Graz University of Technology

Diploma Thesis

Analysis and design of a functional 3D-CAD seat model

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Acknowledgements

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Contents

1. Abstract

This thesis is focused on the development of an automotive mass production seat 3D-CAD model using the CATIA-V5 software package.

CATIA-V5 -Computer Aided Three-dimensional Interactive Application- is one of the world's leading high-end CAD/CAM/CAE software packages developed by the French company Dassault Systemes. CATIA-V5 allows you multiple benefits in the design process [1].

Studying in detail the entire kinematic system in view of different movement functionalities of a car seat, this work give as resulted a complete 3D-CAD seat model in which it is possible to display all its movements in the DMU Kinematics workbench of CATIA-V5.

Furthermore, in order to vary the geometry of the seat model if it is necessary, the model has been parameterized.

2. Nomenclature

FTG: Institute of Automotive Engineering.

TU Graz: Technische Universität Graz.

CAD: Computer-aided design

CAM: Computer Aided Manufacturing

CAE: Computer Aided engineering

CATIA: Computer Aided Three Dimensional Interactive Application

3D: Three-Dimensional space

DMU: Digital MockUp

3. Preface

This thesis treats the creation of a simplified parameterized 3D-CAD model of a car seat created with the CATIA-V5 software package. Due to the necessity to support variant studies and optimization cycles during conceptual development processes this parameterized model has been created in order to be adapted to any internal car space changing its geometry.

3.1. Origin of the Thesis

Currently, to be more competitive in the market, many companies in the automotive industry need to implement modern IT-supported engineering tools. One of these possibilities is the implementation of three-dimensional software for improving the design processes and to make them cheaper and more efficient.

CATIA-V5 in automotive industrial design improves the efficiency of product development processes and company's capacity to use product-related information. This enables companies to make better business decisions and deliver greater value to customers. It also allows collaboration across organizational and geographic boundaries, to improve supply chain communication, business process efficiency, and the ability to innovate.

3D-CAD design allows organizations to make a qualitative leap in innovative product design. It achieves this by reducing cycle times, the reduction of time spent in developing and cutting production costs all at the same time. All this translates into a reduction in the final price and highest quality of the product.

4. Introduction

In order to be as a reference in the first steps of the design process and also be a help for the designer to understand the most important requirements needs the final design , the complete seat model and all his functionalities are created using CATIA-V5.

In this project all components of a car seat are designed simplified based on a real car seat model and afterwards, its complete kinematics functionalities are deeply studied and implemented into the model using DMU Kinematics Workbench of CATIA-V5. Once the kinematic simulation is completed, the kinematic result is deeply studied and all the components of the seat are analyzed individually and jointly again.

When all the components of the seat have the desired shape and measurements, the complete parameterized model can be created according to the requirement of this study. Finally, several analyses can be performed using the different workbenches provided by CATIA V5.

4.1. Objective of this thesis

Taken as a reference a seat from a Toyota Land Cruiser, the first objective of this thesis is to build up a simplified 3D-CAD model of the seat. Once the seat has been performed, the next objective is to recreate the kinematic of the seat on the model. That allows the designer to have a very useful tool to studies the different functionalities, which can assume one particular seat. The final objective of this thesis is to parameterize the most important aspects, which can be changed in the development process of the seat. For this reason, parameters as the width, the length or the height of the seat will be created in the model in order to adjust it to any space.

5. Methodology

For creating the simply seat model, firstly all the components which are important in the real model are studied to simplify as much as possible the final parameterized model. Afterwards, studying the kinematics characteristics of all motions of the seat and measuring all the pieces, the example model is composed. In the same way, a sketch is created to have all reference points that must be respected. This last point will be quiet important for making a good assembly and simulation model.

Then it is possible to start with the design process of all the seat components as Parts in CATIA-V5. All the Parts which have been important in the seat design will be explained in this chapter.

Once all the components have been created, the assembly of all of them is the next step. Each component must be inserted according to its location regarding the coordinate systems of the Parts.

Using the assembly model as a reference, the simulation of all the functionality movement is created with the CATIA DMU-Kinematics Workbench. Each functionality movement is created in CATIA as Commands. All the mechanical Parts have relationship with this functionality. In addition, all the possible functionalities of the seat will be explained in this chapter. If the Kinematics Simulation of the seat has not the desired result, the Parts will be checked and redesigned.

Finally, the parameterization of the model can be performed using the Formula feature contained in the Tools bar of the CATIA-V5 workbench. It will define the required modifications into the parameters of the applied operations.

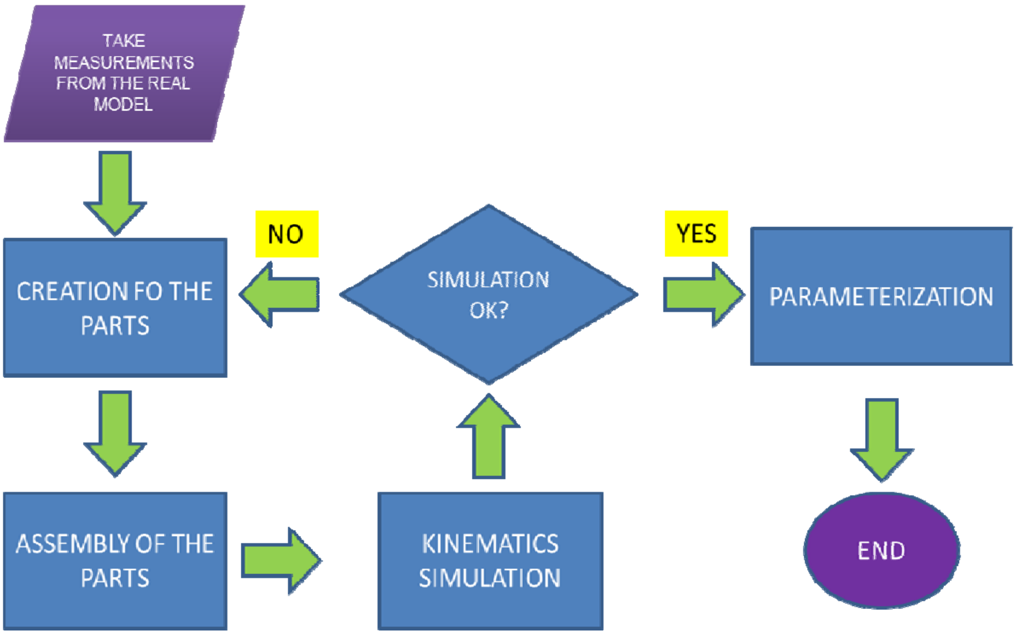


Figure 1 – Followed steps in the general design process.

5.1. Important Parts of the model

The body of a car seat is composed of several pieces which are not all represented the 3D-Model. This 3D-Model is composed of thirty-five Parts and one Sub-Product composed of two Parts more. Three of the Parts are covers and two are plastic shields. The rest of the most important Parts represented in the model will be explained separately under the Mechanisms which they belong or have more influence. The main Mechanisms taken into account on the model are the Longitudinal Translation Regulator (1), the Height Regulator (2), the Backrest Angle Regulator (3) and the Headrest Height Regulator as is shown in the picture bellow.

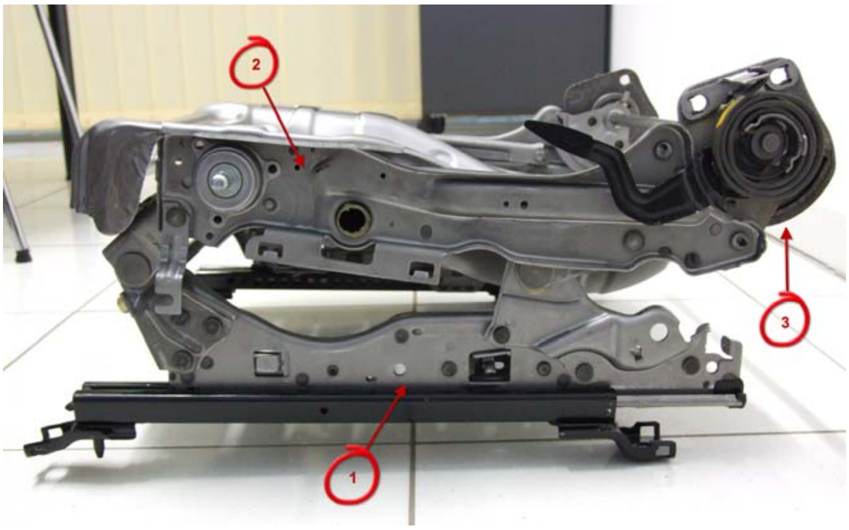


Figure 2 – Picture of the seat body used as reference.

5.1.1. The Longitudinal Translation Regulator Mechanism

Using this mechanism it is possible to vary the distance of the seat to the steering wheel thanks to a longitudinal movement between the body of the seat and guides anchored at the car body.

The most important parts to take into account in this mechanism are the Guide Base (1), the Slider (2), the Lock Base (3), the Lock Base 2 (4), the Lever (5) and the Lock (6) as represent Figure 2.

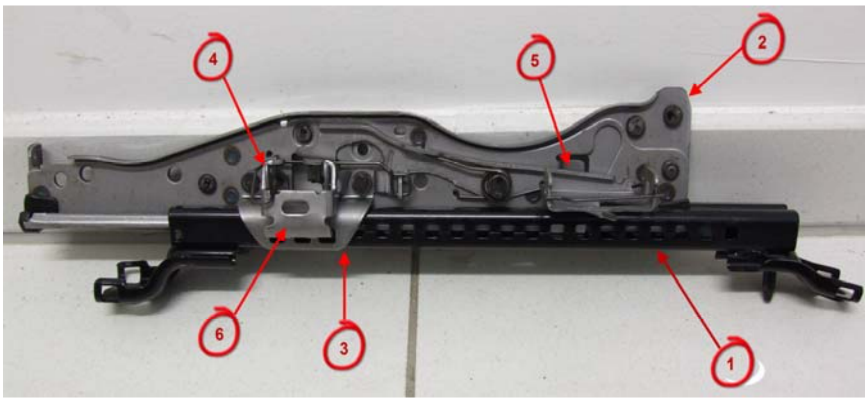


Figure 3 – Detailed picture of the Longiudinal Translation Mechanism used as reference.

5.1.1.1. The Guide Base

The guide base is one of the most important Parts to take into account on the model. The guide base is anchored on the car body and the rest of the seat car body will slide through it. As a main construction characteristic the Guide Base has some holes on the left side by which the final position of the seat will be fixed.

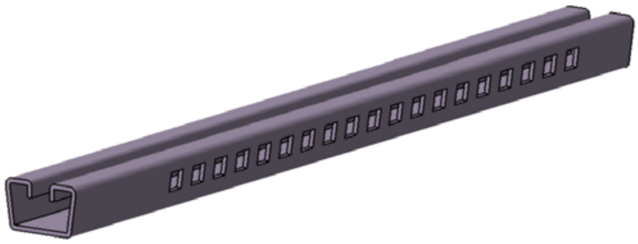


Figure 4 – Guide Base 3D-CAD model

5.1.1.2. The Slider

The slider is a Sub-Product in the model and is composed of two symmetrical Parts, the Slide Left and the Slide Right. The Slider will keep the seat's complete body and is the part which fits into the Guide Base and allows the sliding of the seat car body through the Guide Base.

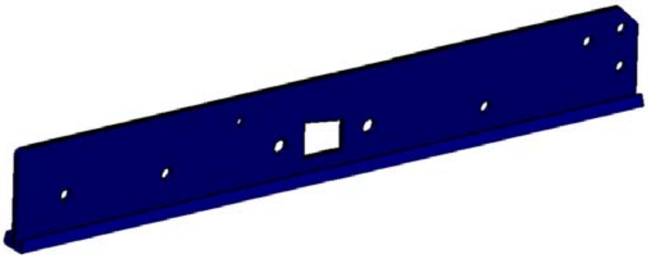


Figure 5 – Slider 3D-CAD model

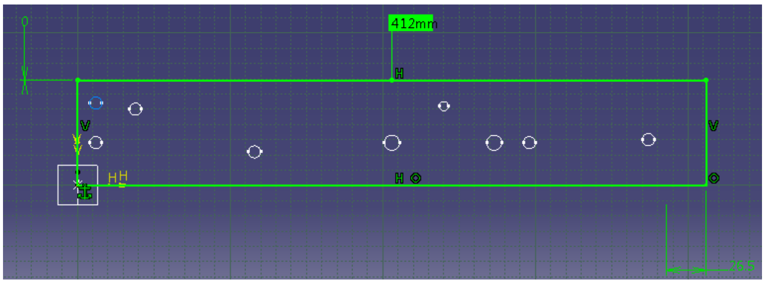


Figure 6 – Slider 3D-CAD model

5.1.1.3. The Lock Base

The lock Base is fixed to the Slider and its main function is to be as base of the Lock. The lock Base has as a main characteristic some holes located at the bottom by which they will allow to fix the final position.

Figure 7 –Lock Base 3D-CAD model

5.1.1.4. The Lock

The Lock performs the function of fixing the desired position of the seat in respect to the steering wheel. It is located on the Lock base and acts interlocking in the desired position of the Guide Base with the Slider. The Lock has as a main characteristic some teeth which penetrate through of the Lock Base and Guide Base holes.

Figure 8 –Different views of the Lock 3D-CAD model

5.1.1.5. The Lever 1 & 2

The Lever commands the mechanism. When the lever is turned slightly moves its movement to the Lock and pushing with the body the backrest of the seat it is possible to adjust the desired position of the seat in respect to the steering wheel.

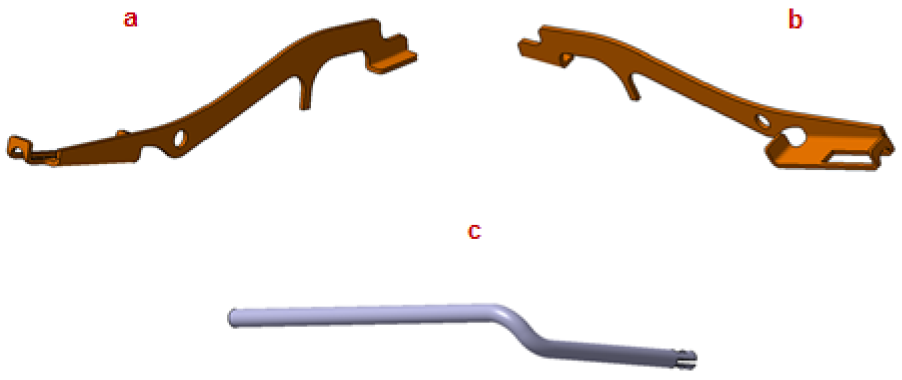


Figure 9 –(a)+(b) Different views of the Lever_1 3D-CAD model
(c) Lever_2 3D-CAD model

5.1.2. The Height Regulator Mechanism

The Height Regulator Mechanism allows the adjustment of the distance between the seat base and the car floor. The seat has several gears which convert their rotational movement into translational movement through their connection with the supports anchored in the Slider.

The most important Parts to take into account of this mechanism are the Gear 1, the Gear 2, the Gear 3 , the Gear 4, the Height mechanism bar, the Rear support 1 and 2, the Front support 1 and 2, and the Seat base.

5.1.2.1. The Gears 1, 2, 3 & 4

The Gears 1, 2 and 4 are located on several connection holes placed on the Seat Base and the Gear 3 is fixed to the Gear 2. When the Lever 2 is turned, the Gear 1 start to turn and transmit its rotational movement through the teeth to the Gear 2. In other words, when the Lever 2 is turned for adjusting the height of the seat, the Gear 1 leads over the rest of gears. Respectively, the Gear 2 translates its rotational movement to the Gear 3 which to become in solidarity with it. The Gear 3 translates its movement to the Gear 4 in which the rotational movement starts to be transformed in translational movement through the Height Mechanism Bar.

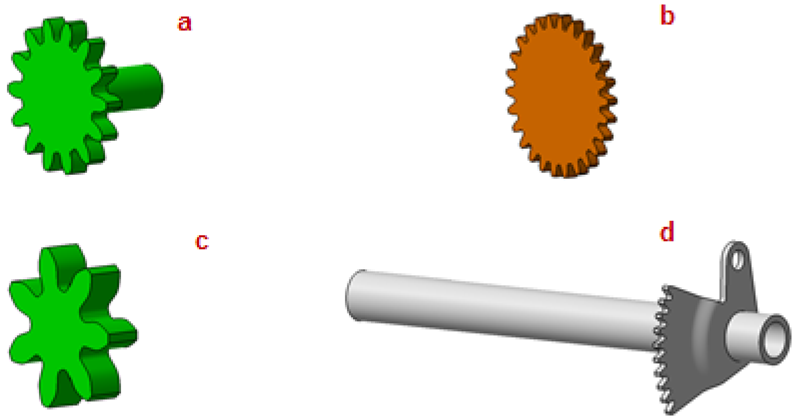


Figure 10 –(a) Gear_1 3D-CAD model
(b) Gear_2 3D-CAD model
(C) Gear_3 3D-CAD model
(d) Gear_4 3D-CAD model

5.1.2.2. The Height Mechanism Bar

The Height Mechanism Bar is the first responsible in the conversion of rotational movement of the gears in vertical translation movement of the Seat Base. Through the connection between the Gear 4 and the Height Mechanism Bar, the movement is translated to the Seat Base supports which rotating freely in the Seating Base and in the supports anchored in the Slider produces the height variation.

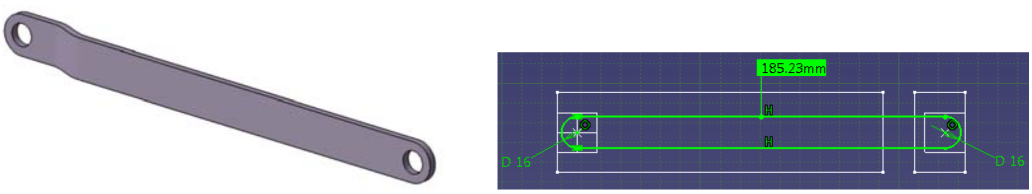


Figure 11 – Height Mechanism Bar 3D-CAD model and the main skech

5.1.2.3. The Front Support 1 & 2

The Front Support 1 & 2 are designed to connect the Slider to the Seat Base. The Front Support 1 has to remain immobile hence is fixed to the Slider by three connection holes due to has to support all the weight of the driver and the body of the seat next to the Rear Support 1.

The Front Support 2 is connected by one hole on the top of the Front Support 1 and another on the Seat Base. By comparison with the Front Support 1 the connection between them is not fixed. Accordingly, it can turns with geometrical limitations around the Front Support 1 and Seat Base connection holes.

Figure 12 – Front Suport 1 & 2 3D-CAD model

5.1.2.4. The Rear Support 1 & 2

The Rear Support 1 & 2 as Front Support 1 & 2 are designed to connect the Slider to the Seat Base. The Rear Support 1 has to remain immobile hence is fixed to the Slider by three connection holes due to has to support all the weight of the driver and the body of the seat next to the Front Support 1.

The Rear Support 2 is connected by one hole at the top of the Rear Support 1 and another at the Seat Base. By comparison with the Rear Support 1 the connection between them is not fixed. Accordingly, it can turn with geometrical limitations around the Rear Support 1 and Seat Base connection holes. Also, it is connected to the Height Mechanism Bar for translate the Gears movement to itself and the Front Support 2 and make possible the height variation of the Seat Base regarding the Slider.

Figure 13 – Rear Support 1 & 2 3D-CAD model

5.1.2.5. The Seat Base

The Seat Base is holds almost all the components of the Height Regulator Mechanism. Also, this Part is created in order to place the Cover of the seat.

Figure 14 – Seat Base 3D-CAD model

5.1.3. The Backrest Angle Regulator Mechanism

The Backrest Angle Mechanism is used for adjusting the angle of the backrest regarding to the seat base. It is composed of several parts which cooperate with each other for allowing the variation of this angle and be locked in the desired driver position.

The most important parts to take into account of this mechanism are the Backrest Lever, The Backrest Lever Base, The Backrest Lever Axis, the Backrest Mechanism Base 1 & 2, the Actuator, the Regulator and the Backrest.

Figure 15 – Detailed picture of the Backrest Angle Mechanism

5.1.3.1. The Backrest Lever

The Backrest Lever commands the mechanism due to its connection with the rest of the parts inside this mechanism. When the lever is turned all the parts of this mechanism are moved at the same time and then, keeping the Backrest lever in his final position allows the possibility to push the Backrest with the back for getting the desired position.

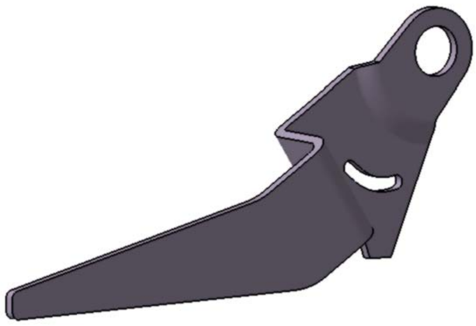


Figure 16 – Backrest Lever 3D-CAD model

5.1.3.2. The Backrest Lever Base

The Backrest Lever Base is where the Backrest Lever is located. Also its function is to be as Backrest Lever Spiral stand which allows the return of the Backrest Lever to its initial position.

Figure 17 – Backrest Lever Base 3D-CAD model

5.1.3.3. The Backrest Lever Axis

The Backrest Lever Axis connects all the parts and is the responsible to translate the rotational movement generated for the Backrest Lever to the rest of this parts inside this mechanism. It has different geometries along its length according to the function does in that exact point.

Figure 18 – Backrest Lever Axis 3D-CAD model

5.1.3.4. The Backrest Mechanism Base 1 & 2

In the Backrest Mechanism Base 1 are placed all the parts of this mechanism. Also, in its back side has a hole with teeth which are jointed with the Regulator, the Actuator and the Backrest Mechanism Base 2 allows to fix the final position. Has to be said that the Backrest Mechanism Base 1 it is motionless and the Backrest Mechanism Base 2 become in solidarity with the Backrest, it means that it has rotational motion.

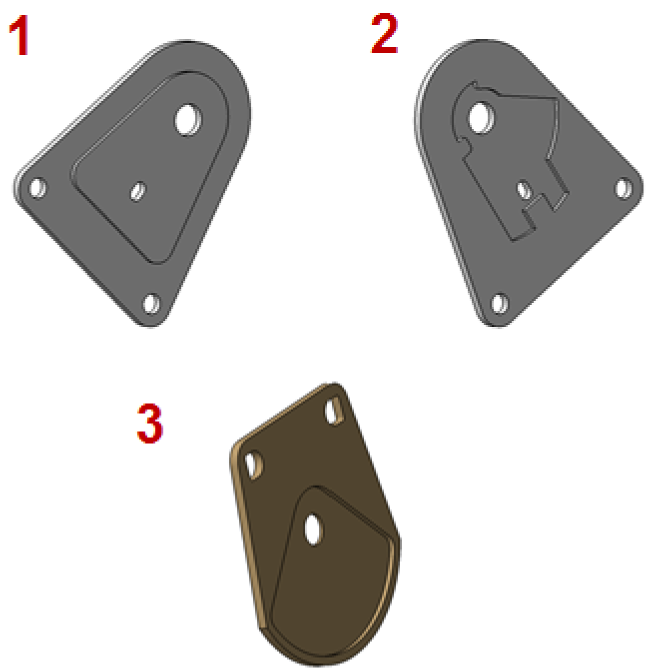


Figure 19 – (1)(2) Differents views of the Backrest Mechanism Base_1 3D-CAD model.
(3) Bacrest Mechanism Base_2 3D-CAD model.

5.1.3.5. The Actuator and the Regulator

The actuator is activated for the Backrest lever axis and it is responsible to keep and fix the Regulator with the teeth of the Backrest Mechanism Base 1. The Regulator is moved directly by the Backrest Lever because they are connected to each other.

On the other hand, they are located between the Backrest Mechanism Base 1 and 2.

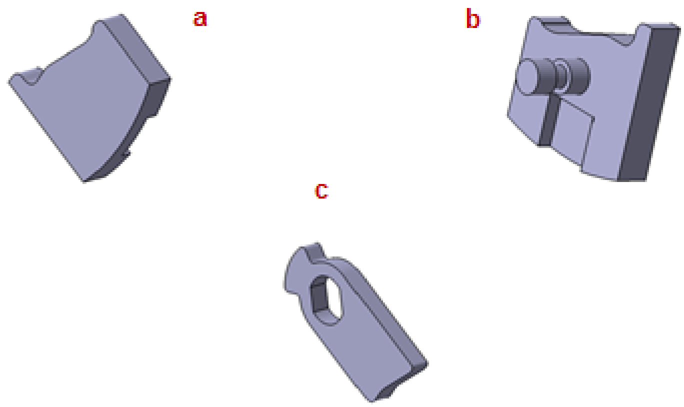


Figure 20 – (a)+(b) Different views of the regulator 3D-CAD model
(c) Regulator 3D-CAD model

5.1.3.6. The Backrest

The Backrest is the main part in this mechanism and one of the most important of the seat due to changing the backrest angle modifies the seat geometry.

Usually it is composed of an inverted U-shaped frame with two side arms having enough internal space for locating a plurality of flexible vertically disposed parallel strips fixed from the upper to the lower part of the frame. The strips have generally S-shaped configuration with a lower convex curve for supporting the lumbar region of the back and upper concave curve for supporting the upper dorsal region.

All the parts in this mechanism are designed to be able to change the angle of this part. Also, it is the part in which is located the cushions to be more confortable to the driver.

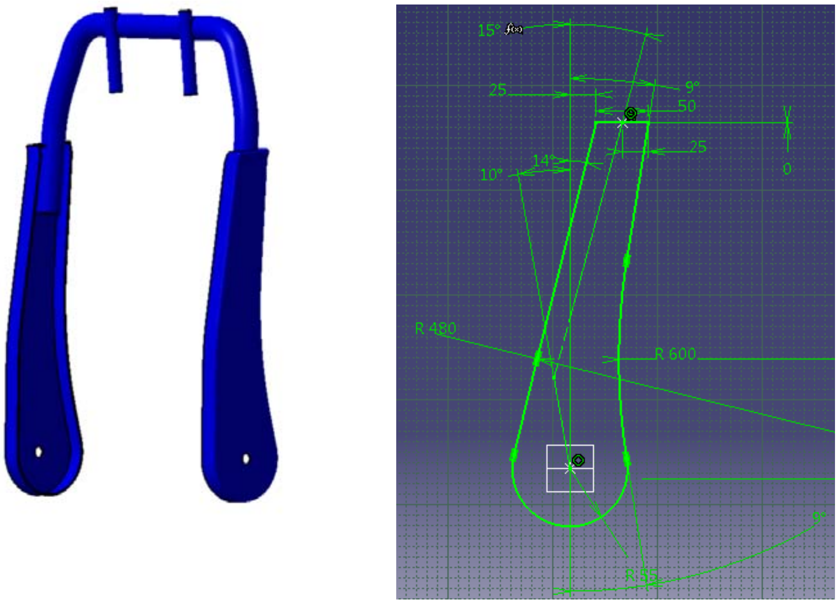


Figure 21 – Backrest 3D-CAD model and the main sketch

5.1.4. The Headrest Height Regulator Mechanism

Vehicle seats are typically provided with a headrest and it is desirable that the position be adjustable in the vertical direction to receive the proportional user head portion desired.

The Headrest Height Mechanism is used to adjust the headrest to the desired position. It consists of one part, the Headrest body, and it can be moved to up and down through two cylindrical guide holes located in the Backrest. This mechanism usually is positioned on the top of the Backrest.

5.1.4.1. The Headrest

The Headrest is designed as a frame of the headrest cushion and also it has a position shafts which are inserted in the two cylindrical guide holes to support the Headrest. These position shafts are parallel to each other and are assembled to the headrest frame.

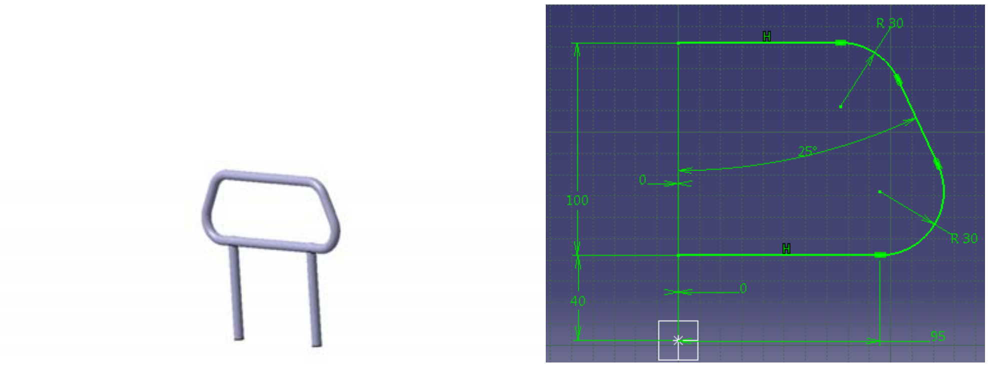


Figure 22 – Headrest 3D-CAD model and the main sketch

5.2. Creating the Seat Model with CATIA-V5

In order to have as support tool in future complex designs a simple seat CAD-model is performed. Usually, the shape complexity of the car seat it is an obstacle at the design process. The extreme difficulty to create a kinematic simulations and parameterizations in a final complex seat CAD-models may be a cause of mistakes in the seat design process. So as to be as useful tool for the designer, this thesis is focused in the kinematic simulation and the parameterization of the complete seat model. For this reason, all the parts into this model have been created as simple as possible.

5.2.1. Overview of Generative Shape Design Workbench in CATIA-V5

The CATIA-V5 Generative Shape Design Workbench is used to create advanced shapes based on complex surfaces. There are a lot of possibilities to reach the desired final model using combination of wireframe and surface features. Also it is possible to combine generated models created with Generative Shape Design workbench with features from Part Design workbench.

As a specific Generative Shape Design toolbars it is important to point out some of them.

5.2.1.1. The Wireframe toolbar

The Wireframe toolbar allows the creation of wireframe geometry such as points, lines and curves. As an example, features such as point, line, plane, circle, spine, intersection, projection or parallel curve can be found in this toolbar.

Figure 23 – Wireframe toolbar

5.2.1.2. The Surface toolbar

The Surfaces toolbar allows the user to model both, simple and complex surfaces. As an example, features such as extrude, offset, swept, fill, multi-sections and blend can be found in this toolbar.

Figure 24 – Surface toolbar

5.2.1.3. The Surface Operations toolbar

The Surface Operations toolbar allows the user to modify existing wireframe or surfaces using features such as Join, Split, Trim, Boundaries, Edge fillet, Translate or Extrapolate.

Figure 25 – Surface Operation toolbar

5.2.2. Creating the Parts of the Seat Model

The entire model contains thirty-five Parts and one Sub-Product. The Parts are: the Guide Base, the Lock Base 1 and 2, the Lock, the Lever 1 and 2, the Front Support 1 and 2, the Rear Support 1 and 2, the Seat Base, the Gear 1, the Gear 2, the Gear 3, the Gear 4, the Height Mechanism Bar, the Backrest Mechanism Base 1 and 2, the Actuator, the Regulator, the Backrest Lever Base, the Backrest Lever, the Backrest Lever Axis, the Cushion Base 1 and 2, the Backrest, the Headrest, the Gear 1 Cover, the Height Mechanism Lever, the Seat Cover, the Headrest

Cover, the Backrest Cover, the Height Connection Bar, the Backrest Mechanism Cover and finally the Skeleton. The Sub-Product will be the Slider and it is composed for the Slider Left and the Slider Right.

Due to the complexity of taking measurements directly from the real piece, the design process of the Parts has been carried out in three steps. First, it has been taken measures related to the entire model as the complete width, the length and the height. Secondly, in order to respect as much as possible these measurements, has been decided which pieces of each mechanism affects directly to these measurements if they suffer changes. Finally, these Parts have been designed trying to respect as much as possible the measurements taken from the real pieces and at the same time the desired measurements of the entire model.

In the next points will be explained the followed steps to build up the most important Parts of the model.

5.2.2.1. The Guide Base

The first Part designed in this mechanism is the Guide Base. It has been designed using the Generative Shape Design workbench of CATIA-V5. Firstly, the sketch of the half front view shape is done and extruded

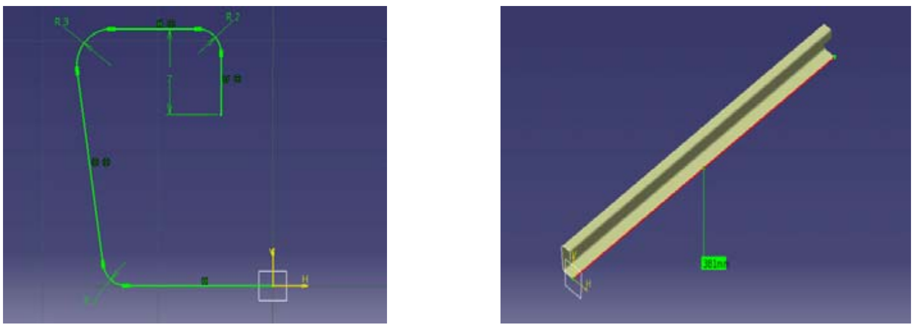


Figure 26 –Sketch + Extruded

Secondly, the extruded profile is transformed by symmetry.

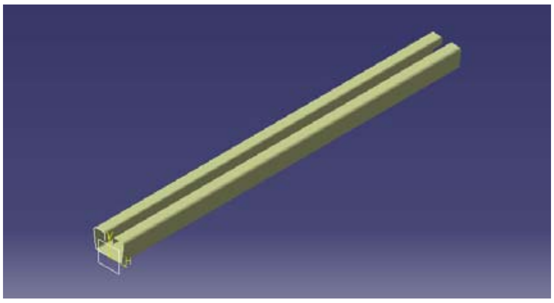


Figure 27 – Symmetry operation

Thirdly, the Sketch of the holes on the left side is done and extruded.

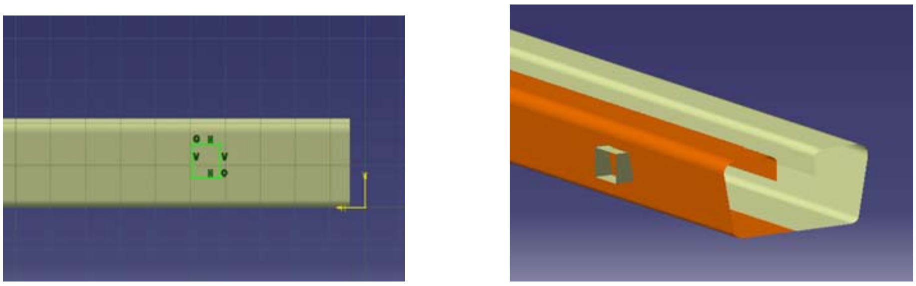
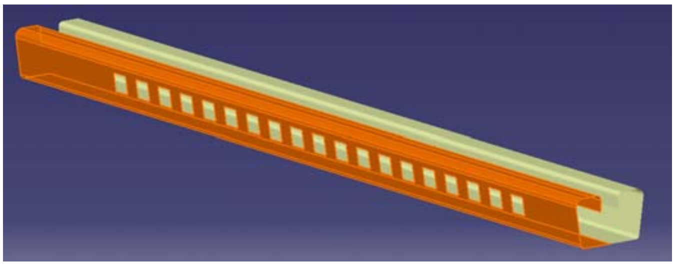


Figure 28 – Sketch and Split of the holes

Finally, the hole is copied several times along its length and in the Part Design workbench it is applied the thickness desired to its surface.



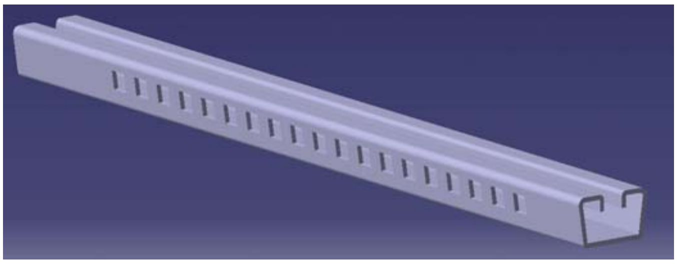


Figure 29 – Final surface + thin application

5.2.2.2. The Lock

The Lock has to be designed according few parameters. On the one hand, the width between the Lock's teeth has to be designed with the same width used between the side holes of the guide base. Likewise, the length of the teeth has to be enough to pass through the Locking Base and the Guide Base holes. Otherwise, the Lock could not perform the function which is designed.

On the other hand, when the Lever is turned the Lock has to define an enough arc to allow the movement between the Slider and the Guide Base.

Firstly, some important support points are created. In the same way, in order to define part of the final geometry the Sketch mode is activated and some 2D sketches are carried out.

The picture bellows represents these first operations.

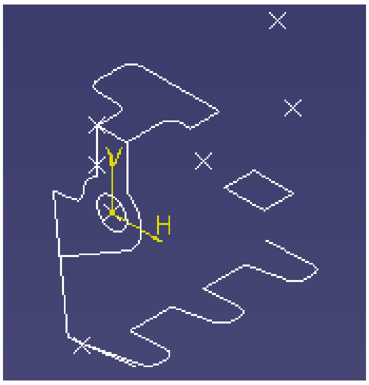


Figure 30 – Basic sketch

Secondly, some surfaces are created using features of the Surfaces tool as Extrude, Fill, Edge Fillet and Blend. At the same time, some changes are done in these surfaces using the Surfaces Operations tool as Split, Symmetry and Join.

The picture bellow shows the final result of applying surface operations.

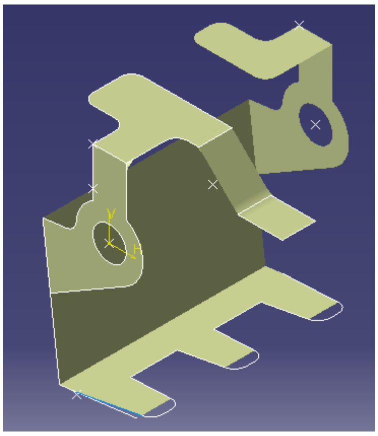
.

Figure 31 – Basic Surfaces

Finally, some changes and the thickness have to be applied. For this reason, it is necessary to change to Part Design workbench. Using tools as Thickness and edge Fillet is achieved the final shape of the Lock.

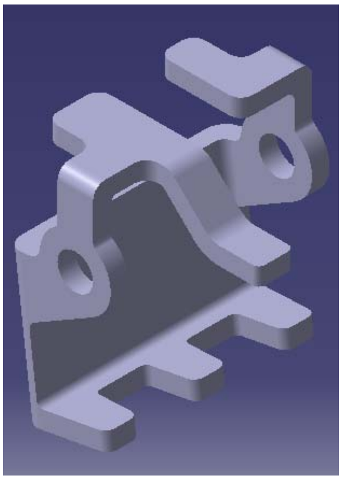


Figure 32 – Final application of thin

5.2.2.3. The Gears

Gears are quite complex machine elements and understanding them is quite difficult. Before explain the steps followed in the gear design with CATIA, one must be aware of what type of gear is going to be designed and which basic parameters are needed in order to design the elaborated aspects of them.

The table below shows the standards parameters and general formulas used to define the spur gear's geometry.

Parameter	Unit	Formula	Description
A	angle	20deg	Pressure angle: (10deg ≤ a ≤ 20deg)
M	mm	-	Module
Z	integer	-	Number of teeth (5 ≤ Z ≤ 200)

Parameter	Unit	Formula	Description
A	angle	20deg	Pressure angle: (10deg ≤ a ≤ 20deg)
M	mm	-	Module
Z	integer	-	Number of teeth (5 ≤ Z ≤ 200)
P	mm	m * π	Pitch of the teeth on a straight generative rack.
E	mm	p / 2	Circular tooth thickness, measured on the pitch circle.
Ha	mm	m	Addendum = height of a tooth above the pitch circle.
Hf	mm	if m > 1.25 hf = m * 1.25 else hf = m * 1.4	Dedendum = depth of a tooth below the pitch circle. Proportionnally greater for a small module (≤ 1.25 mm)
Rp	mm	m * Z / 2	Radius of the pitch circle
Ra	mm	rp + ha	Radius of the outer circle.
Rf	mm	rp - hf	Radius of the root circle.
Rb	mm	rp * cos (a)	Radius of the base circle.
Rc	mm	m * 0.38	Radius of the root concave corner. (m * 0. 38)is a normative formula.
T	range	0 ≤ t ≤ 1	Sweep parameter of the involute curve.
Xd	mm	rb * (sin (t * π) - cos (t * π) * t * π)	Y coordinate of the involute tooth profile, generated by the t parameter.
Zd	mm	rb * (cos (t * π) - sin (t * π) * t * π)	Z coordinate of the involute tooth profile.
Ro	mm	rb * a * π / 180 deg	Radius of the osculating circle of the involute curve, on the pitch circle.
C	angle	sqrt (1 / cos (a)² - 1) / π * 180 deg	Angle of the point of the involute that intersects the pitch circle.
phi	angle	atan (yd(c) / zd (c)) + 90 deg / Z	Rotation angle used for making a gear symetric to the ZX plane.

Figure 33 – Gear formulas [2]

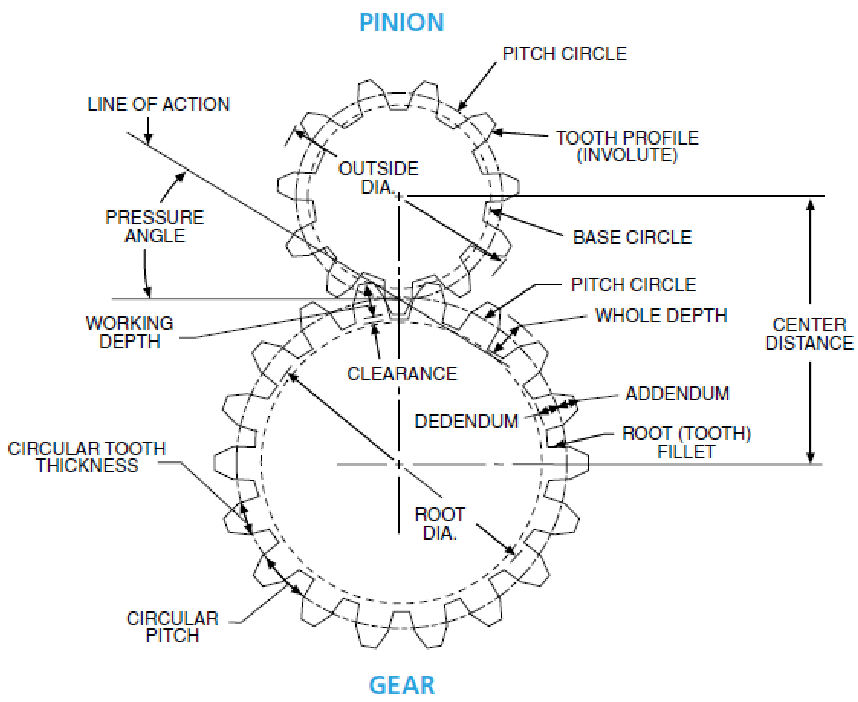


Figure 34 – Gear Sketch [3]

According to the table above and due to most of geometric characteristics are related to each other, it is possible to design a gear using a few parameters, and also they can be controlled by the followed parameters:

The module: M

The pressure angle: A

The number of teeth: Z

Then it is possible to start to create and edit parameters using the feature Formula, which is located in the menu Tools/ Knowledge, in the Generative Surface Design workbench of CATIA.

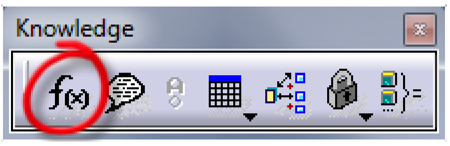


Figure 35 – Knowledge toolbar

At the beginning, the unit is selected and the create parameter button activated. The next step is to create a new parameter name and set the initial value if the parameter has a fixed value. On the contrary, the formula editor allows the parameter to be defined with a formula. Accordingly, using the table on the Fig.31 now the parameters a, m, Z, rp, ra, rb, rf and rc can be defined.

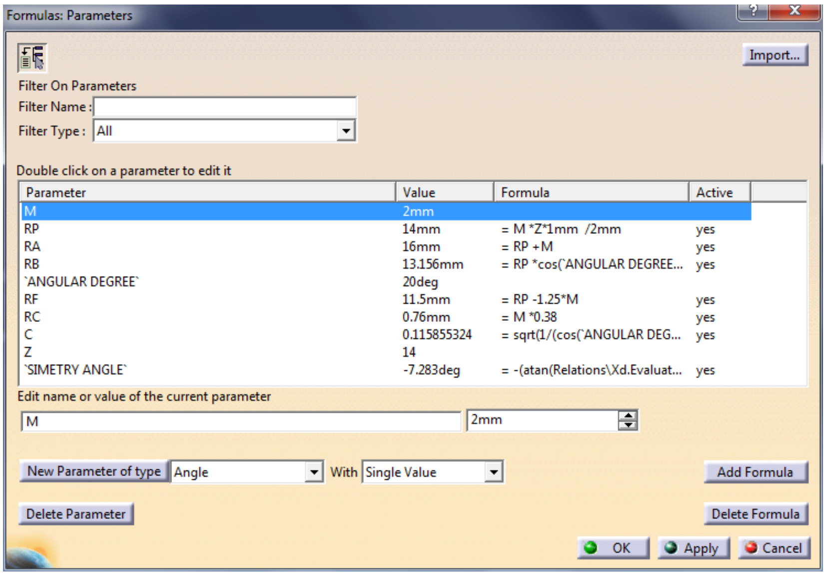


Figure 36 – Edit window of parameters

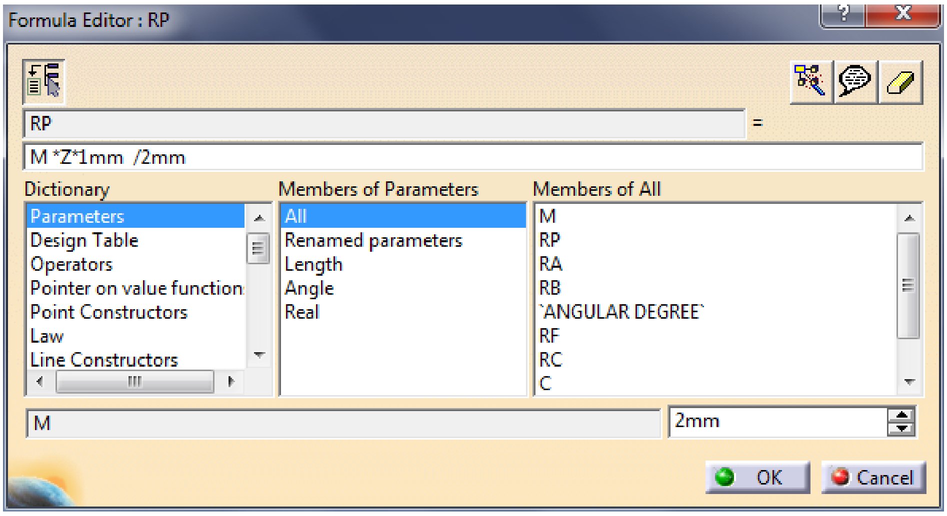


Figure 37 – Formula Editor window

Now, the formula of the Cartesian position of the points of the involute curve of a tooth has to be defined using the Parametric Laws tool.

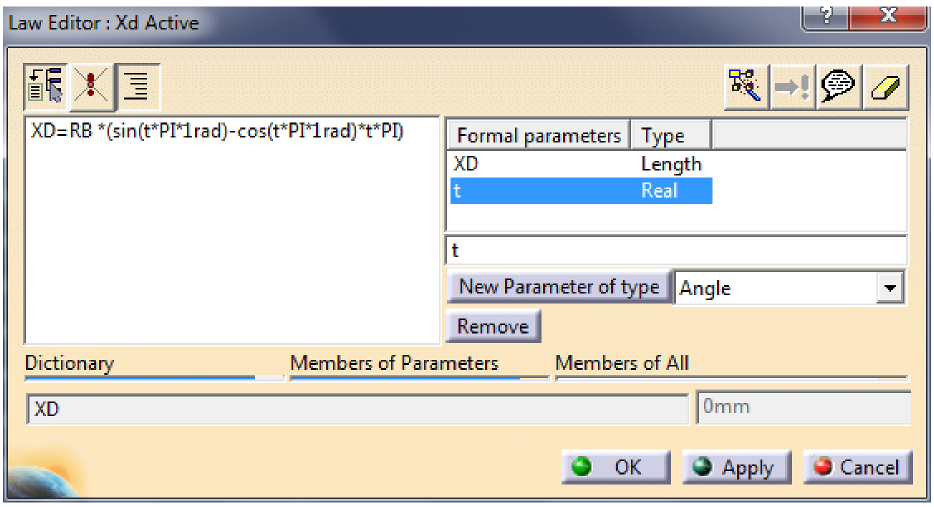


Figure 38 – Law editor window

In order to define the first tooth of the whole gear, 5 points have to be created. Each point will have the x and z coordinates according to the previously defined xd and zd coordinates and the t parameter from t=0 to t=0.4.

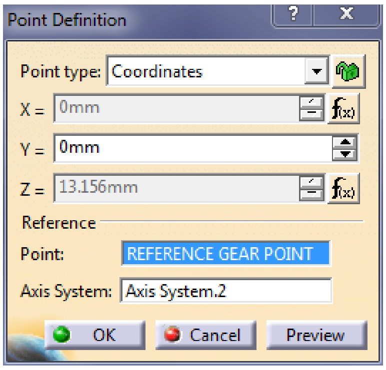


Figure 39 – Example of definition of one of the five points

When the points have been created should be generated a Spine through these points. Afterwards, the Spine curve has to be extrapolated for joining it with the root circle. The result of these operations is rotated from the Y axis as a reference by the c angle, previously created as parameter.

The figure bellow represents this process [2].

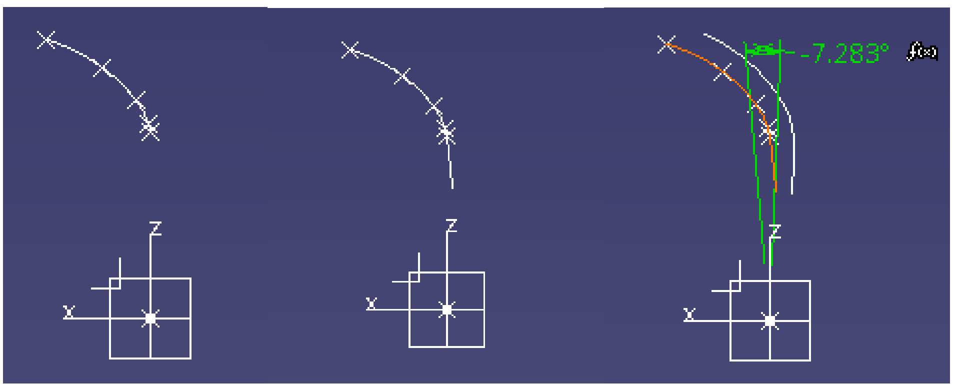


Figure 40 – Construction Example

At this point, two circles have to be created with the ra and rf parameters as a radius of each one. In addition, the rounder corner between the root circle and the extrapolated involute curve can be created using the rc parameter. This rc parameter will define the radius of this corner. Then, the intersections between the involute curve and the circle can be cut using the function split. The result of these operations can be transformed using the tool symmetry by the ZY plane.

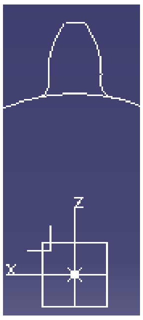


Figure 41 – Definition of the first tooth

Now, it has to be designed the symmetry plane between each tooth. The rotation angle of the ZY plane around the Y axis depends on the number of teeth and it is defined by the next formula:

Symmetry Plane Rotation Angle = 180º / Z [2]

When the plane has been created, the root circle can be cut with it. Next, using the Circular Pattern tool, can be defined the instances will be repeated this tooth around 360º. For this reason, has to be created a formula which makes a relation between the number of instances and the Z parameter.

Finally, the first tooth and the duplicated teeth are joined using the Join tool and the resulting profile can be extruded with the desired thickness.

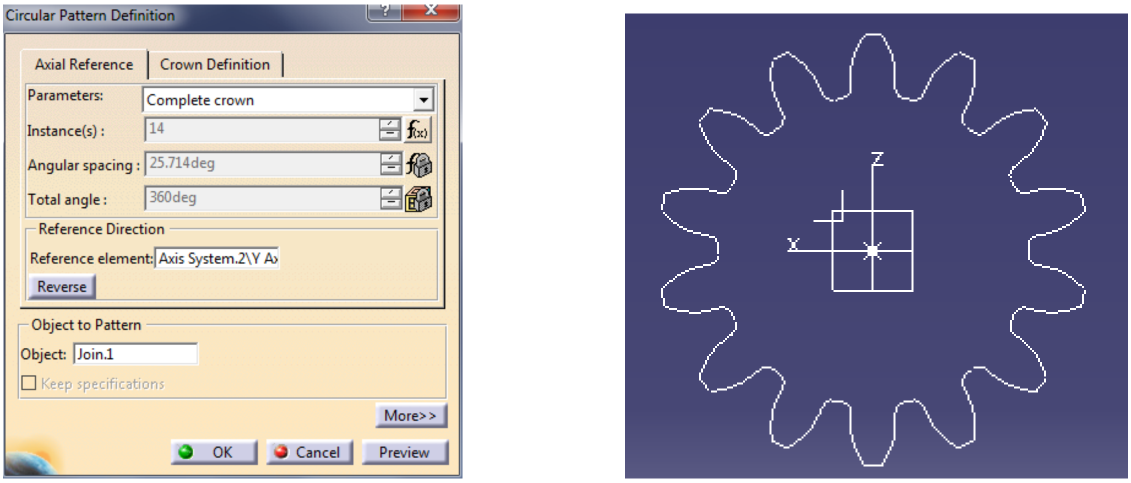


Figure 42 – Application and result of circle pattern

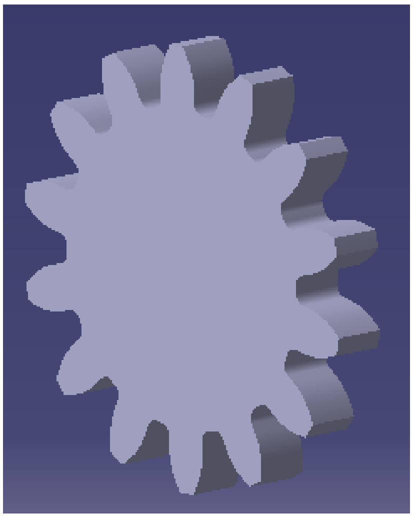


Figure 43 – Finished gear

5.3. The Assembly of the Parts

Once all Parts of the model have been created, the assembly of them can be performed. Using the Assembly Design Workbench is possible to bring together Parts, Products or sub-Products into an assembly as a CATProduct. CATProducts also can be composed of smaller CATProducts to carry out bigger assemblies. In addition, it is possible to use them in Kinematic Simulations converting the constraints used in the assembly design creation process [4].

So as to have a better idea of the most important features of Assembly Design, they are going to be explained in the next point.

5.3.1. Overview of Assembly Design Workbench of CATIA-V5

Before starting to explain the features of this workbench, it is necessary to take a look to the different nodes present in the CATProduct Specification Tree. The most commonly used are the Product, the Part and the Component as is shown in the picture bellow. They can be added to the Specification Tree through the Product Structure Tools toolbar.

Figure 44 – Different nodes found in the specification tree of the assembly

To maintain the position of the Products, Parts or sub-assemblies within the CAT-Product, Assembly Constraints are used. When a constraint is defined, it is attached to the Specification Tree under the Constraints Node. In order to create new constraints between elements inside the CAT-Product, the Assembly Constraints toolbar is used.

Figure 45 – Constraints toolbar

The table below describes the function of each one of the features found in the Assembly Constraints toolbar.

CONSTRAINTS		FUNCTION
Coincidence		Used to constraint the axis of two cylindrical features
Contact		Used between planes, faces, lines or points of different Parts
Offset		Define an offset between two different Parts
Angle		Define an angle between two different Parts
Fix Component		Fix the position of Parts
Fix together		Fix together multiple Parts
Quick		Quick application of constraints between features of different Parts
Flexible/Rigid Assembly		Allows to desactivate constraints temporally
Change Constraint		Change the type of the constraint
Reuse Pattern		Use a pattern from a Part and use it to position multiples instances of a Part in the Product

Figure 46 – Table of types of constraints [5]

After the assembly of the desired Parts, the Space Analysis toolbar offers the possibility of make analysis using the Clash Analysis feature. Once the Clash Analysis is selected, the control panel of the feature allows you specify the type of check and where is going to be performed.

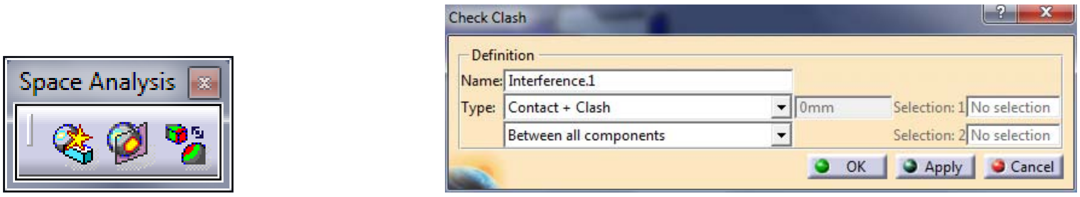


Figure 47 – Space Analysis toolbar and definition window of Clash Analysis

5.3.2. Creating the Assembly of the Parts

The assembly has to be created with all the components previously explained in the point 5.1 “Important Parts into the model”. In that case, in order to make the model more stable, all the Parts have been designed according to the final position. For that reason, Fix constraints have been attached to all the components in order to maintain the position. The resulting Specification Tree, the mechanisms and the complete seat after the assembly are shown in the picture bellow.

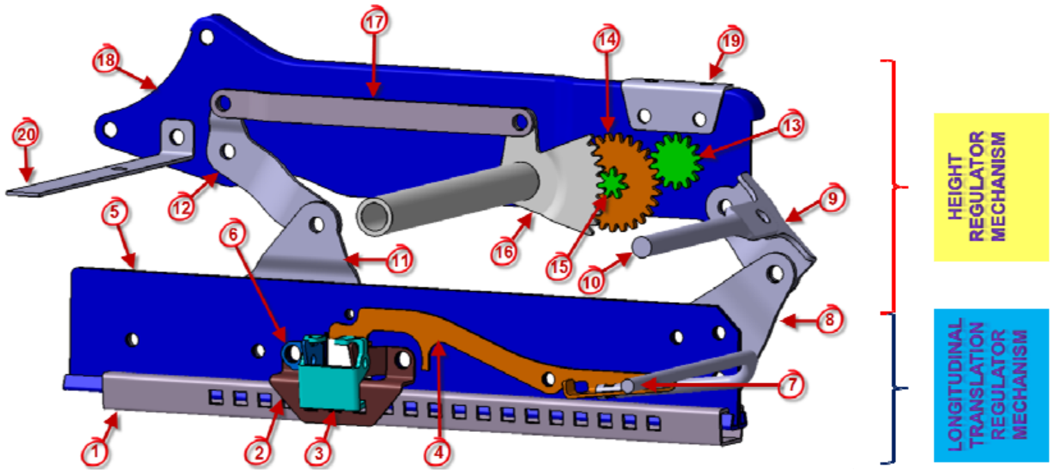


Figure 48 – Final assembly of the components of the Longitudinal Translation Mechanism and the Height Regulator Mechanism.
(1).Guide Base, (2).Lock Base, (3).Lock, (4).Lever_1, (5).Slider, (6).Lock Base_2, (7).Lever_2, (8).Front Support_1, (9).Front Support_2, (10). Height Connection Bar, (11).Rear Support_1, (12). Rear Support_2, (13). Gear_1, (14).Gear_2, (15). Gear_3, (16). Gear_4, (17). Height Mechanism Bar, (18). Seat base, (19). Cushion Base_1, (20). Cushion Base_2

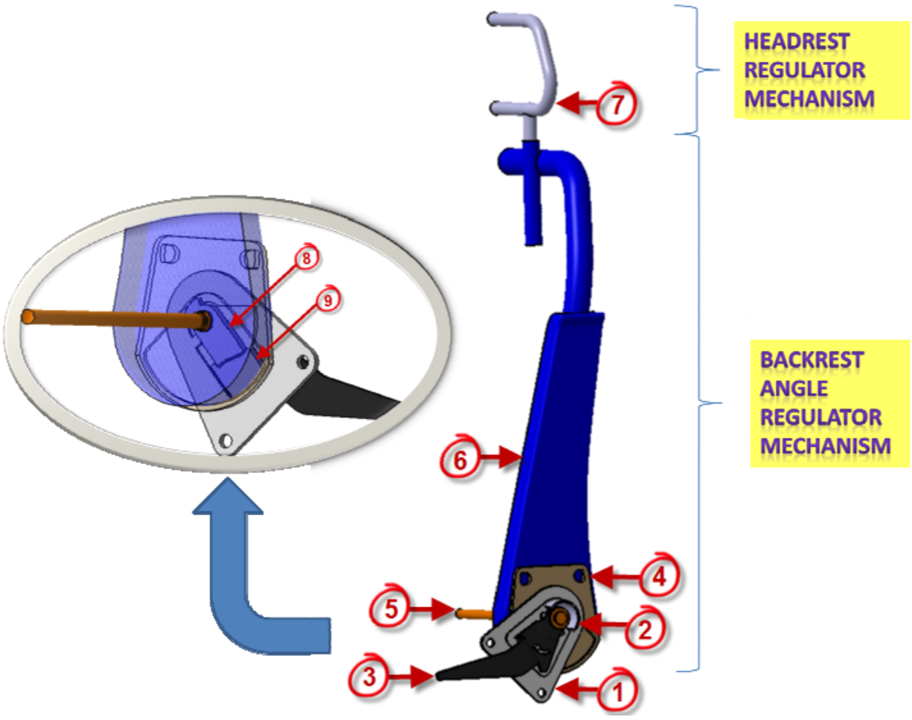


Figure 49 – Final assembly of the components of the Backrest Angle Regulator Mechanism and the Headrest Regulator Mechanism.
(1). Backrest Mechanism Base_1, (2). Backrest Lever Base, (3). Backrest Lever, (4). Backrest Mechanism Base_2, (5). Backrest Lever Axis, (6). Backrest, (7). Headrest, (8). Actuator, (9). Regulator

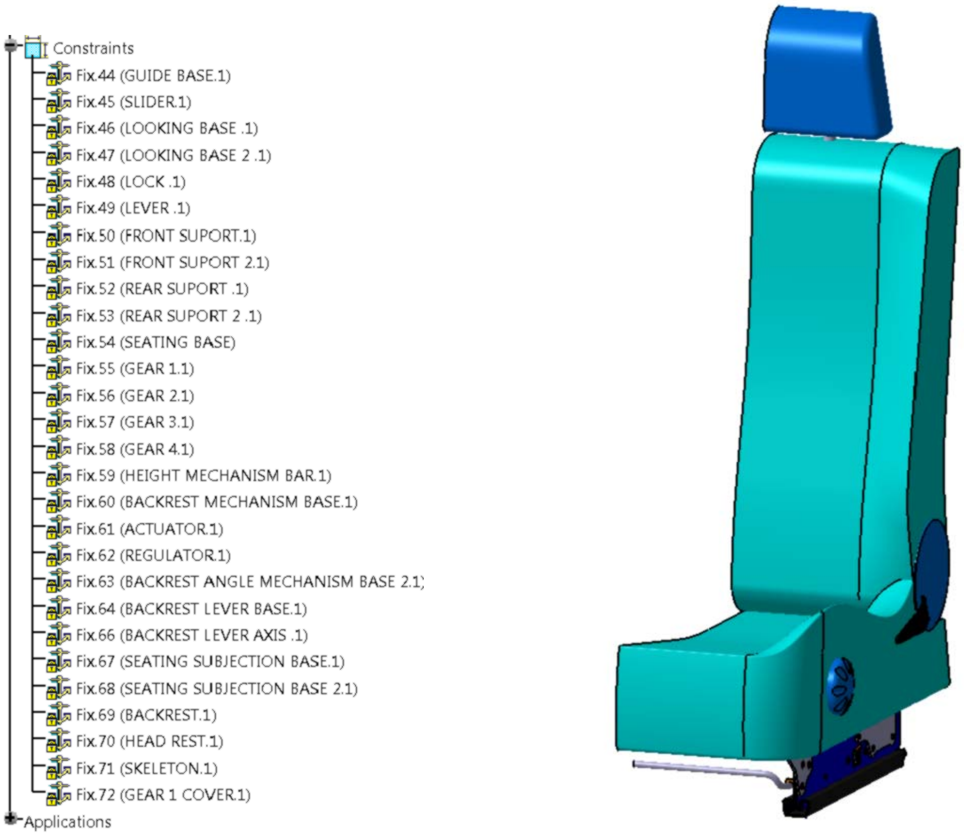


Figure 50 – Final Specification Tree and Seat Model

5.4. The Kinematics Simulation with CATIA

Kinematics is the branch of classical mechanics that describes the motion of objects or particles according to the time without consideration of the causes leading to the motion. Kinematics use usually a coordinate system to describe the trajectories, called the reference system. The function that describes the trajectory depends on the speed and acceleration.

Kinematics simulation can show a wide variety of design issues that may occur before the manufacturing process. In particular, the kinematic simulation allows analysis of the relative movement between two parts, to detect possible interferences or calculate velocities and accelerations.

For that purpose, it is possible to put the Assembly into motion using DMU Kinematics Workbench of CATIA-V5.

5.4.1. Overview of DMU Kinematics Workbench of CATIA-V5

The DMU Kinematics Workbench is a separated design module which is dedicated to simulate the Assembly movements. Using this workbench, it is possible to define kinematics mechanisms, simulate movement mechanisms and to detect collisions.

Next, some of the most important terms used in this Workbench will be explained.

5.4.1.1. The Kinematics Mechanism

Kinematics mechanism is a series of rigid elements connected with joints to form a closed chain, or a series of closed chains. Each element has two or more joints, and the Joints have various degrees of freedom to allow the motion between the links. The kinematics mechanism is usually used to convert or transmit the movement. Every mechanism has a Fixed Part through the movement occurs.

5.4.1.2. The Degrees of Freedom

An isolated body that moves freely in the three dimensional space has three independent rotations and three translation displacements according to a fixed axis which defines the three directions of a base referred to a three dimensional space. For instance, a piece on a plane can be moved in vertical and horizontal and any other translation is a result of these two components. Also, it can rotate on itself. Therefore, if this piece has not any restriction it will have three degrees of freedom.

In that case, in order to define the degrees of freedom of the two dimensional mechanism, it can be calculated through the Grübler-Kutzbach equation.

$$m = 3(n-1) - 2j1 - j2 \quad [6]$$

Where: m= mobility

n= number of elements of the mechanism

j1= number of one degree of freedom joints

j2= number of two degree of freedom joints

5.4.1.3. The Joints

The unions between two members of a mechanism which can move with respect to each other are connected together through kinematic joints. This union creates a Kinematic Pair and the properties of joints between them such as the relative motion and degrees of freedom determine the resultant motion of a mechanism. For this reason, the joints are one of the most important parts of a mechanism.

The kinematic Joints can be classified into two categories based on the type of contact between them as Lower Pairs or Higher Pair. Likewise, when the connections are reduced to simple forms of construction, the Lower Pairs are divided into six types of connections. The foregoing six types of connections are shown in the picture bellow. The common denominator of these connections appears to be the area contact between links.

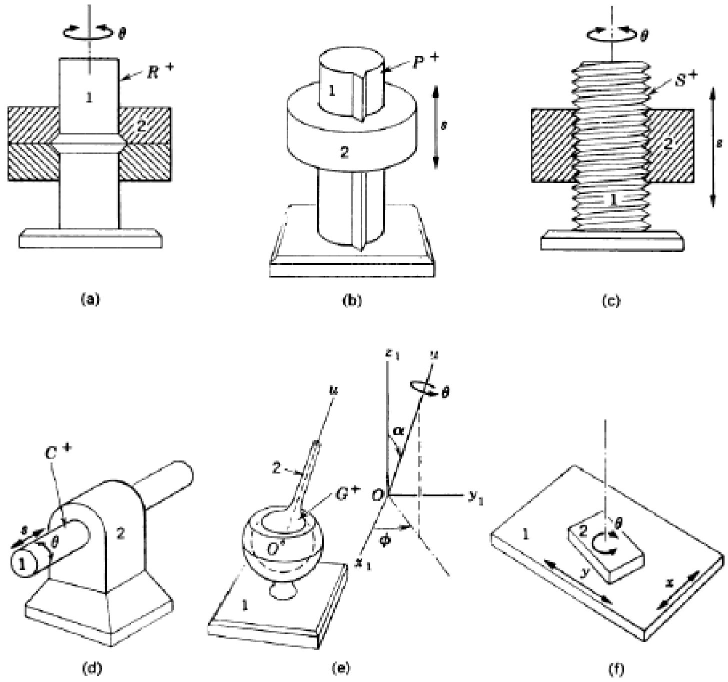


Figure 51 – Types of Joints [7]

In the Higher Pairs connections the contact between the two members is a point or line geometry. Point contact is found in ball bearings and the line contact is characteristic of cams, roller bearings and most gears. Higher Pairs connections on occasions can be replaced by a combination of Lower Pairs.

DMU-Kinematics Workbench of CATIA-V5 offers sixteen different joint types. These joints and their characteristics are shown in the picture below.

V5 NAME		DOF	COMMAND TYPE
Revolute		1 Rotation	Angle
Prismatic		1 Translation	Lenght
Cylindrical		1 Rotation 1 Translation	Lenght + Angle AND/OR Angle or Lenght
Spherical		3 Rotations	-
Planar		2 Translations	-
Rigid		-	-
Roll Curve		1 Rotation 1 Translation	Lenght
Slide Curve		2 Rotations 1 Translation	-
Point Curve		3 Rotations 1 Translation	Lenght
Point Surface		2 Translations	-
U Joint		1 Rotation	-
Gear joint		1 Rotation	Angle1 or Angle2 (exclusive)
Rack Joint		1 Rotation or 1 Translation	Lenght1 or Angle2 (exclusive)
Cable Joint		1 Translation	Lenght1 or Lenght2
Screw Joint		1 Rotation or 1 Translation	Angle or Lenght (exclusive)
CV Joint		-	-

Figure 52 – Types of Joints found in the DMU Kinematics workbench [8]

5.4.1.4. The Commands

The commands are angular or linear values which define the kinematics motion. So as to define commands into the mechanism, it is important to think about which Part will drive the mechanism. For instance, the gear drives the pinion by the belt in the case of a bike.

In order to define a command in DMU Kinematics Simulation, a Joint has to be created before. Going inside the Joint properties and defining the option Angle Driven or Length the command is created. Even so, there are Joints that do not allow add commands.

5.4.2. Creating the Kinematic of the Mechanism

The previous step before starting to translate the kinematic of the real model to the 3D-CAD model is to study its functionalities. The real model has four functionalities as is shown in the picture below.

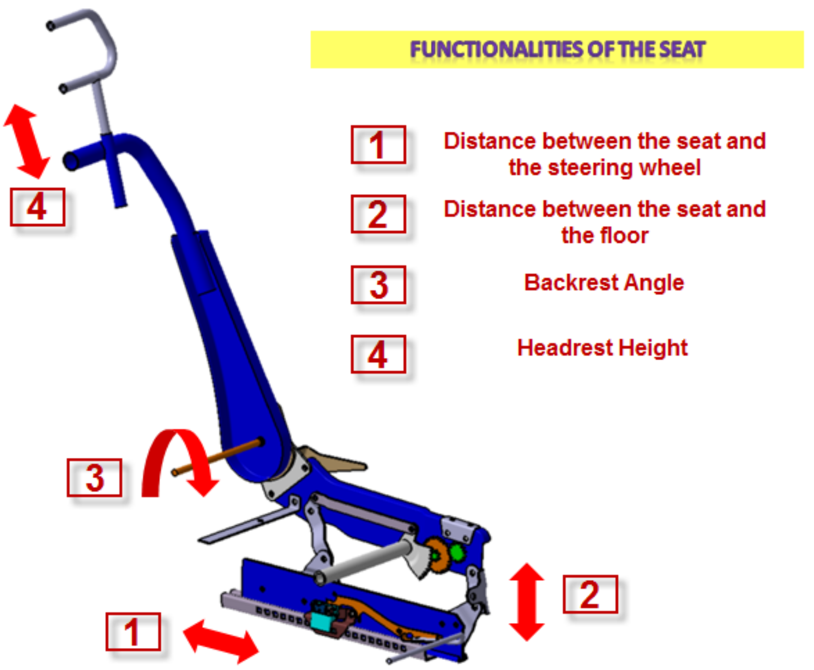


Figure 53 – Functionalities of the seat.

When the assembly is done, the first step to perform the kinematics simulation in the 3D-CAD model is to fix a Part using the Fix feature from the DMU Kinematics Toolbar. After clicking the Fix feature but before defining the Fixed Part, it will appear another pop up box which allows you to create a new mechanism. Once the mechanism is created it can be defined the fixed Part. In that case, the fixed Part will be the Guide Base.

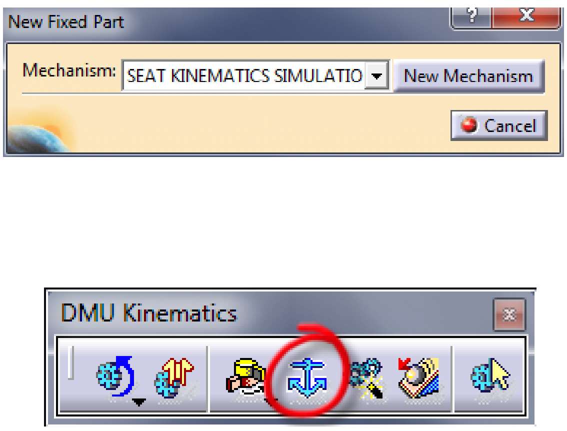


Figure 54 – New Mechanism window and DMU kinematics tool bar with Fix Part feature selected.

Then, Joints can start to be created. Opening the Kinematic Joints toolbars it is possible to select the desired joint. First, the forward and backward motion of the seat will be defined. For that purpose, a Prismatic Joint will be created between the Slider and the Guide Base.

Figure 55 – Kinematics Joints toolbar with Prismatic Joint selected

This motion must be defined with a certain range. For this reason, the Length Driven box is activated. This allows you to change the limits. According to CATIA-V5 terminology, specifying some Joints as Angle or Length driven joint is synonymous to defining a Command. This is observed by the creation of Command.1 line in the tree.

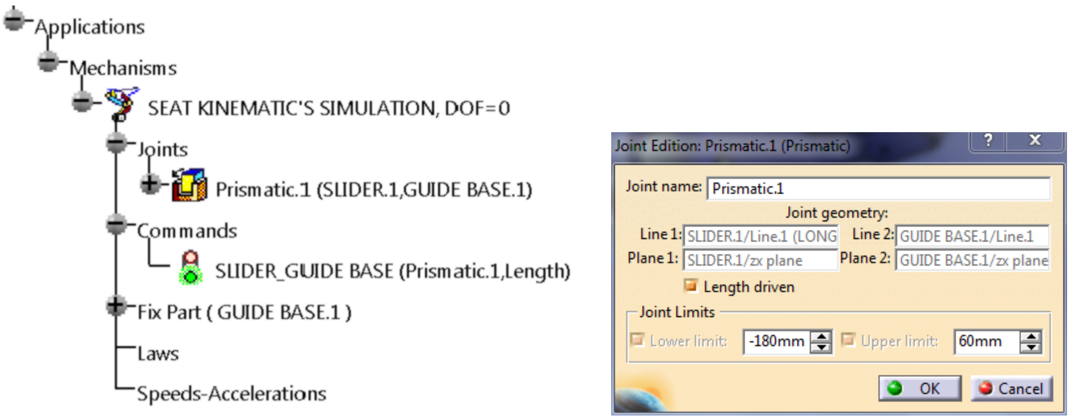


Figure 56 – Specification Tree and Prismatic Joint editor window

Next, the movement of the lever between the slider has to be defined. Turning the lever allows the foregoing movement and it has to be defined with a Revolute Joint. This motion must be defined with a certain range. At the same time, the movement between the Lock and the Lock Base it is created. Also, it is defined with a Revolute Joint. In order to combine the two rotational movements, a Point Surface Joint will be created between them. Using this joint, the rotational movement of the Lock between the Lock Base will be activated through the rotational movement of the Lever between the Slider. Now, a new command can be observed in the tree.

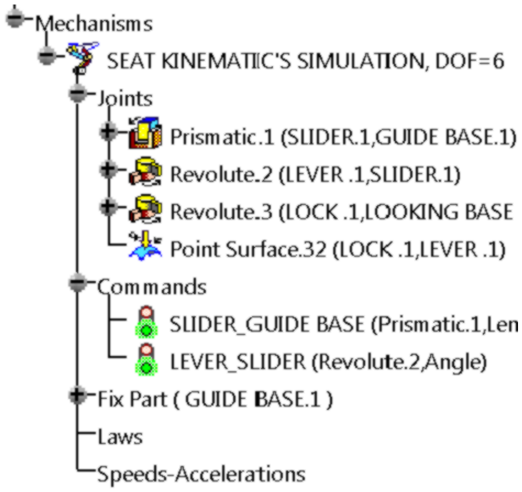


Figure 57 – Specification tree

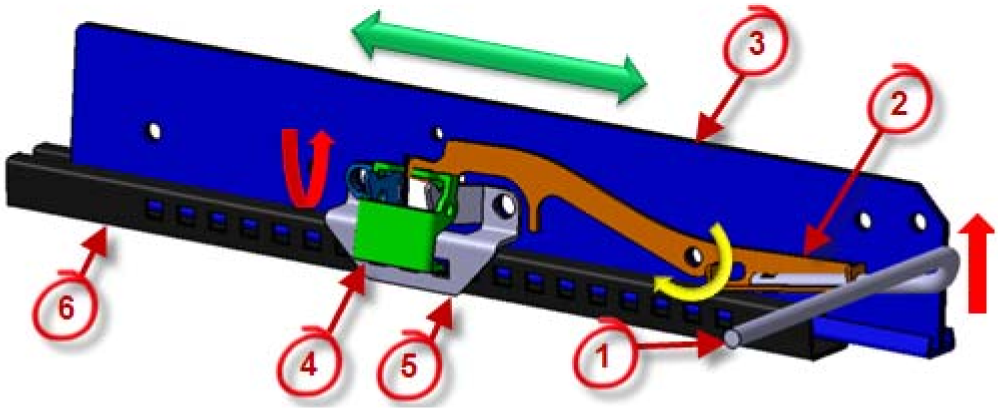


Figure 58 – Motions of the Longitudinal Translation Mechanism.
The involved Parts are: (1) Lever_2, (2) Lever_1, (3) Slider, (4) Lock, (5) Lock Base, (6) Guide Base

Afterwards, in order to define the related movement between Parts when the Height Regulator Mechanism is activated, several Revolute Joints must be created. This joint will be established to the Gear 1, the Gear 2 and the Gear 4 between the Seat Base. Also, it will be established to the Front Support 1 and 2 between each other and the Front Support 2 between the Seat Base. In the same way, it will be established to the Rear Support 1 and 2 between each other and the Rear Support 2 to the Seating Base. Finally, it will be carried out between the Height Mechanism Bar and the Gear 4 and the Rear Support 2.

The next step is to define the movement between gears. This is achieved creating Cylindrical Joints. Also, it is necessary to determine which Part will command the mechanism. For this reason, an angle driven will be added to the Gear 1. Then, the Cylindrical Joint will be established between the Gear 1 and the Gear 2 and between the Gear 2 and the Gear 4. The picture bellow shows how the tree is created.

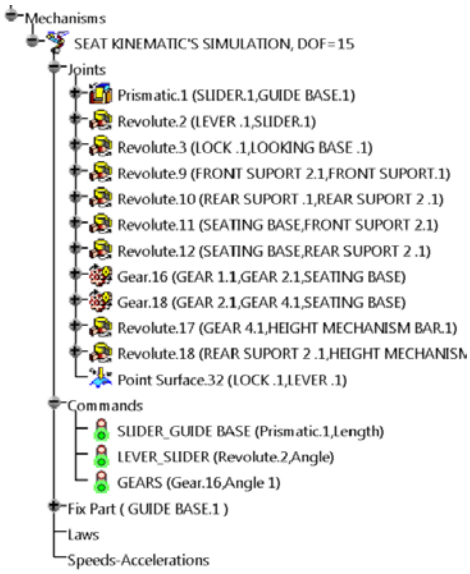


Figure 59 – Specification Tree

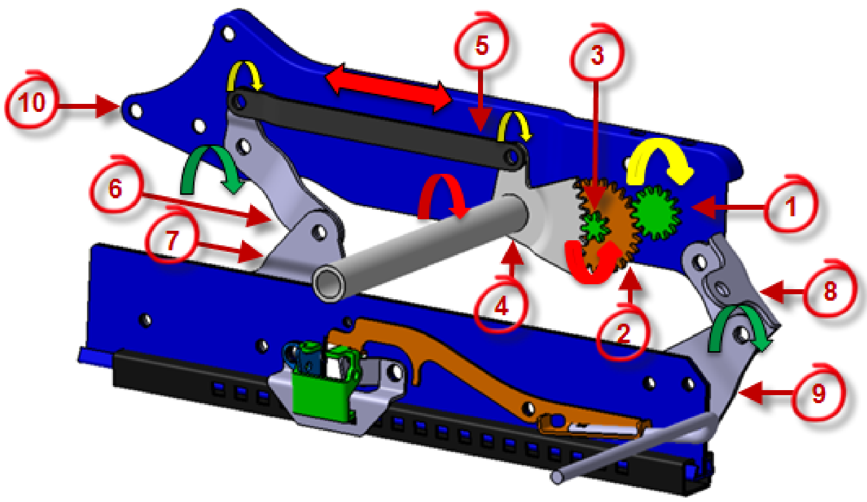


Figure 60 – Motions of the Height Regulator Mechanism.
The involved Parts are: (1) Gear_1, (2) Gear_2, (3) Gear_3, (4) Gear_4, (5) Height Mechanism Bar, (6) Rear Support_2, (7) Rear Support_1, (8) Front Support_2, (9) Front Support_1, (10) Seat Base

The motion of the Backrest Lever between the Backrest Mechanism Base is defined using a Revolute Joint. The movement of the Backrest Lever determinates the rest of Parts involved in this mechanism. Accordingly, an angle driven is defined. To describe the Regulator movement between the Backrest Mechanism Base, which is made by the Backrest Lever, a Prismatic Joint is created. Afterwards, a Point Curve Joint will be created in order to combine these two movements. Thus, a new command is added in the tree.

Once finished the precedent command, the last two movements between Parts of the model are created quickly. These movements are the variation of the Backrest Angle and the Headrest Height Regulator. In order to define the Backrest Angle, a Revolute Joint between the Backrest Angle Mechanism Base 1 and 2 are created whereas the movement between the Headrest and the Backrest are carried out using the Prismatic Joint.

Finally, to reduce the Degrees of Freedom of the mechanism several Rigid Joints are added to different Parts of the model. As a result, the expansion tree should be looks like the picture bellow.

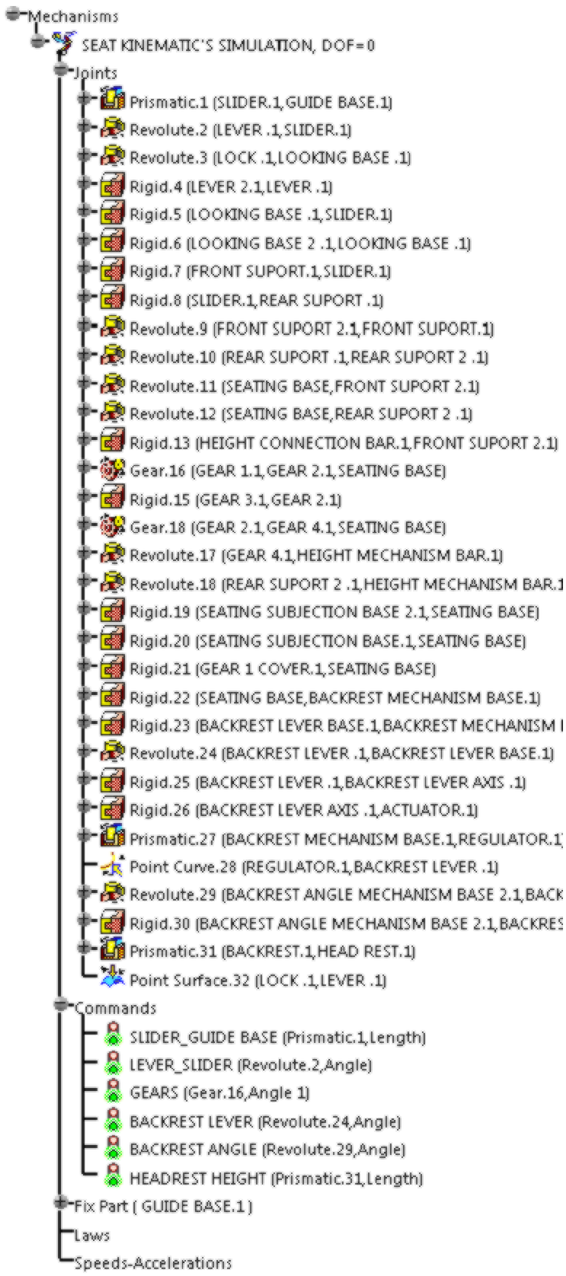


Figure 61 – Final Specification tree

5.4.2.1. Simulating the Mechanisms

When the DOF are zero the mechanism can be simulated. However, CATIA offers many options to carry out this aim. In that case, the mechanism is going to be simulated with commands and afterwards a record of the simulation is done.

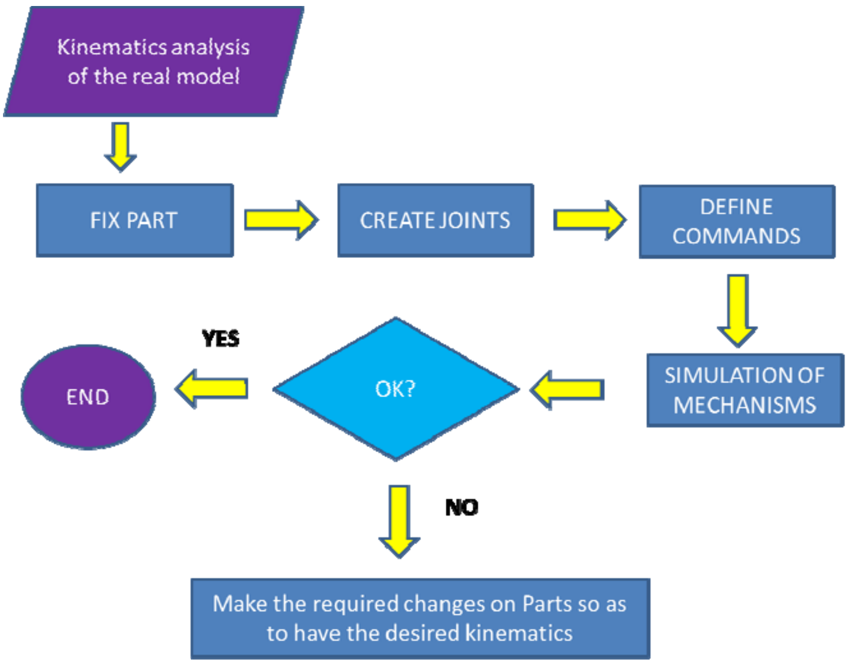


Figure 62 – Followed Steps in the Kinematics design

5.5. The Parameterization with CATIA

Design changes are commonly found in the product development process. When a product is being developed, a geometric change in one of its components can entail modifications in the rest of the product components. Consequently, affects the entire product assembly. Nowadays, new CAD software packages offer many possibilities to solve this problem as the parametric-associative design.

Using the Formula feature of CATIA, it is possible to create parameters and relations between components in order to define the geometry of a Part or a Product carrying out a new cycle of design.

5.5.1. Creating the Parameterized Model

There are many strategies to define parameterized models. On the one hand, it is possible to create parameters within a component related to an internal dimensional constraint. Then, this dimension will change simply modifying the parameter value inside the Part. If the parameterization has to be carried out in single components maybe it is the best strategy. On the other hand, it does not very useful when the parameterization has to be carried out on an assembly of several components or sub-assemblies. Then, creating parameters on the assembly and making relations between them and constraints, positions and parameters inside Parts, the geometry of the entire model can be controlled. One accurate way is to create several reference points, lines or planes in order to define the structure and the most important points of the model in a new Part inside the assembly commonly called Skeleton. These reference elements will control the position of the Parts in the space or geometrical constraints of a single or multiple Parts. For that purpose, the position of these elements inside the Skeleton is controlled creating parameters related to them in the assembly and afterwards, relations between them and parameters, geometrical constraints or reference points inside Parts have to be carried out. Using this method, the control of all geometrical characteristics of the Parts can be executed only changing the position of the Skeleton reference elements. In that case, due to the model is composed for several Parts it was the followed strategy.

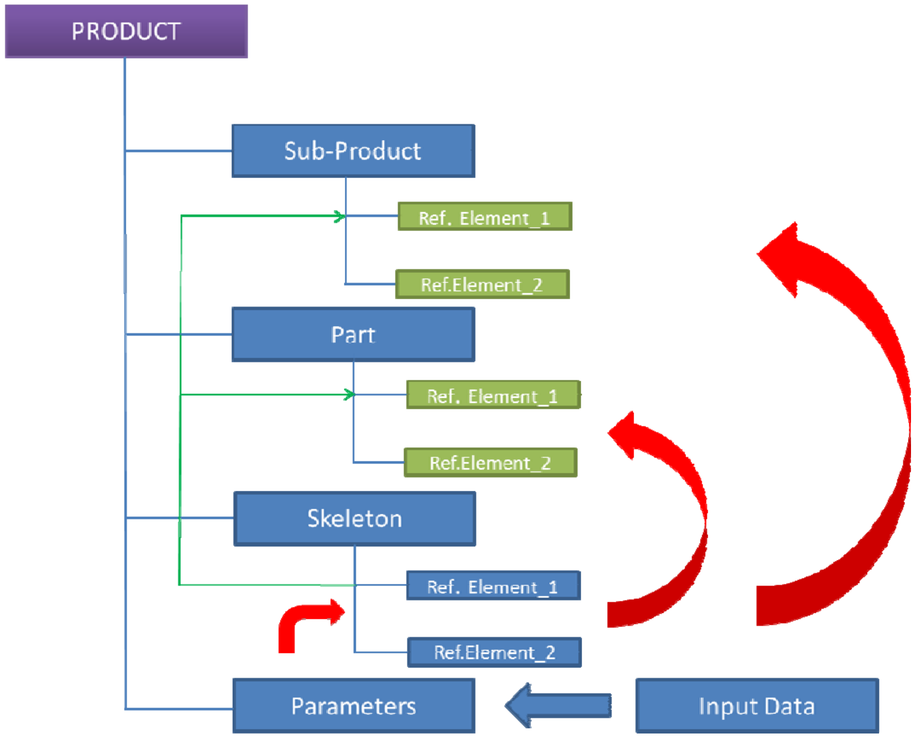


Figure 63 – Parameterization Strategy

In this thesis, in order to have a model as versatile as possible, three parameters will define the geometry of the entire model: the Width, the Length, and the Height whereas four parameters will define geometry of single Parts on the assembly level: the Backrest Height, the Backrest Angle, the Headrest Height and the Headrest Width. All of them have been created related to the body of the seat. Modifying these parameters it is possible to adjust the seat model to the desired space or geometry.

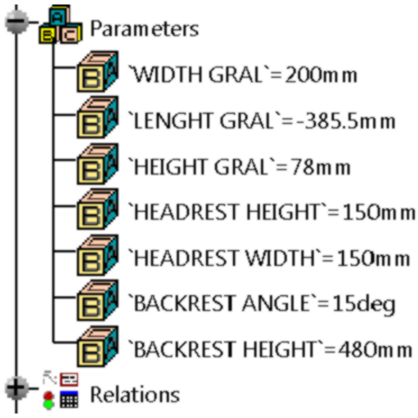


Figure 64 – Parameters created in the assembly

The next points have been created so as to explain the points created in the Skeleton Part and all the characteristics related to the parameters.

5.5.1.1. The Skeleton

The Skeleton is composed of seven points. The Parameterized_1 point is created to define the reference point for all of them. It is located in the default origin point of CATIA, it means it is on the (X=0, Y=0, Z=0) of the coordinate system. The Parameterized_2 point Y coordinate value is controlled by the Width parameter. Then, will define the distance in the Y axis between the Parameterized_1 point and itself. The Parameterized_3 point X and Y coordinate values are controlled by the Length and the Width parameter respectively. Consequently, it defines the distance between the Parameterized_1 and the Parameterized_3 points in the X and Y axis. In the same way the Parameterized_4 point Y and Z coordinate values are controlled by the Width and the Height parameter respectively. Therefore, it defines the distance between the Parameterized_1 and the Parameterized_4 points in the Y and Z axis. Likewise, the Parameterized_5 point X, Y and Z is controlled by the Length, the Width and the Height parameters respectively. In other words, the distance between the Parameterized_1 and the Parameterized_5 points is defined in the three axis of the coordinate system. Basically, these points have been created in order to define the position of the reference point of each component of the model although in some cases have reference on some geometrical constraints.

While the previously mentioned points position are related directly with the Parameterized_1 point, the Parameterized_6 point is related with the Parameterized_5 point and the last one with the Parameterized_7 point. The Parameterized_7 point defines the top of the Backrest and the Parameterized_6 point the vertical distance between the Parameterized_7 point and the reference point of the Backrest Part. The reason is that these two points have a specific function inside the model. The Parameterized_6 and the Parameterized_7 points are the responsible to define several characteristics of the Backrest, the Backrest Cover, the Headrest and the Headrest Cover.

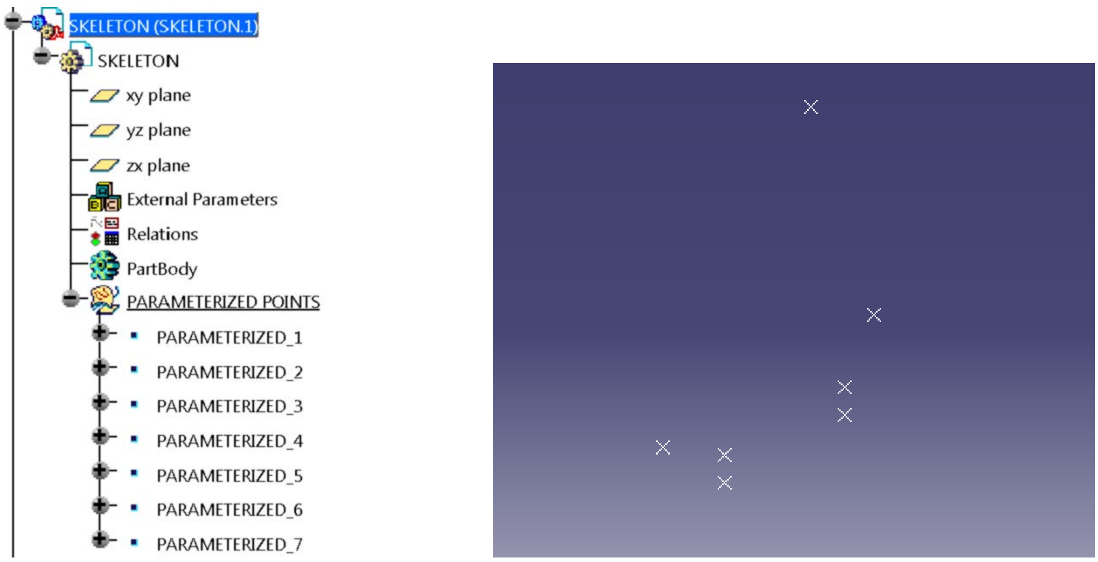


Figure 65 – Points of the Skeleton in the Specification Tree and in the space

5.5.1.2. The Width parameter

This parameter is related to the distance between the center line of the seat and the center line of the Guide Base according to the Y axis although at the same time, it has effect on the entire model. In other words, modifying the Width parameter is possible to adjust the seat to the desired width. As a result, the Parts with elements located on the Y axis will be modified according to the input value. It has been created with a specific range but it can be modified in order to satisfy the needs of the designer. This range comprises the values between 125 and 250 mm each included. By contrast, the seat development process can be focused first on the measurements of the cushion. In this model, the distance between the center line of the Guide Base and the exterior surface of the cushion is 36 mm. For instance, if the desired width of the cushion must be 500 mm then, the input value of this parameter will be 214 mm.

Figure 66 – Width Parameter on the assembly level

5.5.1.3. The Length parameter

This parameter defines the length of the Guide Base according to the X axis. When the input value of this parameter is changed some Parts located on the X axis will suffer changes on its geometry. These Parts are the Guide Base, the Seat Base and the Height Mechanism Bar which increases or reduces its length. Also, the Length Parameter has been designed with a specific range comprised from 385 to 450 mm each included. In the same way, as the Width parameter the seat development process can be focused on other measurements in order to define the length. As reference, the difference between the reference measure of this model and the rotation axis of the Backrest Part is to rounding off 31 mm.

Figure 67 – Lenght Parameter on the assembly level

5.5.1.4. The Height parameter

This parameter controls the height between the floor of the car and the Seat Base according to the height of the Slider. In fact, it has only geometrical relation in the height of the Slider but all the remaining Parts of the model will be modified its position in the space. In the same way, it has been defined with a certain range from 78 to 150 mm each included. If the needs of the designer are to know the distance between the Seat base and the floor the difference between the input value and the top of the Seat Base is always 114 mm.

Figure 68 – Height Parameter on the assembly level

5.5.1.5. The Backrest Height parameter

This parameter controls the height of the Backrest and the Backrest Cover Part according to the vertical distance between the center axis of the pipe which defines the invert U-shaped of the Backrest and the rotation axis. Still, when the input value is changed it has no relation with geometries of other Parts into the model. Only, the position of the Headrest will change according to changes. Also, it has been designed with a specific range from 480 to 620 mm each included.

Figure 69 – Backrest Height Parameter on the assembly level

5.5.1.6. The Backrest Angle parameter

This parameter controls the arm angle of the inverted U-shaped of the Backrest. This angle is referenced to the Z axis of a coordinate system located on the rotation axis of the Backrest. When the input value of the parameter is changed, it has only impact on the space position of the Headrest. The range designed to its parameter is from 10 to 20 degrees.

Figure 70 – Backrest Angle Parameter on the assembly level

5.5.1.7. The Headrest Height parameter

The Headrest Height parameter controls the distance between the axis of the lower cylinder that forms the body of the Headrest and the top of the cushion. This parameter it has no impact in the position of any Part of the seat model, only in the geometry of the Headrest Cover geometry and the Headrest. The range of this parameter is comprised from 120 to 220 mm.

Figure 71 – Headrest Height Parameter on the assembly level

5.5.1.8. The Headrest Width parameter

The Headrest Width parameter controls the distance between the center line of the seat and the side of the inferior side of the Headrest Cover. This parameter has no impact on the position or the geometry of any Part of the seat model, only on the geometry of the Headrest Cover and the Headrest. The range of this parameter is comprised from 140 to 220 mm.

Figure 72 – Headrest Width Parameter on the assembly level

6. Conclusions

This Thesis includes the creation of a 3D-CAD model based on an existing real car seat. Detected a certain problem in the development of this type of products, this work has been focused specially in a detailed study of the kinematics of the seat functionalities. At the same time, it was important to be specific about the parameters which define the geometry of the single components or the entire model of the seat. The design of the seat kinematics and the parameters should be useful tool to implement the model in future development processes. So as to achieve the objectives of this Thesis, it was chosen to create the model with CATIA-V5 R16 version and the Generative Shape Design workbench and try to simplify the complex shapes found in the seat as a method. Studying all the components of the seat and their functionalities individually and regarding the influence on the entire model, the kinematics of the seat has been carried out. That involves the creation of several mechanisms on the 3D-CAD model and the possibility of changing the values that are defining them. Then the model can be useful to analyze opportunities to improve these kinematics mechanisms or the geometry of components using the model as a base to the study. On the other hand, due to the different application fields in which the model can be implicated, the external control of the geometry of the entire model or some components must be carried out. For this reason, parameters or relations in a single part or regarding to the entire model were implemented on the assembly level. The strategy followed was the creation of a new Part in which several elements were defined and related to the already existing reference points of the components previously created. Also, the parameters are related to the elements of the mentioned new part. It allows the designer to put new input values and consequently to change components geometries, positions, etc.

6.1. Further overview of possible studies

The scope of this thesis was to obtain a reliable and useful tool to improve the development process of automotive seat, but the main problem found was that there was not enough available information. But as a first and very important step a good kinematic model was created. Also it is possible to vary the motions making little changes in the geometry of some components.

Based on this thesis further studies could be made. The first step that could be performed is the definition of the construct parameters of the seat under laws in order to give a clear idea of which measurements are controlling changing the input values of the parameters. This point could be extended widely inserting standards procedures to the first steps of the seat design.

In the same way, a lot of improved or different mechanisms can be added to the seat model and study their results and the differences between them.

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