	-0.192	0.144	-
-	8	96344	0	-0.2	0.144	-
-	9	28942	-0.12	-0.184	0.144	-
-	10	28801	-0.176	-0.136	0.144	-

Table 4.5.3.1 – Geometry of I/O points.

	11	617081	0.415	0.075	0.393	-
-	12	659293	-0.216	0.064	0.144	LW226919
Acc. 1, <i>x</i> , <i>y</i> , <i>z</i>	13	659328	-0.176	0.152	0.144	-
-	14	68758	-0.128	0.192	0.144	-
-	15	68786	-0.072	0.04	0.04	-
-	16	68806	0.072	0.04	0.04	-
-	17	114018	-0.096	-0.08	0.04	-
-	18	139793	0	-0.08	0.04	-
-	19	160109	0.096	-0.08	0.04	-
Acc. 2, <i>x</i> , <i>y</i> , <i>z</i>	20	160091	-0.03	0.16	-0.10	LW139287
-	21	510291	-0.01	0.01	0.07	-
-	22	569583	-0.08	0.02	0.09	-
-	23	569282	-0.21	0.01	0.09	-
-	24	581911	-0.11	0.16	0.10	-
_	25	581891	-0.03	0.16	0.10	-
-	26	511638	-0.07	0.06	0.07	-
-	27	368225	-0.06	0.06	0	-

During impact testing measurements, the parameters in Table 4.5.3.2 are selected:

Table 4.5.3.2 – Independent and dependent parameters of acquisition.

Quantity	Symbol	Relationship	Value	Units
Total acquisition time	Т	-	1	S
Sample frequency	$f_s$	-	8192	Hz
Total samples	N	$N = f_s T$	8192	-
Bandwidth (max frequency)	$f_b$	$f_b = \frac{f_s}{2}$	4096	Hz
FRF frequency resolution	Δf	$\Delta f = \frac{f_s}{N} = \frac{1}{T}$	1	Hz
Number of spectral lines	$N_f$	$N_f = \frac{N}{2}$	4096	-

# 4.5.4 Experimental data analysis

The natural frequencies and damping ratios are extracted in the frequency range  $0\div1800$  Hz from Test.Lab v.17 Impact Testing environment where the values were extracted as shown in Figure 4.5.4.1 the model size was equal to 500 and values were reported relatively in Table 4.5.4.1.



Figure 4.5.4.1 – Stabilization FRF diagram.



Mode	Freq. [Hz]	ζ, damping ratio [%]	Description
1	305.1	0.123	1 <sup>st</sup> global bending YZ
2	339.4	0.415	2 <sup>nd</sup> global bending YZ
3	493	0.185	3 <sup>rd</sup> bending XZ-YZ
4	778.6	0.292	local bending YZ
5	862.3	0.418	4 <sup>th</sup> global bending XY-YZ
6	990.2	0.285	5 <sup>th</sup> global bending XY-YZ
7	1175	0.187	6 <sup>th</sup> global bending XY-YZ
8	1297	0.388	7 <sup>th</sup> global bending
9	1366	0.292	8 <sup>th</sup> global bending XZ-YZ
10	1432	0.36	9 <sup>th</sup> global bending
11	1556	1.88	10 <sup>th</sup> global bending
12	1615	0.599	local bending
13	1701	0.832	11 <sup>th</sup> global bending XZ-YZ
14	1762	1.02	local bending XZ-YZ







Mode 5 - 862.3 Hz, 0.41858 %



Mode 7 - 1175 Hz, 0.18772 %





Mode 6 - 990.2 Hz, 0.28538 %



Mode 8 - 1297 Hz, 0.38813 %





Figure 4.5.4.2 – Plate first 14 complex mode shapes.

Following the same MACX procedure as in Equation (4.1.4.1) the correlation between the complex modes was evaluated.



Figure 4.5.4.3 – Experimental mode shapes Auto-MACX.

Similarly, as in Equation (4.1.4.2) the MPC is considered again to check the modes complexity.

Table 4.5.4.2 – Modal P	hase Collinearity values.
-------------------------	---------------------------

Mode	MPC			
1	0.926			
2	0.867			
3	0.937			
4	0.868			
5	0.956			
6	0.972			
7	0.882			
8	0.884			
9	0.928			
10	0.952			
11	0.821			
12	0.821			
13	0.755			
14	0.848			
1				
	0			
5	•			
0				



Based on the obtained results, it is possible to indicate that the real identification may be sufficient in this case.









Mode 10 - 1432 Hz



Mode 12 - 1615 Hz





Figure 4.5.4.5 – Plate re-identified real mode shapes.



Figure 4.5.4.6 – Experimental Auto-MAC, between complex and real modes.

## 4.5.5 Post-processing

To check the quality of the results and the linearity of the component several ways can be performed to do so as auto-inertance Equation (4.1.5.1) or (4.1.5.2) and cross receptance following Maxwell approach Equation (4.1.5.3). In the following figures the auto-inertances experimental (solid blue line), synthetised with lower and upper residuals (LR and UR) (dash-dot red line) and synthetised with modal superposition that are presented as well. At low frequencies, the clamped structure is characterized by inertances that tends to 0 is noticed from the blue curve. At 0 Hz it was not available the suitable way to identify have an MS tending to the correct value. Initially, the auto-inertances were evaluated to check to the quality of done identification.



Figure 4.5.5.1 – Auto-inertance: E1x-R1x.



Figure 4.5.5.2 – Auto-inertance: E1y-R1y.



Figure 4.5.5.3 – Auto-inertance: E1z-R1z.



Figure 4.5.5.4 – Auto-inertance: E13x-R13x.



Figure 4.5.5.5 – Auto-inertance: E13y-R13y.




10<sup>3</sup>

10<sup>2</sup>

10<sup>1</sup>

10<sup>0</sup>



Figure 4.5.5.7 – Auto-inertance: E20x-R20x.



Exp E20y R20y Syn E20y R20y

~

Figure 4.5.5.8 – Auto-inertance: E20y-R20y.



Figure 4.5.5.9 – Auto-inertance: E20z-R20z.

To control the linearity of the structure it is convenient to check the receptances of the structure as in Equation (4.1.5.3) and the following results were obtained.



Figure 4.5.5.10 – MAX index in 300÷1800 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.5.5.11 – MAX index in 300÷1200 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.5.5.12 – MAX index in 300÷800 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.5.5.13 – Reciprocity test experimental inertance: E1x-R20y (left) E1y-R20y (right).



Figure 4.5.5.14 – Reciprocity test experimental inertance: E1x-R13y (left) E20x-R13y (right).



Figure 4.5.5.15 – Reciprocity test experimental inertance: E20x-R1y (left) E20z-R13z (right).

The experimental modal analysis carried out on the Plate gives as a final result the identification of 14 mode shapes in the frequency range  $0\div1800$  Hz, where the global mode shapes identified are reported.

The identification of the mode shapes can be considered complete and satisfactory, as it can be noted in the previous figures looking at the good correspondence among the auto-inertances and the Maxwell reciprocity comparing the respective experimental and synthetized FRFs.

## 4.6 Clamped plate

This Plate is used for Iveco truck vehicles was modelled with different elements in different software and there was a great convergence to the exact solution, and this was assured by the EMA. The experimental test was performed in 2022-03-04 and postprocessed at the same day on which the main activity was concluded.



Figure 4.6.1 – Numerical model of clamped plate.

## 4.6.1 Experimental – numerical post processing

The numerical and experimental post processing was not fulfilled for this numerical model since no consistency was found between the 2 models, this maybe be because of the conditions in which the experiment was performed. Since the numerical model has no errors or unusual aspects, where the numerical modes are clear and well understandable. However, the experimental modes were extremely mixed and complex. For example, some numerical modes are related only to the beams, differently the experimental modes always involve a deformation shape of the plate, while the plate in certain modes, as checked numerically, must not have any mode shape contribution.

This inconsistency may be caused by the experimental boundary conditions that was constructed, since the nodes were hammered and the data was acquired correctly, but probably some screws were not clamped sufficiently. This caused a chaos in the experimental mode shapes.

## 4.6.2 Experiment Setup and sensor positioning

The experimental modal analysis is performed adopting free-free conditions, exciting the structure with roving hammer and evaluating the response through accelerometers, either mono-axial or triaxial. The Plate is fixed on a huge test rig machine supplied by soft springs that do not affect the component natural frequencies the excitations were done on 55 nodes along all the possible directions allowed by its geometry, obtaining as an overall 104 excitations to the system. The responses of 3 nodes are measured: 1, 13, and 20. The response for each node is averaged on a set of 5 responses acquired to give more repeatability and stability to each measurement. For each response:

• the spatial coordinates are continuous for each instant of time;

- the time/frequency data and the modal content are acquired and identified with Siemens Test.Lab v.17;
- it evinces those modal coordinates enable approximate solutions, according to the number of modal coordinates used, thus in nonlinear problems this property can improve computational efforts in the best way.



Figure 4.6.2.1 – Experimental setup scenario.

Both pictures of the entire system, with different points of view and details of the hardware implemented and of their location are requested information.

Importance has the reference system (x, y, z) used, according also to suitable models. In case of already developed models, consider the same reference system.



Figure 4.6.2.2 – Overview of the experimental setup.





Figure 4.6.2.3 – Details of the implemented hardware, from left to right in reading order: Tri-axial-node 1, Tri-axial-node 13, Tri-axial-node 20.

The same procedure of free-free plate for sensor positioning was followed hence same nodes and similar accelerometers were used.

Component	Channel ID	Accelerometer ID	Node ID	Weight [kg]	Sensitivity [mV/g]	Acc. Direction [-]
	9	Tri_3_x			102	-X
LW226918	10	Tri_3_y	1	14.2.10-3	102.1	-Z
	11	Tri_3_z			102.3	-Y
	1	Tri_1_x		14.2·10 <sup>-3</sup>	102.5	-X
LW226919	2	Tri_1_y	13		97.5	-Y
	3	Tri_1_z			98.6	+Z
	4	Tri_2_x			96.1	-X
LW139287	5	Tri_2_y	20	$14.2 \cdot 10^{-3}$	97.3	-Y
	6	Tri_2_z			99.3	+Z

Details of the experimental geometry are supplied in Table 4.6.2.2 in which directions of actuation and reference system for hammer direction are listed. As a reference approach, the hammered directions are always orthogonal to nearest surface to defined nodes, some nodes however had a defined orientation with respect to the global reference frame hence Euler angles were also imposed precisely.

Гаble 4.6.2.2 –	Geometry	of I/O	points.
-----------------	----------	--------	---------

Identifier	Exp. point	Model node	<i>x</i> [m]	<i>y</i> [m]	<i>z</i> [m]	Notes
Acc. 3, $x, y, z$	1	113118	0	0.224	0.144	LW226918

_	2	366196	0.096	0.2	0144	_
-	3	636483	0.090	0.2	0 144	-
	<u> </u>	661488	0.184	0.12	0.144	
	5	678408	0.104	0.12	0.144	
	6	619376	0.210	-0.088	0.144	
	7	366296	0.104	-0.192	0.144	
_	8	96344	0.101	-0.2	0.144	_
-	9	28942	-0.12	-0.184	0.144	_
-	10	28801	-0.176	-0.136	0.144	-
	11	617081	0.415	0.075	0.393	_
-	12	659293	-0.216	0.064	0.144	LW226919
Acc. 1, <i>x</i> , <i>v</i> , <i>z</i>	13	659328	-0.176	0.152	0.144	-
-	14	68758	-0.128	0.192	0.144	-
-	15	68786	-0.072	0.04	0.04	-
-	16	68806	0.072	0.04	0.04	-
-	17	114018	-0.096	-0.08	0.04	-
-	18	139793	0	-0.08	0.04	-
-	19	160109	0.096	-0.08	0.04	-
Acc. 2, <i>x</i> , <i>y</i> , <i>z</i>	20	160091	-0.03	0.16	-0.10	LW139287
-	21	510291	-0.01	0.01	0.07	-
-	22	569583	-0.08	0.02	0.09	-
-	23	569282	-0.21	0.01	0.09	-
-	24	581911	-0.11	0.16	0.10	-
-	25	581891	-0.03	0.16	0.10	-
-	26	511638	-0.07	0.06	0.07	-
-	27	368225	-0.06	0.06	0	-
-	118	118	-0.418	0.0067	0.222	-
-	119	119	-0.4185	0.0067	-0.222	-
-	205	205	-0.1015	0.0067	0.222	-
-	214	214	-0.1015	0.0067	-0.222	-
-	408	408	0.1125	0.0067	0	-
-	416	416	0.1125	0.0067	-0.222	-
-	417	417	0.1125	0.0067	0.222	-
-	1020	1020	0.1815	0.0617	0.197	-
-	1022	1022	0.0815	0.7217	0.375	-
-	1023	1023	0.0815	0.7217	-0.375	-
-	1024	1024	-0.2985	0.7217	-0.375	-
-	1025	1025	-0.2985	0.7217	0.375	-
-	1026	1026	0.3315	0.7817	-0.237	-
-	1029	1029	0.3315	0.7817	0.237	-
-	1031	1031	0.1815	0.0617	-0.197	-
-	1033	1033	0.0015	0.8617	0.197	-
-	1038	1038	0.0015	0.8617	-0.197	-
-	2004	2004	-0.1258	0.7817	0.237	-
-	2015	2015	-0.2985	0.7217	0.0895	-
-	2020	2020	-0.2985	0.7217	-0.0895	-
-	2054	2054	0.0815	0.7217	0.0179	-
-	2074	2074	-0.1258	0.7817	-0.2370	-
-	2129	2129	0.0015	0.5423	-0.1970	-

-	2133	2133	0.0015	0.3988	-0.1970	-
-	2137	2137	0.0015	0.2552	-0.1970	-
-	2145	2145	0.0015	0.5423	0.1970	-
-	2149	2149	0.0015	0.3988	0.1970	-
-	2153	2153	0.0015	0.2552	0.1970	-

During impact testing measurements, the parameters in Table 4.6.2.3 are selected:

Table 4.6.2.3 – Independent and dependent parameters of acquisition.

Quantity	Symbol	Relationship	Value	Units
Total acquisition time	Т	-	1	S
Sample frequency	$f_s$	-	8192	Hz
Total samples	N	$N = f_s T$	8192	-
Bandwidth (max frequency)	$f_b$	$f_b = \frac{f_s}{2}$	4096	Hz
FRF frequency resolution	Δf	$\Delta f = \frac{f_s}{N} = \frac{1}{T}$	1	Hz
Number of spectral lines	$N_{f}$	$N_f = \frac{N}{2}$	4096	-

## 4.6.3 Experimental data analysis

The natural frequencies and damping ratios are extracted in the frequency range  $100 \div 4096$  Hz from Test.Lab v.17 Impact Testing environment where the values were extracted as shown in Figure 4.6.3.1 the model size was equal to 1300 and values were reported relatively in Table 4.6.3.1.



Figure 4.6.3.1 – Stabilization FRF Sum diagram.

Table 4.6.3.1 – Independent and dependent parameters of acquisition.

Mode	Freq. [Hz]	ζ, damping ratio [%]	Description
1	141.72	0.63	-

2	154.25	0.501	-
3	183.38	0.571	-
4	205.29	1.67	-
5	257.163	0.647	-
6	266.28	0.751	-
7	286.43	0.745	-
8	345.46	0.645	-
9	369.85	0.652	-
10	425.47	0.399	-
11	437.09	0.576	-
12	473.638	1.17	-
13	552.359	1.12	-
14	561.85	0.62	-
15	673.56	0.866	-
16	687.97	0.748	-
17	976.39	0.732	-
18	1005.43	0.893	-
19	1084.77	0.405	-
20	1098.95	0.455	-

Mode 1 - 141.7 Hz, 0.63038 %





Mode 2 - 154.3 Hz, 0.50166 %





Mode 4 - 205.3 Hz, 1.6765 %

























Axis z [m]

















Figure 4.6.3.2 – Plate first 14 complex mode shapes.

Following the same MACX procedure as in Equation (4.1.4.1) the correlation between the complex modes was evaluated.



Figure 4.6.3.3 – Experimental mode shapes Auto-MACX.

Similarly, as in Equation (4.1.4.2) the MPC is considered again to check the modes complexity.

Table 4.6.3.2 – Modal Phase Collinearity values.



Figure 4.6.3.4 – MCP values for each mode.

Based on the obtained results, it is possible to indicate that the real identification may not be sufficient in this case.

#### 4.6.4 Post-processing

The inertance is a frequency response function (FRF) that is defined in the frequency domain as the ratio between the acceleration and the external forces. To check the quality of the results and the linearity of the component several ways can be performed to do so as auto-inertance Equation (4.1.5.1) or (4.1.5.2) and cross receptance following Maxwell approach Equation (4.1.5.3).



Figure 4.6.4.1 – Auto-inertance: E1x-R1x.



Figure 4.6.4.3 – Auto-inertance: E13x-R13x.



Figure 4.6.4.2 – Auto-inertance: E1y-R1y.



Figure 4.6.4.4 – Auto-inertance: E13y-R13y.



Figure 4.6.4.5 – Auto-inertance: E13z-R13z.



Figure 4.6.4.6 – Auto-inertance: E20x-R20x.

Figure 4.6.4.7 – Auto-inertance: E20y-R20y.



Figure 4.6.4.8 – Auto-inertance: E20z-R20z.

To control the linearity of the structure it is convenient to check the receptances of the structure as in Equation (4.1.5.3) and the following results were obtained.





Figure 4.6.4.9 – MAX index in 100÷4096 Hz: on experimental receptances (left) and on synthetised receptances (right).

Figure 4.6.4.10 – MAX index in 100÷2500 Hz: on experimental receptances (left) and on synthetised receptances (right).



Figure 4.6.4.11 – MAX index in 100÷800 Hz: on experimental receptances (left) and on synthetised receptances (right).







Figure 4.6.4.13 – Reciprocity test experimental inertance: E13y-R20y (left) E13z-R20z (right).

500

1500

2000

Frequency [Hz]

2500

3000

1000

10

0

1.1

3500

3000

4000

The experimental modal analysis carried out on the Plate gives as a final result the identification of 49 mode shapes in the frequency range 100÷4096 Hz, of which 20 modes were shown. The identification of the mode shapes cannot be considered complete and satisfactory, despite the

good correspondence among the auto-inertances and the Maxwell reciprocity comparing the respective experimental and synthetized FRFs. This structure was particular and hard to identify or to get good compatibility with numerical modes, experimental modes were complex and not related only to plate but also to the beams of which the structure is made, consequently the experiment must be performed again paying attention to the clamping between each beam and the plate in order to get better results.

## 5. Whole steering assembly

IIIIIIIIII

500

1000

1500

2000

Frequency [Hz]

2500

10-4

In order to have a more complete understanding of the behaviour of the complex system in particular, the system composed of the steering wheel and its base support connected to the whole car system that was furnished by IVECO a new model configuration has to be designed where it is going to be under study by a new EMA. To explain, the new model brings out a configuration similar to the car base system that holds the steering system, so it shows how the wheel is mounted to its base and how was it recognized to achieve the suitable and convenient stiffness through the joints and beams.

4000

3500



Figure 5.1 – General view of whole system numerical model.

## 5.1 Model configuration

This configuration was designed before in order to perform the last experimental modal analysis tests. However, due to certain issues related to other activities in the laboratory certain parts that were available are no more available hence a different configuration was adopted as noticed in chapter 4.6.4.



Figure 5.1.1 – General view of whole system.

The system simulates a bench on which the whole system is mounted with Bosch beams found in the Polito laboratory and they were be exploited for the desired purpose.

Table 5.1.1 – Assembly components details.

Part name	<i>l,</i> Length [mm]	b, Width [mm]	h, Height [mm]	Material	Q
-----------	--------------------------	------------------	-------------------	----------	---

Bench	850	620	14	Steel	1
Beam 1	344	100	50	Aluminium	2
Beam 2	631	80	40	Aluminium	4
Beam 3	533	100	50	Aluminium	2
Side_plate	353	100	10	Steel	2
100041035636	-		-	Steel	1
Top plate	444	89	10	Steel	1
Bottom plate	444	89	10	Steel	1
Beam 4	500	80	40	Aluminium	1
Connector	-		-	-	2
Beam 5	493	80	40	Aluminium	2
Beam 6	340	40	40	Aluminium	1

In the previous table structure components with some details are reported. The undefined numbers or material is due to the complex part geometry or because a component is made up of two or more different materials.

Table 5.1.2 –	Used	material	properties.
---------------	------	----------	-------------

Material	ρ, Density [kg/m <sup>3</sup> ]	<i>E</i> , Young module [GPa]	v, Poisson coefficient [-]
Steel	7800	210	0.33
Aluminium	2790	75	0.3

#### 5.1.1 Bench



Figure 5.1.1.1 – Modelled and real bench respectively.

The bench adopted for the structure is symmetric with four small holes in the upper part with a diameter of 14 mm and the rest are M20 threaded holes. These holes will be exploited for mounting the whole structure basically to connect the beams with bench that will form the base of the system. The beams, however, are connected to each other by T-screws M8x60 family

#### **5.1.2 Connectors**

The connectors provided by "Item" company are suggested to be adopted for stiffening the structure and connecting the parts with each other since the given bench has a relative low number of holes of which low number are used, hence a low number of joints that results in a decreased stiffness. These connectors have holes of 8mm that sustains the desired connection.



Figure 5.1.2.1 – Isometric and internal views of connectors.

These connectors are used for clamping between beams, and base bench plate.

## 5.1.3 Steering wheelbase

The Wheelbase is going to be connected to the vertical beams by connecting the side bottom and top plates on it then clamping it on the beams through the passive holes present in the structure by means of M8x60 hex cap screws. The rest of holes however are present to guarantee a good connection between the base and plate. In chapter 4.3.5 details are reported

## 5.1.4 Beams

The beams used in the assembly are those present in the laboratory provided by Bosch. Different beam sections are present hence different mechanical properties can be identified.



Figure 5.1.4.1 – Beam different sections.

The left section beams, being the stiffest, have been used to withstand the column and its base with the plates, while the right section beams have been used for the anterior part of structure. Hence the

the stiffest beam was fixed in the front part of structure to achieve the suitable stiffness that withstand the wheelbase structure.

# Conclusions

To bring it too short, the numerical models that were available, were rechecked to confirm their results then further deep analysis for similar models were also discussed and compared with the previous models analysing the behaviour of each component with different configurations and the effect of the non-structural mass as rubber and boundary condition was studied deeply. About 11 different Steering Wheel numerical models, and various numerical models for the Steering Plate finally the clamped steering plate assembly and the whole assembly was modelled. So finite element method FEM was exploited strongly in both Optistruct and Lupos solvers, furthermore the validation of these models was studied deeply through a series of experimental modal analysis. These tests were performed on the real components of each numerical model, certain models were mounted on certain machines or other structures as the modelled structure using CAD/CAE software, as Solidworks. Then these numerical models were adjusted accordingly in order to increase their consistency, then Modal Assurance Criterion was used for consistency verification. Adding to that, by post processing the extracted experimental data using Matlab, further analysis was achieved to check the structure behaviour and its dynamic characteristics using efficient tools such as inertance and receptance. Not less important the tools furnished by Lupos, such as MoGeSec tool in order to have a high experiment quality by identifying the main nodes where accelerometers must be mounted.

Last but not least the clamped steering plate experimental test must be rechecked and the final experiment for the whole assembly that is composed of Steering Plate Steering bar, and Steering Wheel is now ready to be tested.

# Reference

[1] Averame S., *Numerical and experimental tools for NVH characterization of a steering-column*, MSc Thesis, Politecnico di Torino, Torino, Italy, 2020.

[2] https://altairuniversity.com/wp-content/uploads/2017/03/1D\_Elements\_Extract.pdf available in 2021-11-30.

[3] https://forum.altair.com/applications/core/interface/file/attachment.php?id=4132 available in 2021-11-30

[4] Bonisoli E. "LUPOS Lumped Parameter Open Source FEM code Tutorial", *Politecnico di Torino, department of Mechanical and Aerospace Engineering* 

- [5] Allemang R.J., "The modal assurance criterion (MAC): Twenty years of use and abuse", *Proc. 20<sup>th</sup> IMAC*, 2002, Los Angeles, California, February 4-7, 2002, pp. 1-9. Allemang\_The modal assurance criterion (MAC)\_Twenty years of use and abuse\_IMAC2002.pdf
- [6] Bonisoli E., Delprete C., Rosso C., "Proposal of a modal-geometrical-based master nodes selection criterion in modal analysis", *Mechanical Systems and Signal Processing*, 23(3), April 2009, ISSN: 0888-3270, DOI: 10.1016/j.ymssp.2008.05.012, pp. 606-620. Bonisoli, Delprete, Rosso\_Proposal of a modal-geometrical based master nodes selection criterion in modal analysis\_MSSP2009.pdf

[7] Reduto Fernandes A., *MineCraft project : Iveco steering column base*, BSc Thesis, Politecnico di Torino, Torino, Italy, 2021.