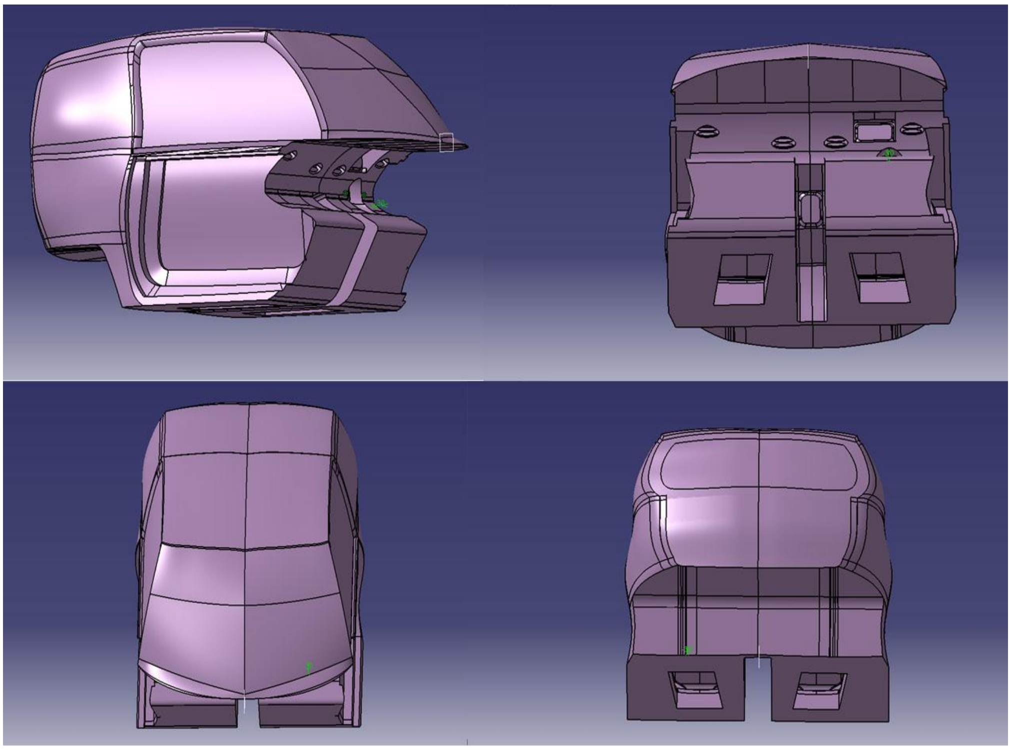
Technical University of Crete
School of Production Engineering and Management

Diploma Thesis



Three-dimensional modeling of the interior of an urban car

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Acknowledgements

I would like to express my thanks to the associate professor and supervisor of the current thesis Ioanni K. Nikolo for both the technical knowledge he provided me and for his guidance during the whole period of the project's completion.

Especially, I would like to express my gratitude to my entire family, who supported me in every stage of my studies in every possible way.

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1. ABSTRACT

The primary scope of this thesis was the detailed construction of a 3-D CAD model of an urban car, with characteristics and dimensions close to that of the currently designed vehicles of that type. Both the vehicle’s exterior shell solid geometry and its internal cabin were defined. These two entities were designed separately and were then integrated together to form the vehicle’s final digital mock up. The purpose of the interior definition was to allow for the simulation of the air-conditioning air flow by the means of Computational Fluid Dynamics (CFD) software. This simulation is not part of the current thesis. For the construction of the vehicle’s CAD model the CATIA V5R19 software was used, being one of the most advanced CAD software available in the market. In the following chapters the detailed procedure used to design all the parts of the vehicle is described in detail, using a number of pictures.

2. DEFINITION OF VEHICLE’S CABIN PARTS

2.1 DESIGN OF THE BACKREST PART OF THE SEAT

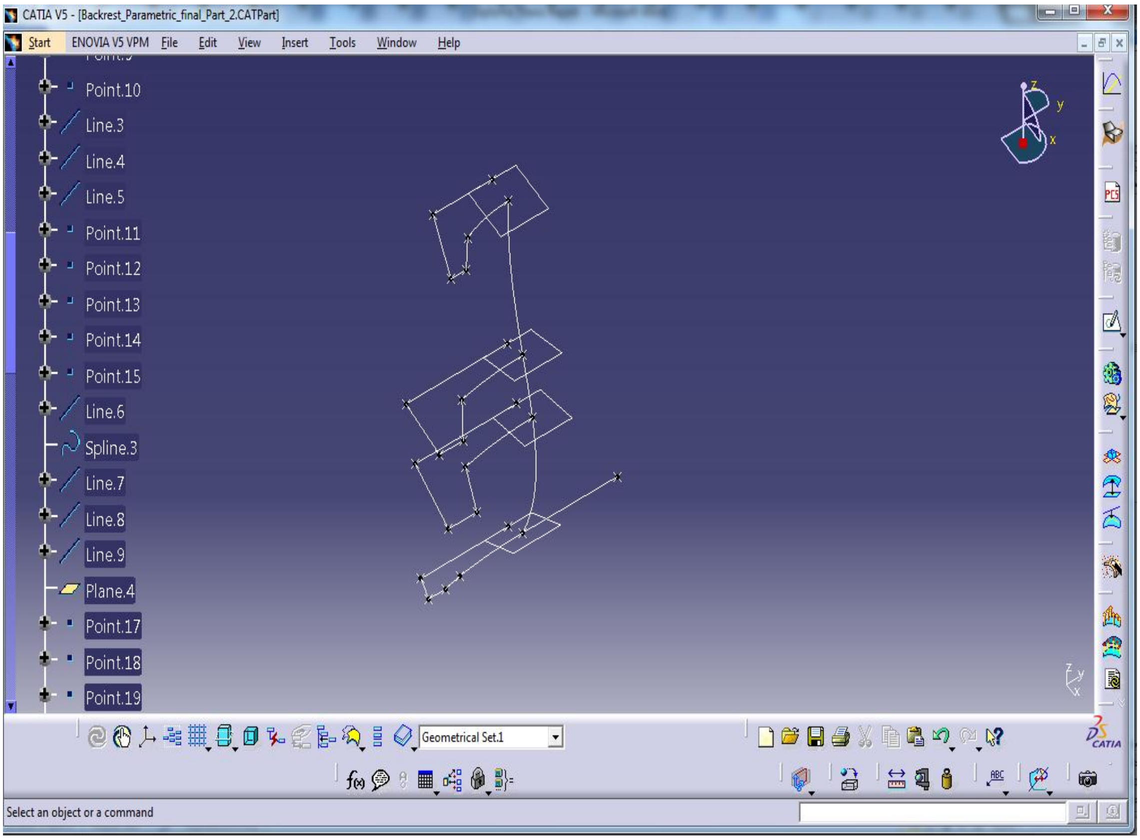
The design of the backrest part took into consideration both comfort and ergonomic criteria. Moreover, for reasons of functionality and efficiency of the CAD design, the 3-D geometry was constructed as a parametric solid volume. The final shape of the solid part was created after taking into account the design of a real passenger seat that has already been used in a real urban car design.

The first step for the definition of the 3-D geometry was the insertion of four planes that represent the number of defining sections inside the part and the design of the wireframe model on these reference planes. The number of planes was determined as four in order to produce a solid geometry that resembles the factual initial geometry as closely as possible. Moreover, to add flexibility to the final backrest part, an additional B-spline curve was introduced, which connects the four section planes acting as the spine of the solid geometry.

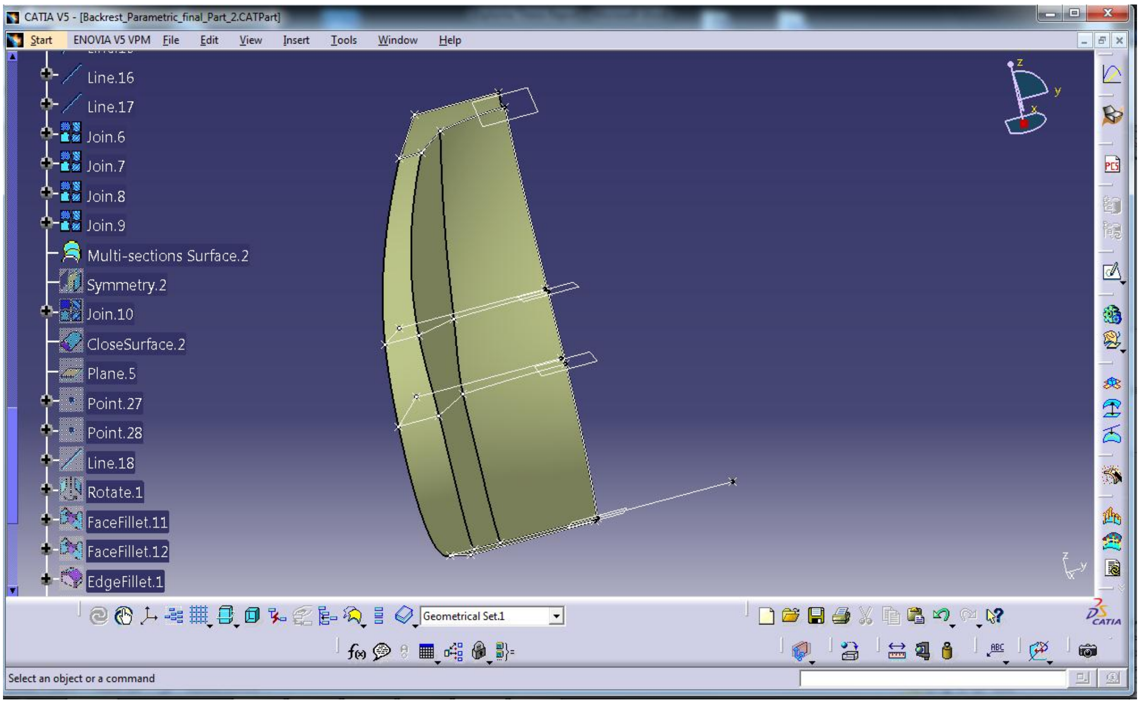
For reasons that are imposed by the final assembly of the backrest part with the underest part of the vehicle seat, the base plane of the wireframe geometry should be defined with a certain degree of inclination. As it became clear from the wireframe model, only half of the part needs to be defined, due to the existing symmetry with respect to xz-plane (as defined in Catia software, picture 2.1.1).

The next step of the design procedure was the creation of the resulting surfaces, based on the wireframe model. In order to complete this task the section curves of the model were unified with the use of “*Join*” command before the execution of the surface-creation commands. By using the “*Multi-Sections Surfaces*” command, the first half of the main body surface of the model was created (Pictures 2.1.2 to 2.1.5). Then, the creation of the symmetrical part was achieved with the use of “*Symmetry”* command, which is available in Generative Shape Design Platform of Catia software (Picture 2.1.6).

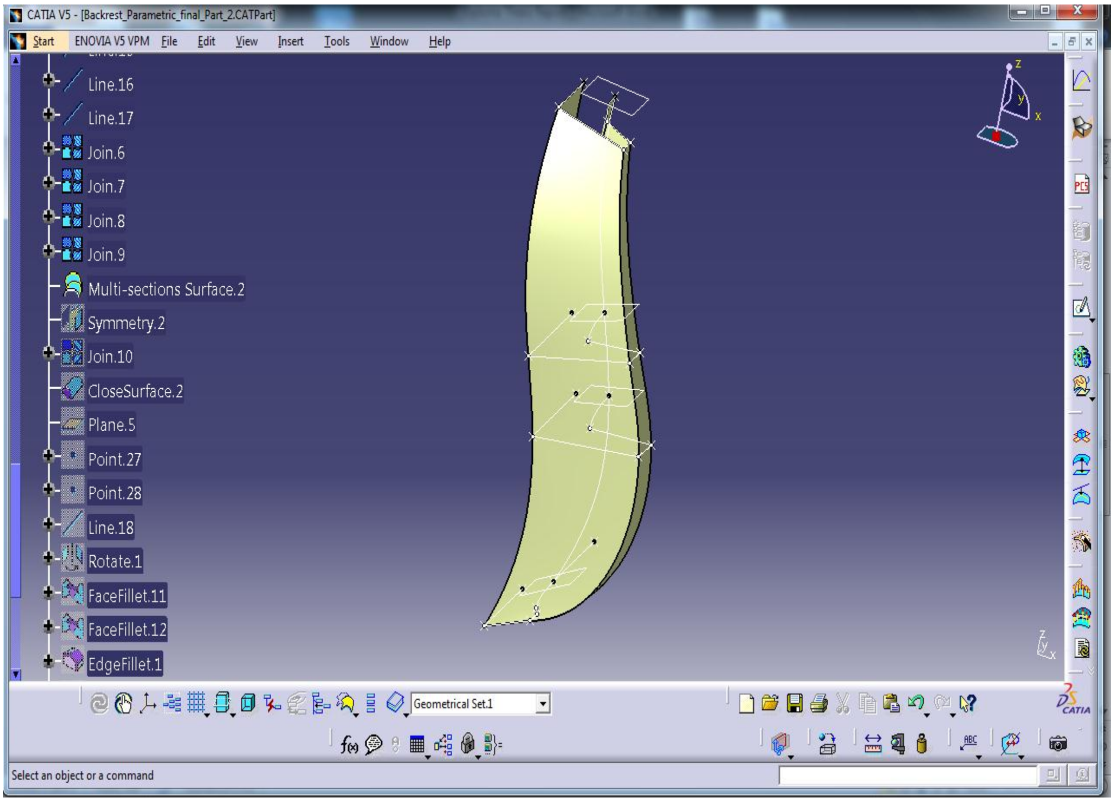
The final step towards creating the solid volume is to join together these two surfaces for the purpose of creating of a new unit and then use the “*Close Surface”* command, in order to fill the surface with material. The result of the procedure is shown in Picture 2.1.7. For aesthetic purposes, a number of "rounding operations" was applied to the sharp edges of the solid. This procedure was conducted by the “*Edge Fillet”* command. The final form of the solid part is depicted in Picture 2.1.8.



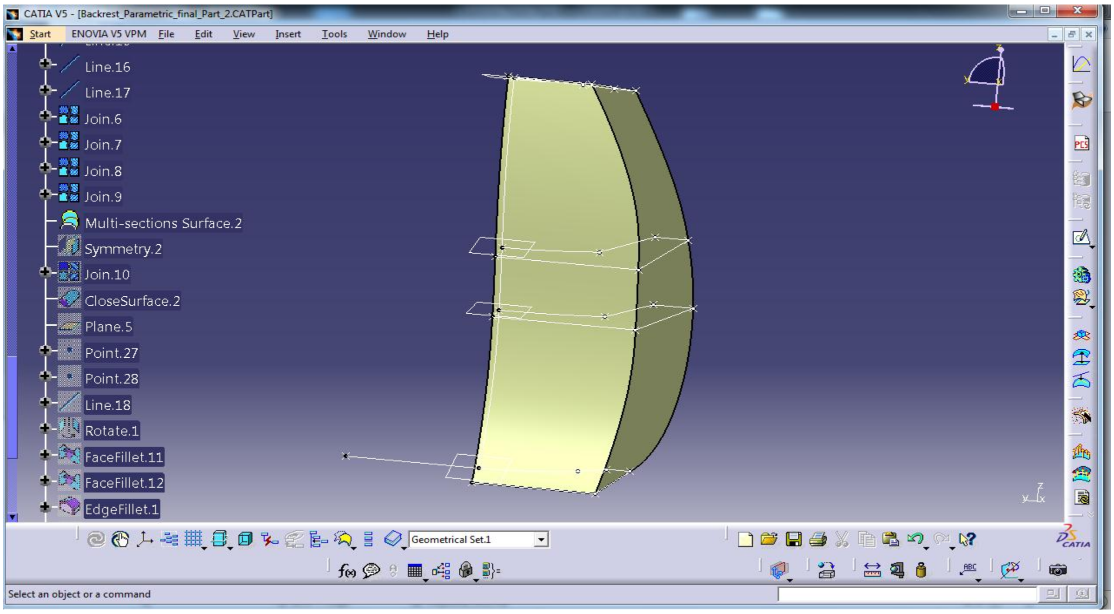
Picture 2.1.1: Design of backrest’s wireframe model.



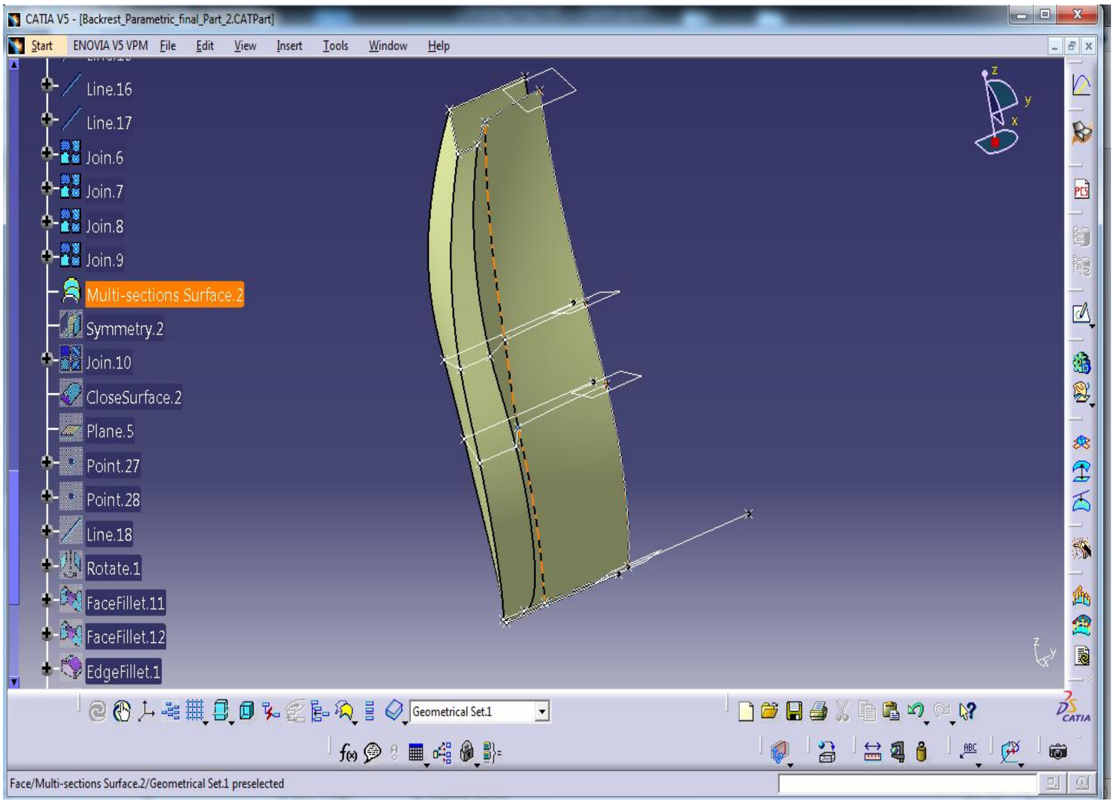
Picture 2.1.2: Surface model front view.



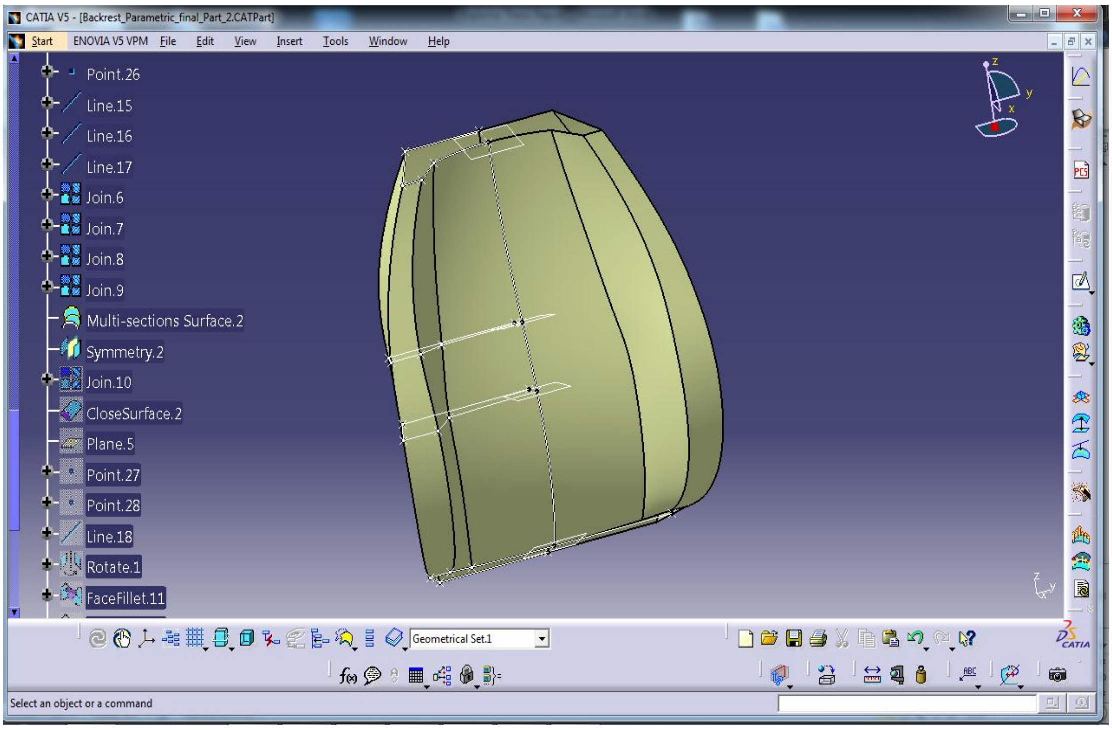
Picture 2.1.3: Surface model side view.



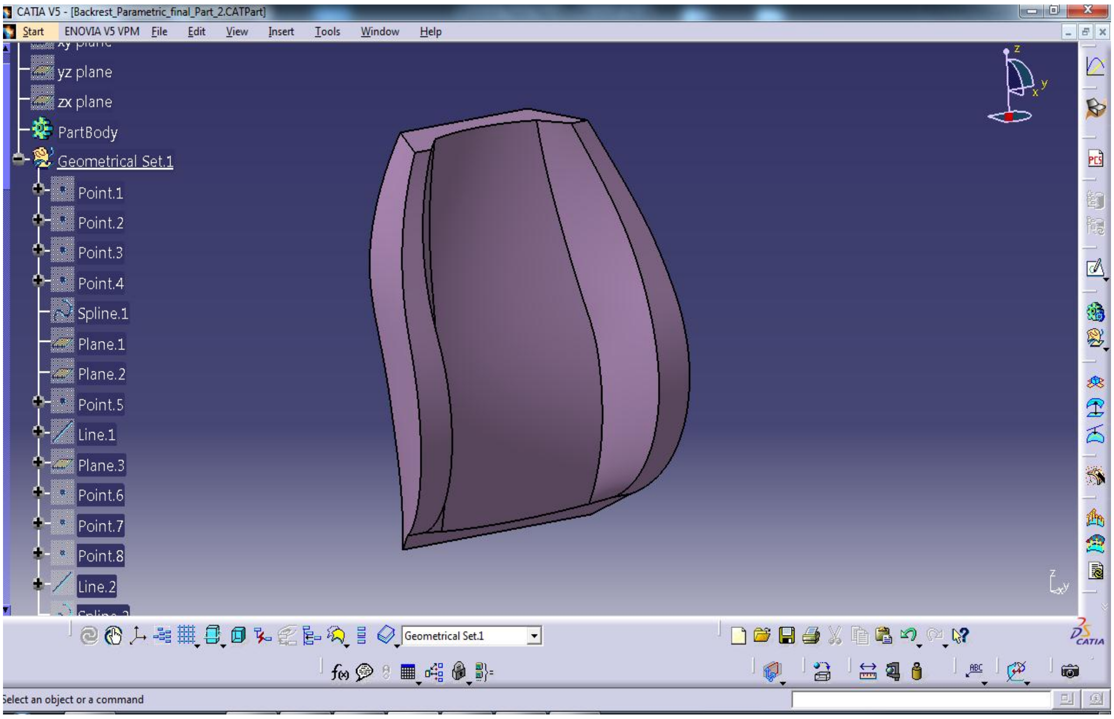
Picture 2.1.4: Surface model back view.



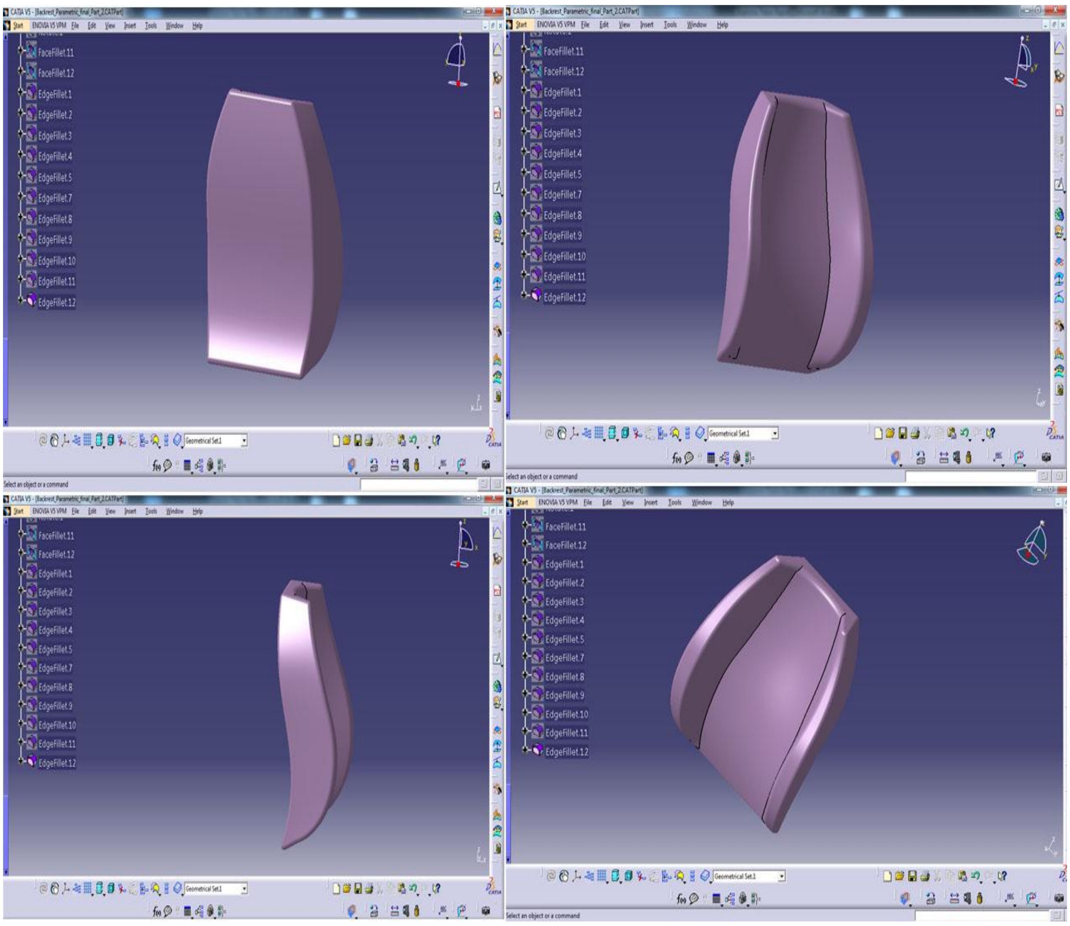
Picture 2.1.5: Surface model – first symmetrical half.



Picture 2.1.6: Surface model with second symmetrical half.



Picture 2.1.7: Backrest's solid volume – digital mock up.

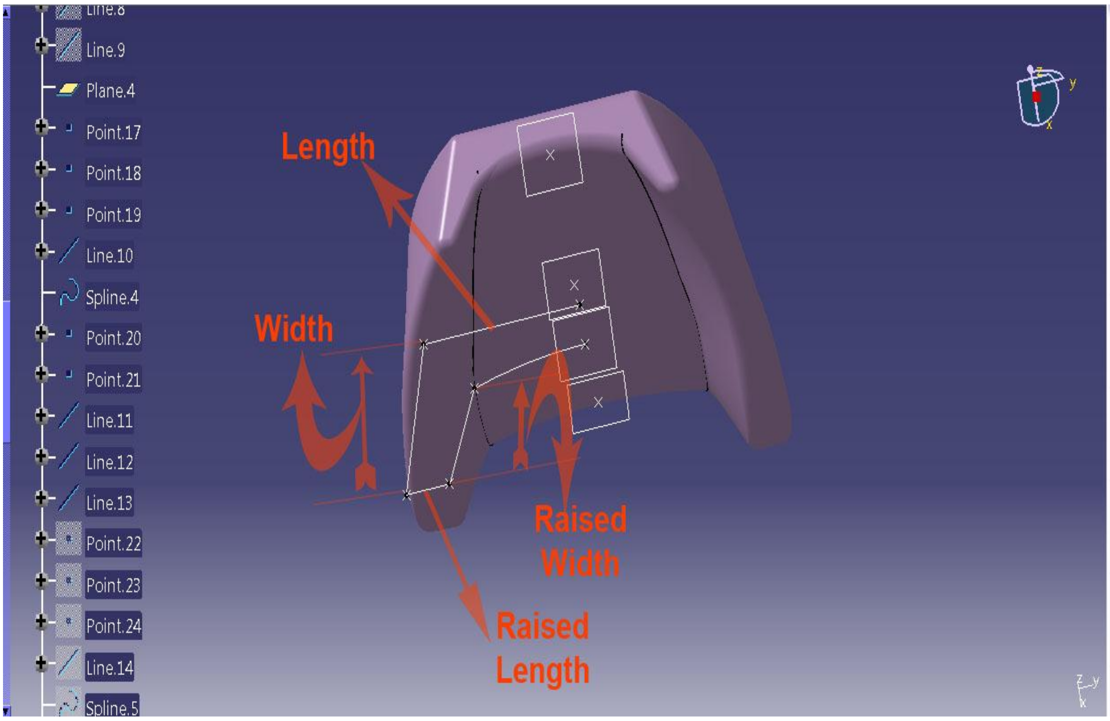


Picture 2.1.8: Backrest’s final solid volume.

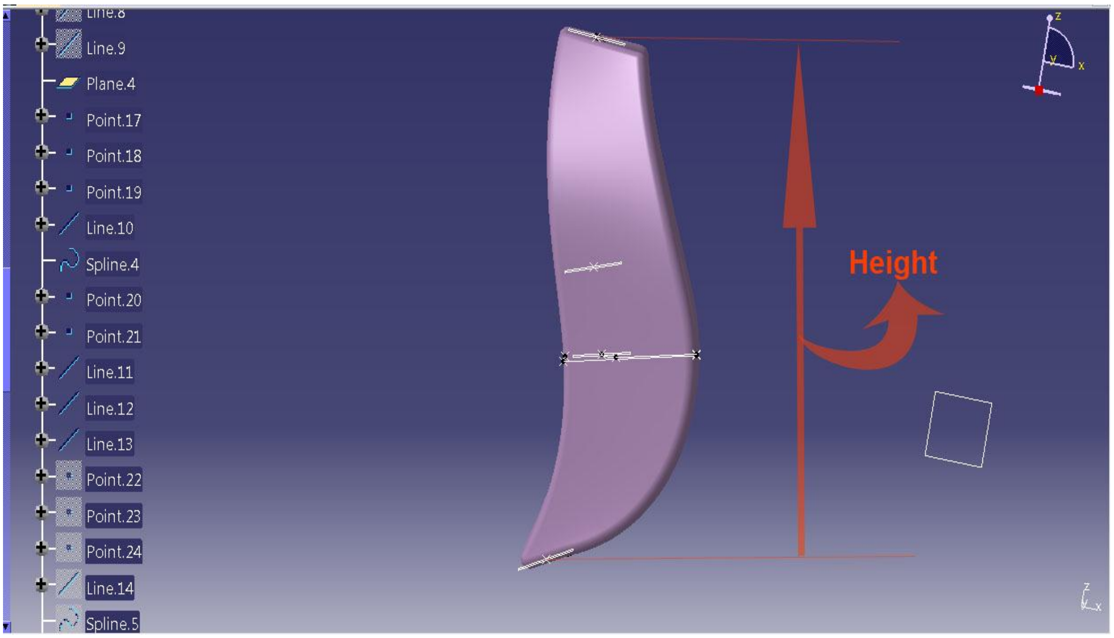
The dimensions of the solid part were measured on an already existing vehicle seat. The dimensions’ primary components were the length, width and height of the part. Moreover, some additional dimensions were determined. These were the height and the width of the raised sides of the seat. The precise numerical values of those dimensions are depicted in the following table (Table 2.1.1). The definition of each dimension on each section is shown in pictures below (Picture 2.1.9).

Table 2.1.1

Dimensions (mm) /Sections	Section 1	Section 2	Section 3	Section 4
Length	125	215	215	185
Distance From Previous Section	-	270	100	253
Width	120	170	180	85
Raised Length	32,5	50	60	35
Raised Width	50	100	110	15
Total Height	623	-	-	-



Picture 2.1.9: Dimensions' definition.



Picture 2.1.10: Height definition.

2.2 DESIGN OF THE UNDEREST PART OF THE SEAT

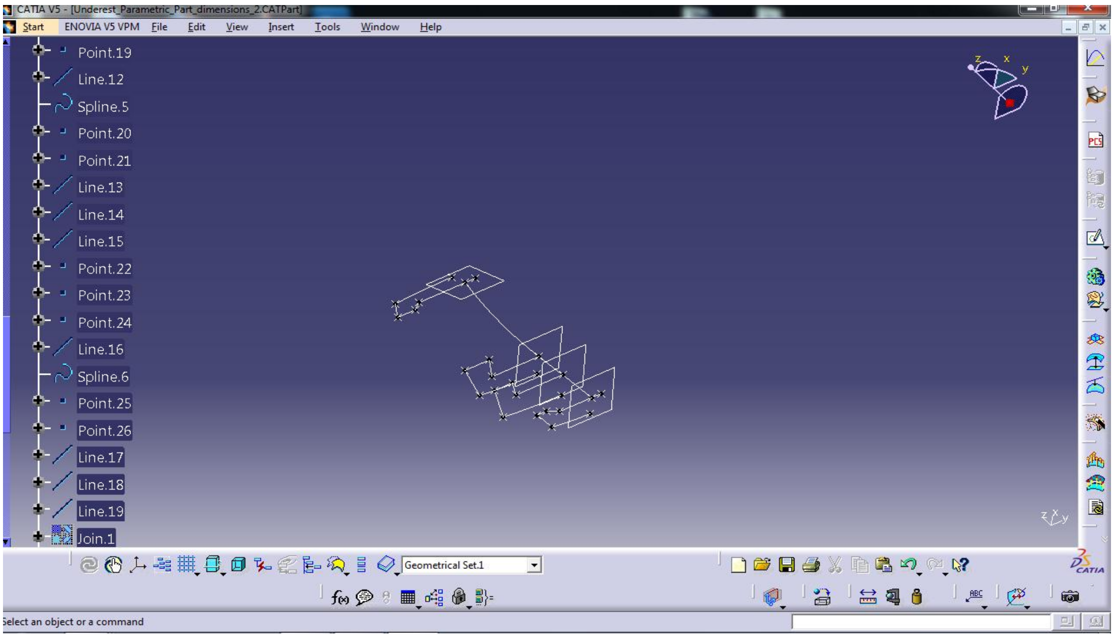
Similar to the previous design, this part was designed as a unit of an assembly of three entities, in order to get the final integrated part of the seat. This model is also a parametric solid, with similar design criteria as with the other parts of the thesis. Following the same methodology as before, the wireframe model was designed first by defining the points necessary for the representation of the geometry with lines and B-Spline curves (Picture 2.2.1).

From the representation of the part's wireframe model the same ''spine technique'' was used as a method for a comfortable and flexible way of shape transformation. Same as with the previous part, this part presents a symmetry with respect to xz-plane, thus only half part of the section needed to be defined. The next step of the design process was to unify the sections of the wireframe model with the "*Join*" command, in order to construct the surface geometry of the body. Then, by using the *"Multi Sections Surface"* command, the surface of the body was created (Picture 2.2.2). Next, the mirrored surface was created using the "*Symmetry*" command, defining the xz – plane as a reference plane for the mirroring process (Picture 2.2.3). For the final step of the creation of the solid volume, an additional *"Joining"* operation should be performed for the unification of the two surfaces. The solid 3-D geometry was created with the "*Close Surface*" command (Picture 2.2.4). For further aesthetic purposes some rounding operations were applied on the sharp edges of the resulting solid (Picture 2.2.5).

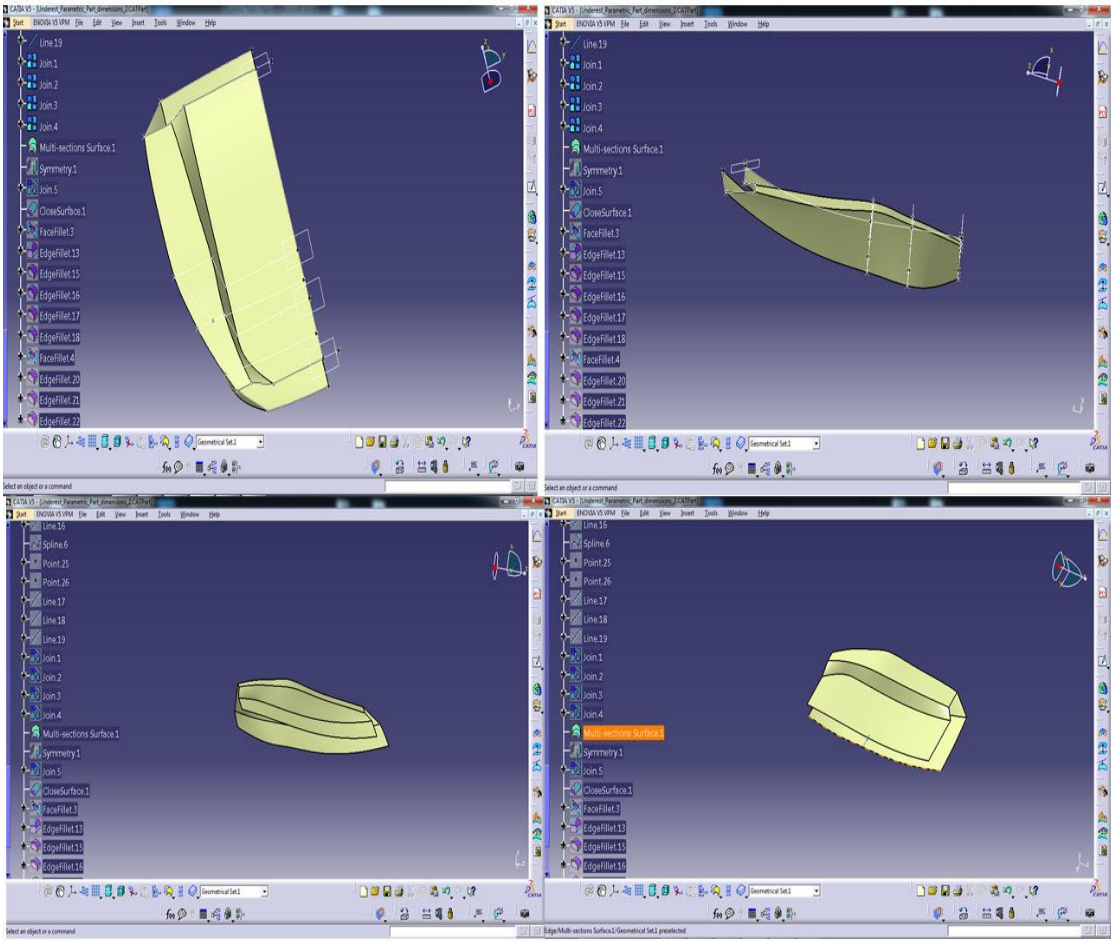
The final stage for the completion of the design was the dimension measurement of each section. The major dimension components were the total length, the height and the width of each section. Moreover, some secondary dimensions, such as the height and width of the raised ends (which form the "bucket" style) of the part were determined. The exact value of each dimension derived from measurement of such values from a real vehicle seat. The precise numerical values for every section are shown in Table 2.2.1 below. The definition of each dimension is shown in Pictures 2.2.6 and 2.2.7. The radius of the rounding is 5 mm for each sharp edge of the geometry (Picture 2.2.8).

Table 2.2.1

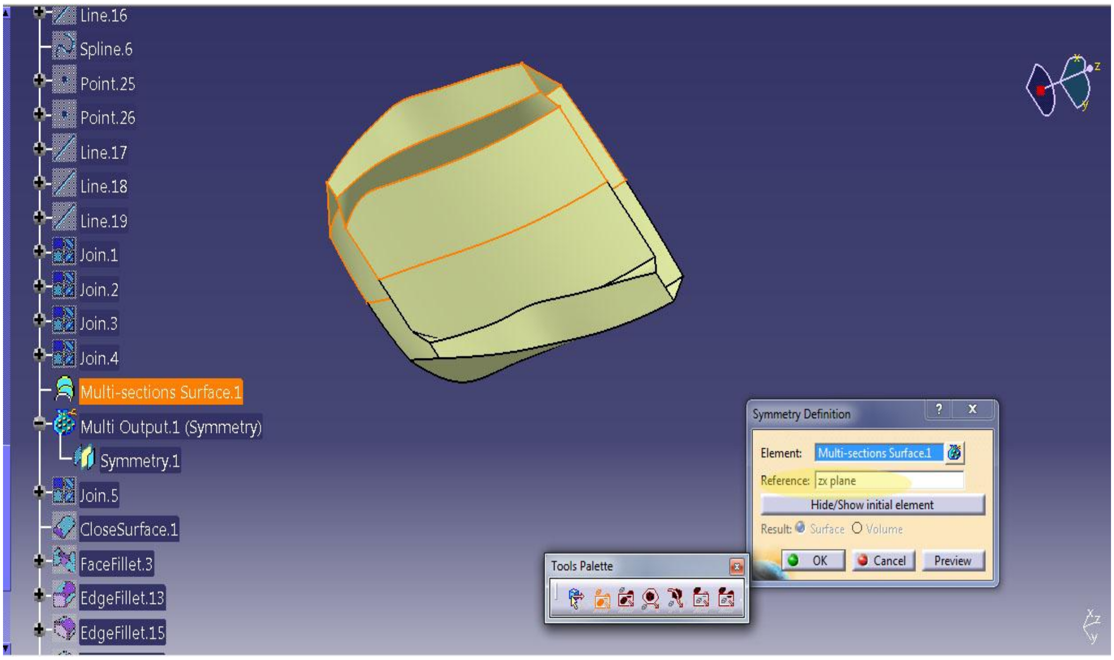
Dimensions (mm) \Sections	Section 1	Section 2	Section 3	Section 4
Height	185	180	180	185
Width	65	100	105	65
Raised Height	25	45	60	25
Raised Width	55	55	75	55
Distance from the Previous Section	-	110	90	290
Total Length	490	-	-	-



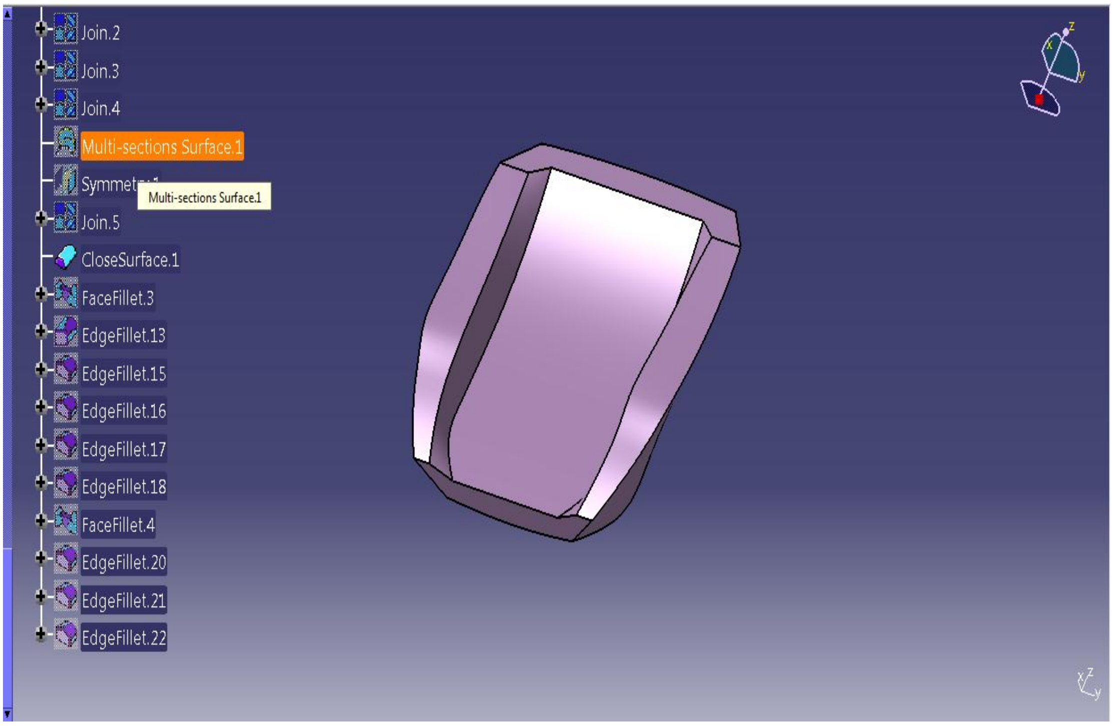
Picture 2.2.1: Underest wireframe model.



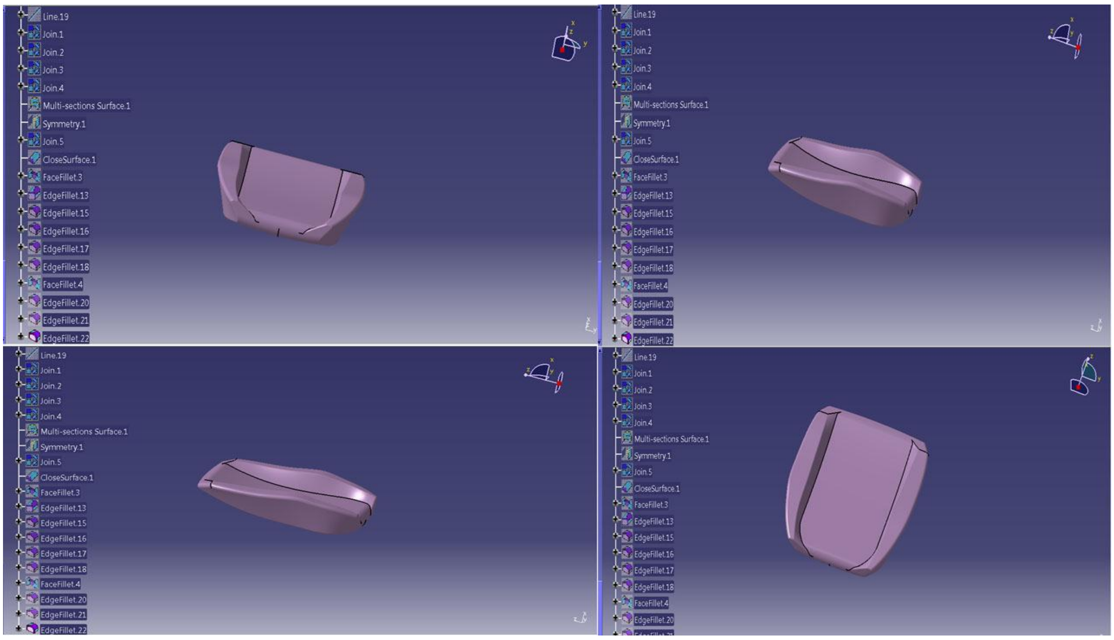
Picture 2.2.2: Surfaces of the underest part.



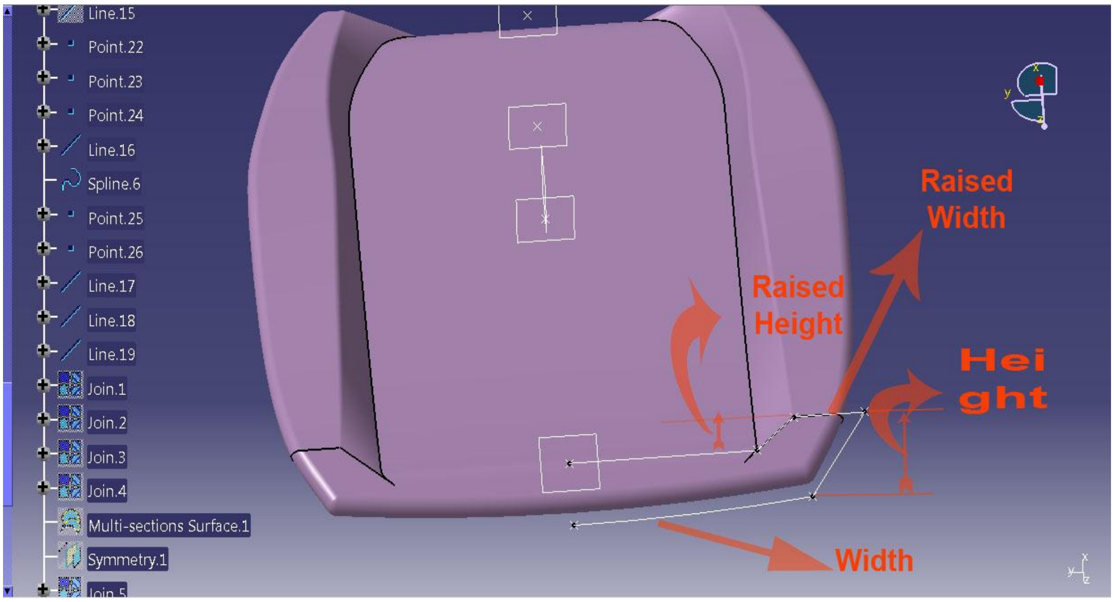
Picture 2.2.3: Construction of the symmetry surface.



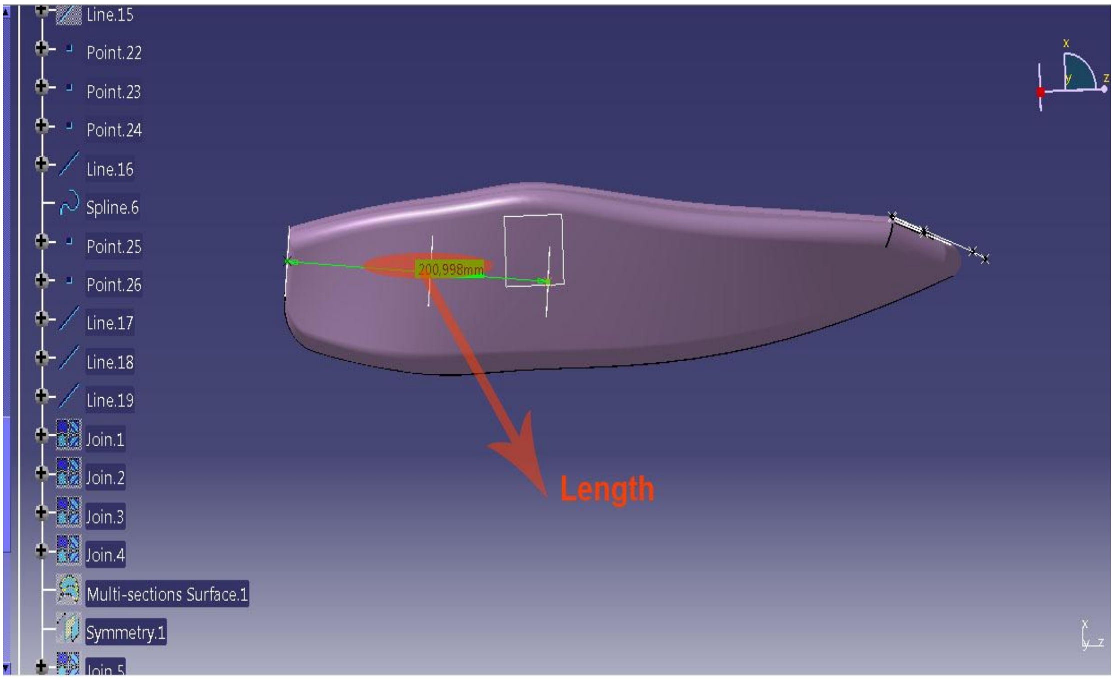
Picture 2.2.4: Construction of the underest solid volume.



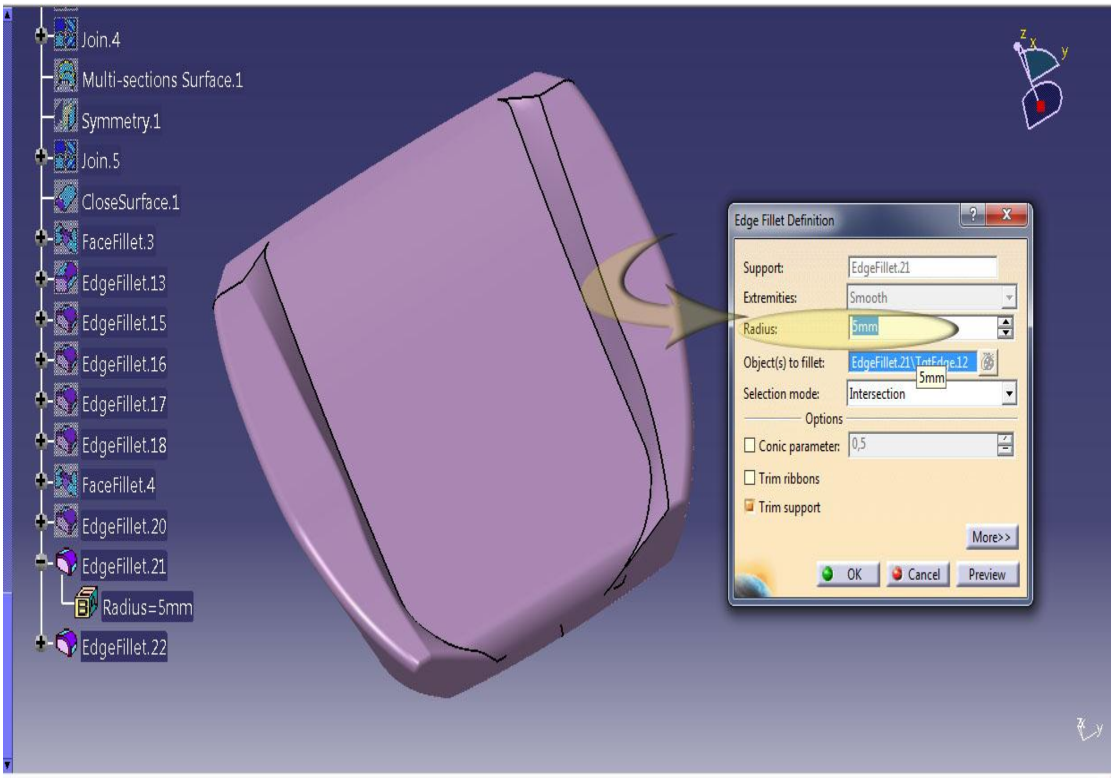
Picture 2.2.5: Filleted solid volume of the underest part.



Picture 2.2.6: Definition of the underest part dimensions.



Picture 2.2.7: Length dimension definition.



Picture 2.2.8: Rounding definition.

2.3 DESIGN OF THE HEADREST PART OF VEHICLE SEAT

The headrest part contains many similar characteristics with the previous two parts that have already been described in terms of design scope, methodology and implementation commands. A slight difference between the current part and the previous ones is that the resemblance with the actual solid can be obtained with three sections (one less compared with the other two). The design of the wireframe model followed the same structure as the previous ones, and is shown in Picture 2.3.1. In the wireframe layout it can be clearly seen that a B-Spline curve on each section has a certain curvature. This happened because of the tangency conditions that were imposed on the first point (highlighted in Picture 2.3.2) of each curve. This condition makes the B-Spline tangent to a line (the highlighted element in Picture 2.3.3) so that the tangent vector of the curve at the first point coincides with the corresponding tangent vector of each line at the same point (in terms of both arithmetic value and direction).

For this certain part, a slightly different technique was used for the creation of the part's surfaces. Instead of unifying the three curves of each section to a single entity, three different sets of surfaces were created, each of them containing the corresponding curve of each section. Each one of the three sets is shown in Pictures 2.3.4 to 2.3.6. Then, three different surfaces were created using the *"Multi-Sections Surface"* command (Pictures 2.3.7 to 2.3.9). For the implementation of the Multi-Sections Surfaces commands at the ''Sections'' option of each command, the curves that belonged to each set were imported as inputs. At the ''spine'' frame of the command the B-Spline that connects the three planes of the model was used as an input (Picture 2.3.7). In order for the mirror surface to be created the three surfaces were unified with the help of the "*Healing*" command (the essential use of this command will become clearer later in this work). The same result could have been achieved by using the "*Join*" command instead of the *"Healing"* command. Then, the construction of the mirror surface took place with the use of the "*Symmetry*" command, taking as reference plane for the mirroring operation the xz-plane and as reference surface the healing surface. After that, the two separated surfaces were joined together with the use of "*join*" command.

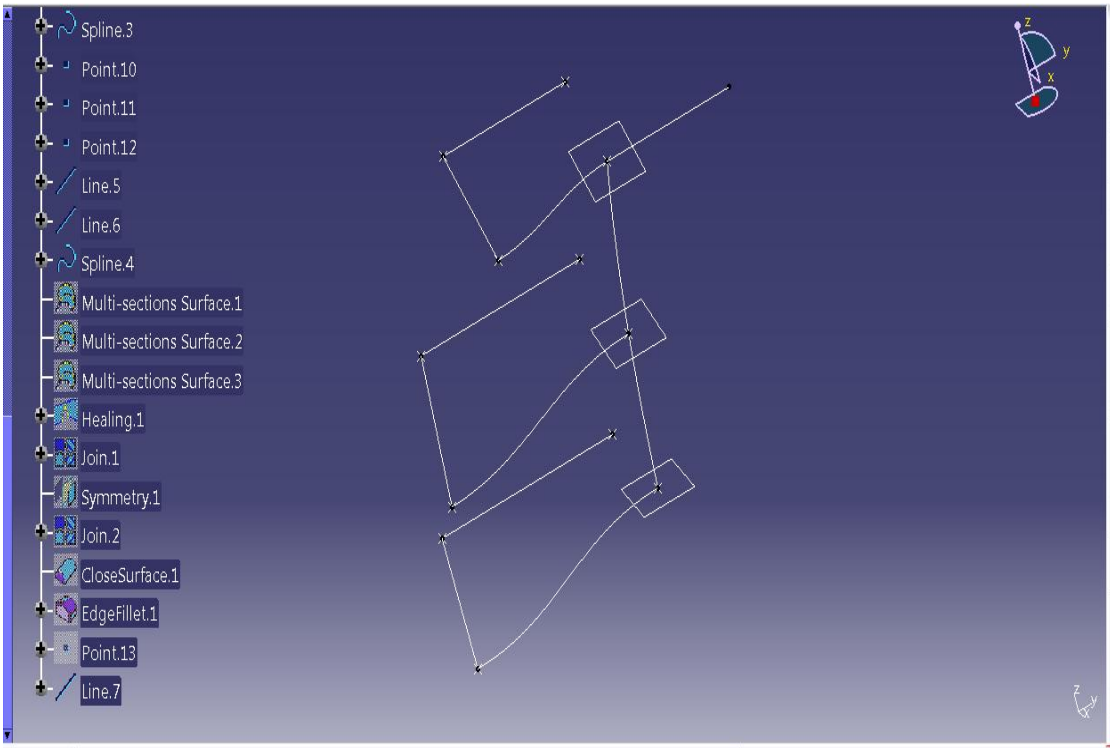
Finally, the solid volume was produced using the "*Close Surface*" volume creation command (Picture 2.3.11). Additionally, all the sharp edges of the corresponding solid were rounded using the "*edge fillet*" command. The curvature radius of the rounding used was set to 20 mm (Picture 2.3.12). The final solid volume of the part is depicted in Picture 2.3.13.

The final stage for the completion of the geometry was the assignment of proper values to the corresponding dimensions of the corresponding part. The most significant dimension components that were determined was the length, height, and width of the part; additionally the distance between the part's sections was defined.

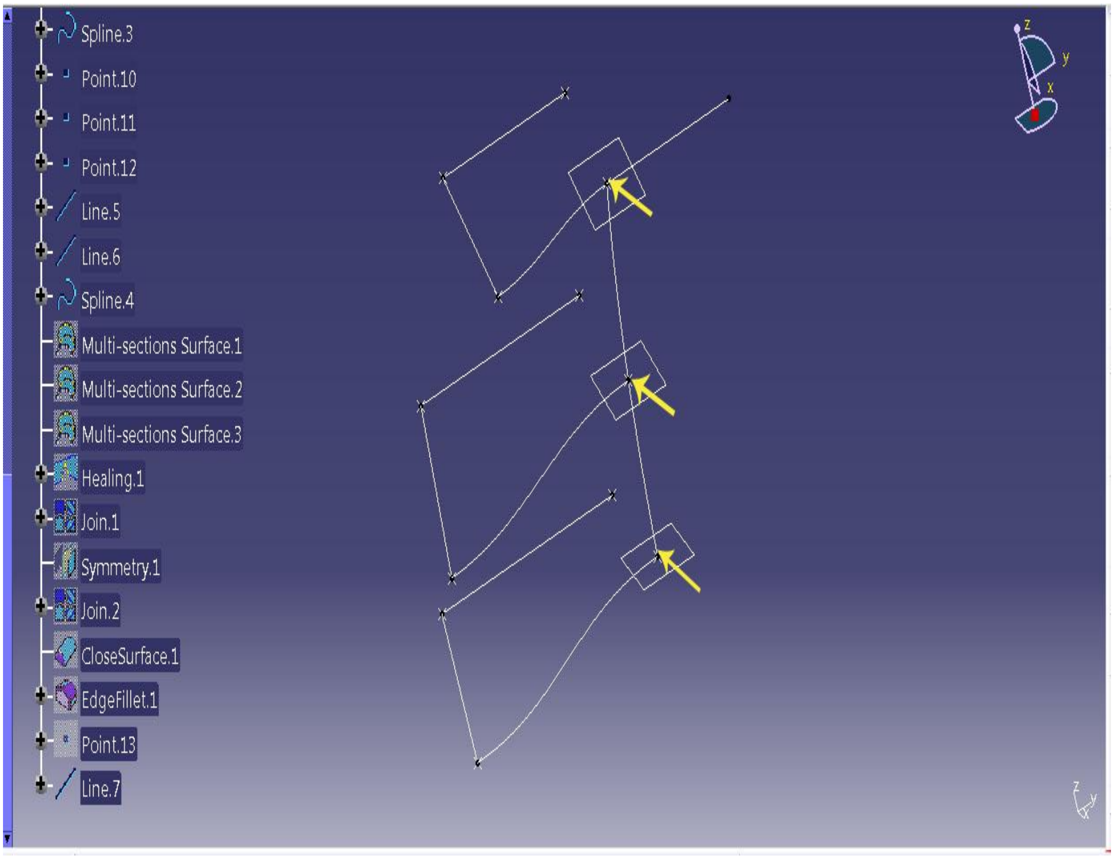
The exact value for each assigned dimension inside the part is depicted in Table 2.3.1.

Table 2.3.1

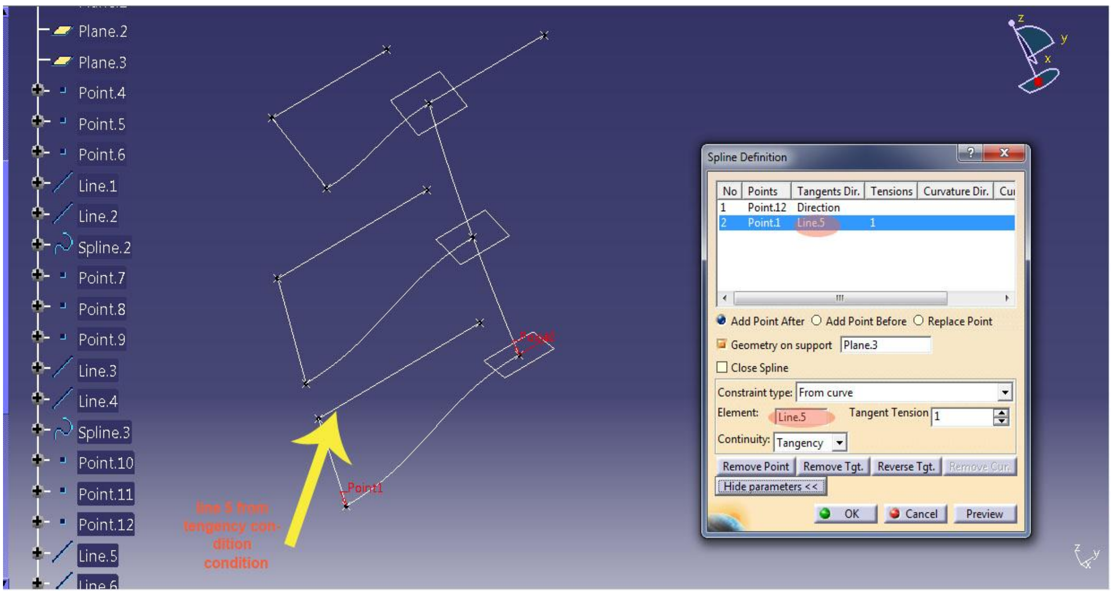
Dimensions (mm) \Sections	Section 1	Section 2	Section 3
Length	210	300	320
Width	100	100	100
Distance From Previous Section	-	130	100
Total Height	250	-	-



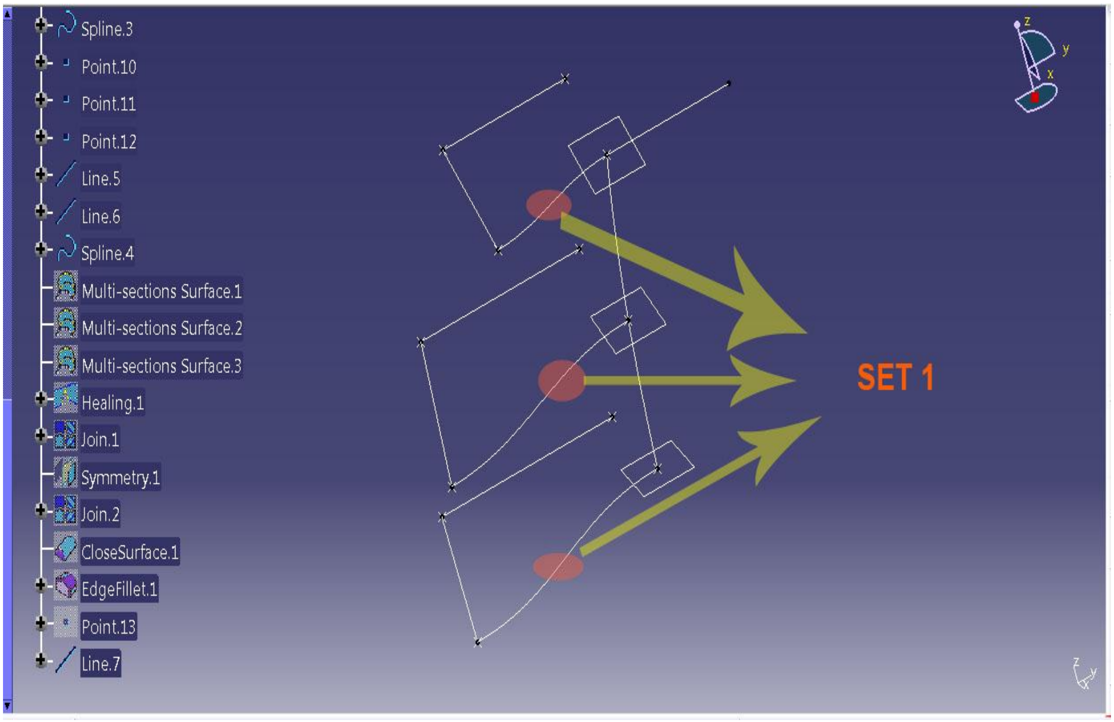
Picture 2.3.1: Headrest wireframe model.



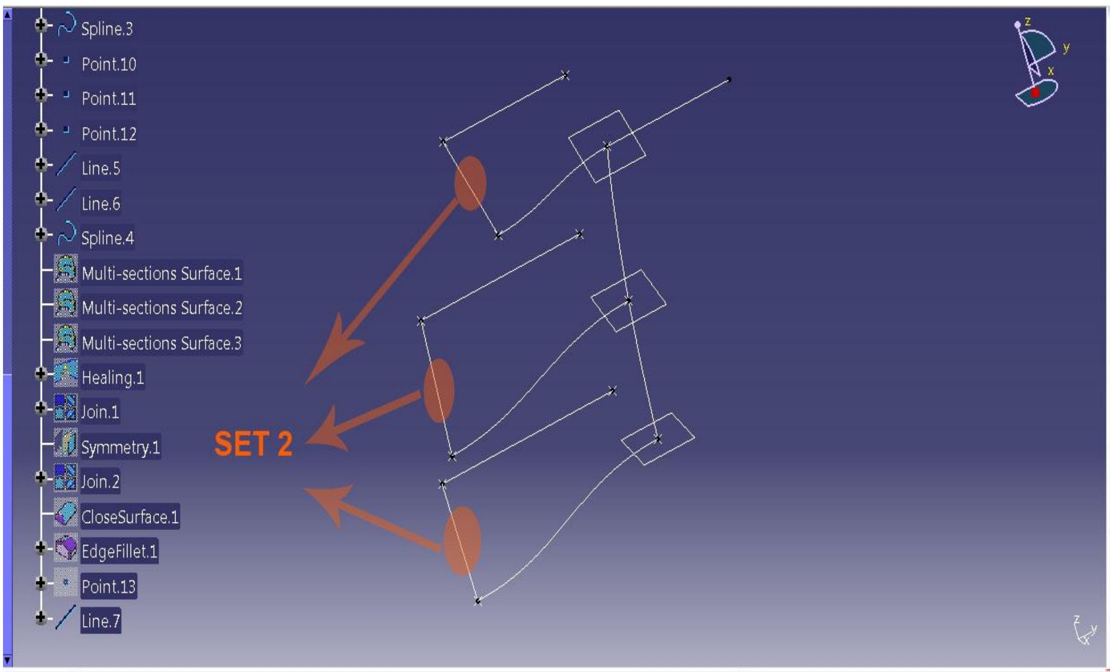
Picture 2.3.2: Points with implied tangency constraint.



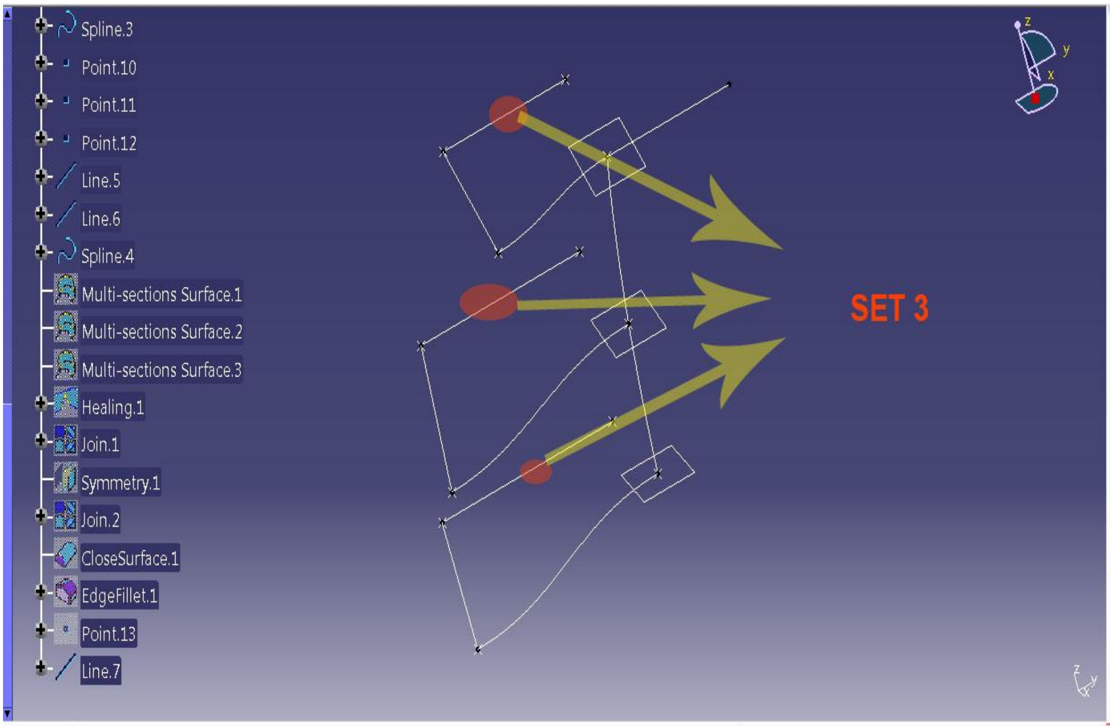
Picture 2.3.3: Definition of tangency condition at B-spline curve of first section.



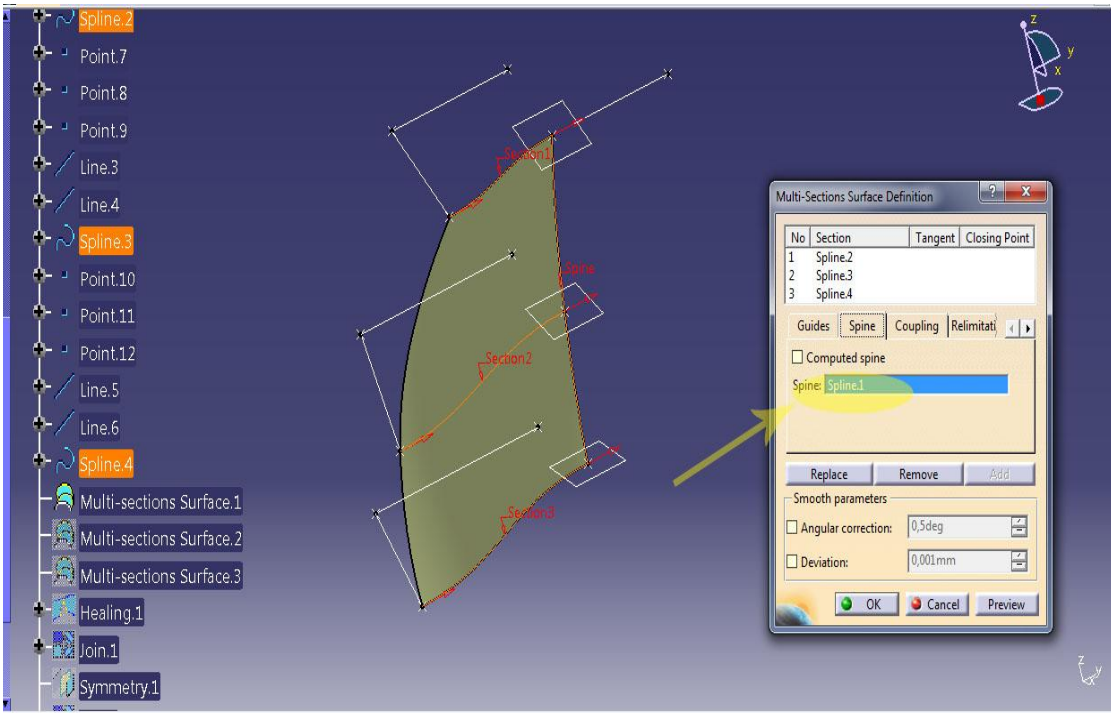
Picture 2.3.4: Definition of the first set of curves for the creation of the first surface.



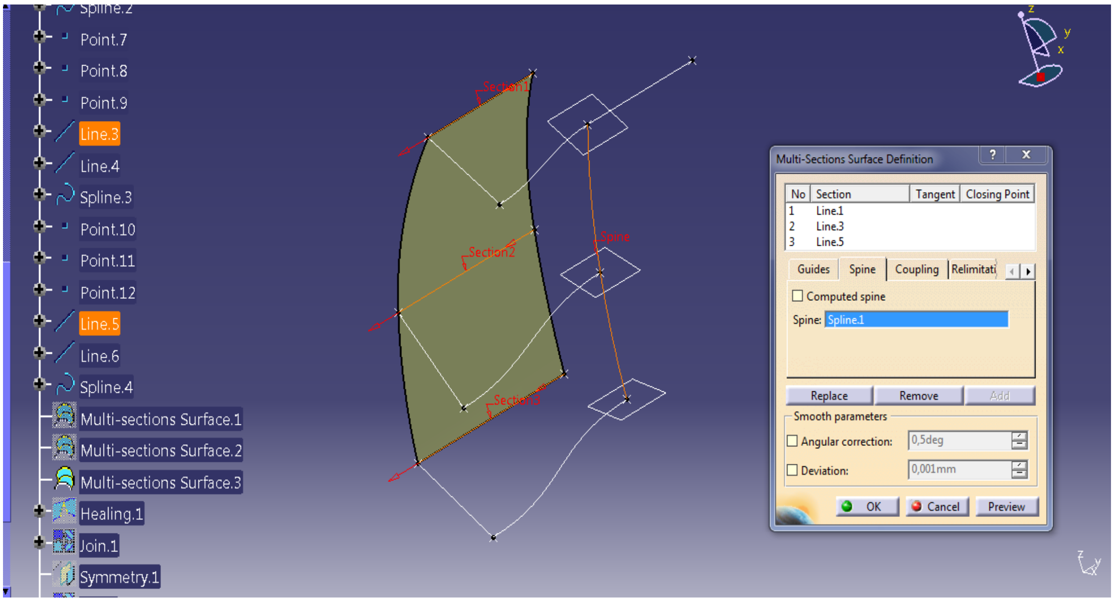
Picture 2.3.5: Definition of second set of curves for the creation of the second surface.



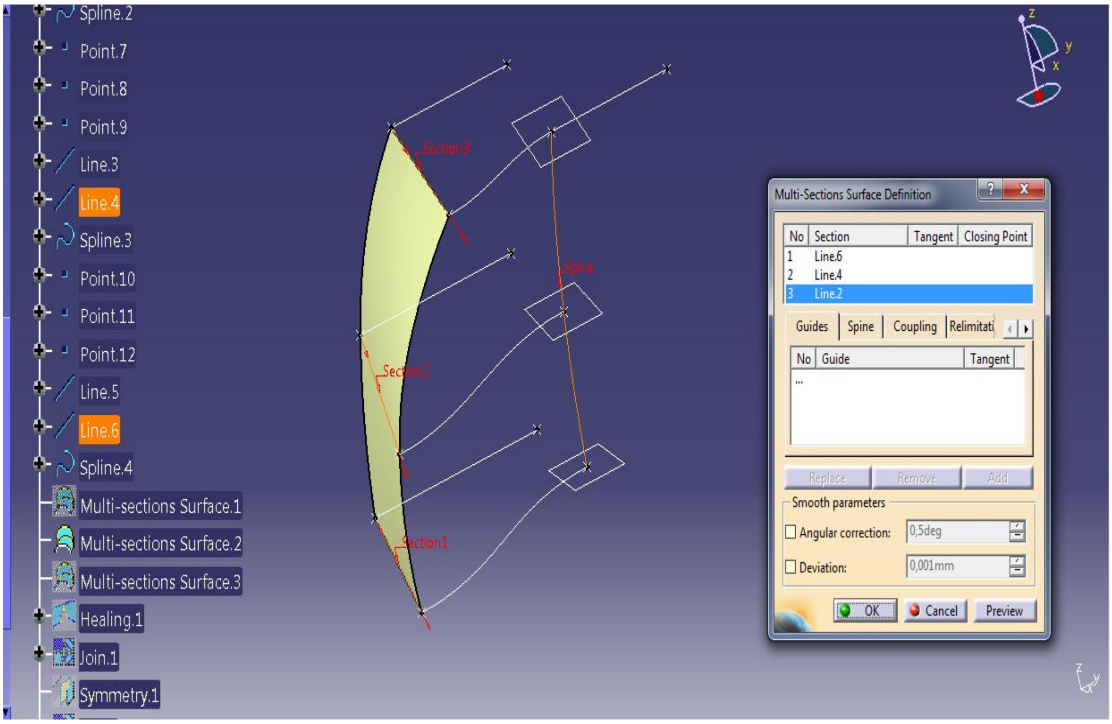
Picture 2.3.6: Definition of third set of curves for the creation of the third surface.



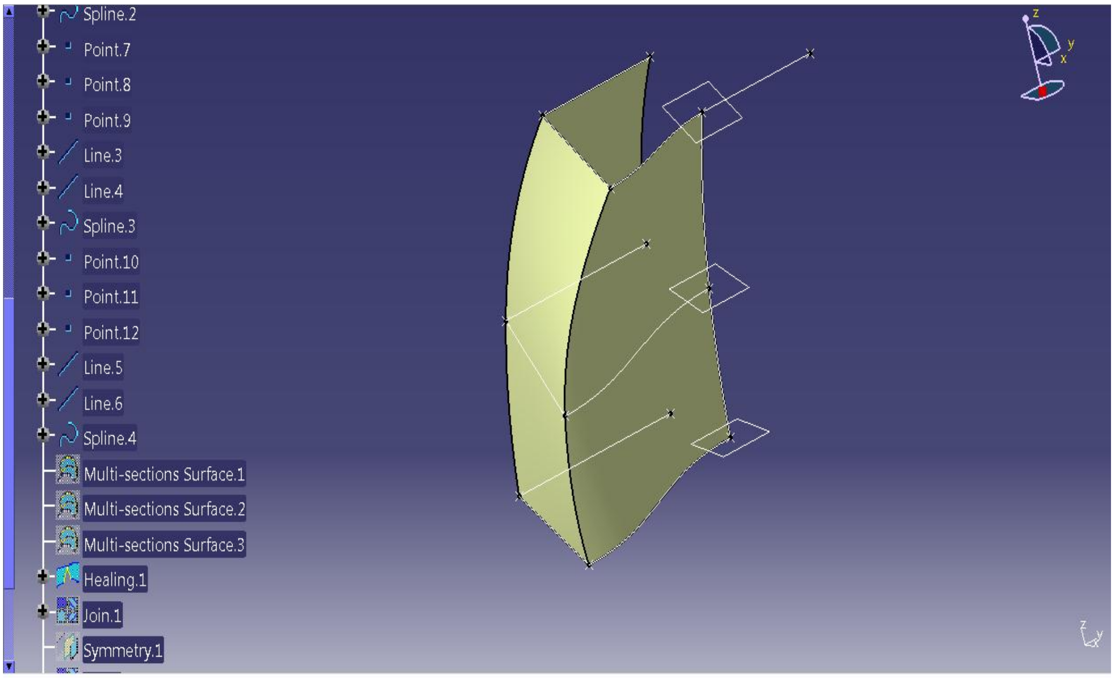
Picture 2.3.7: Definition of the first multi-section surface.



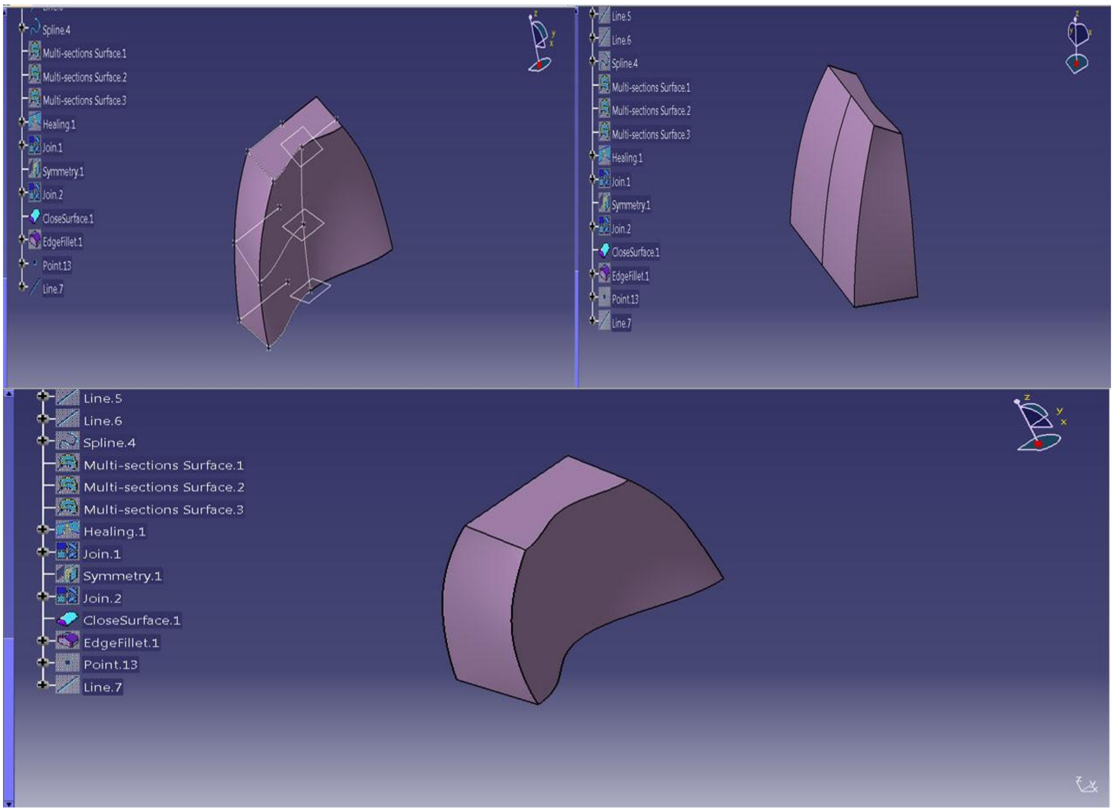
Picture 2.3.8: Definition of second multi-section surface.



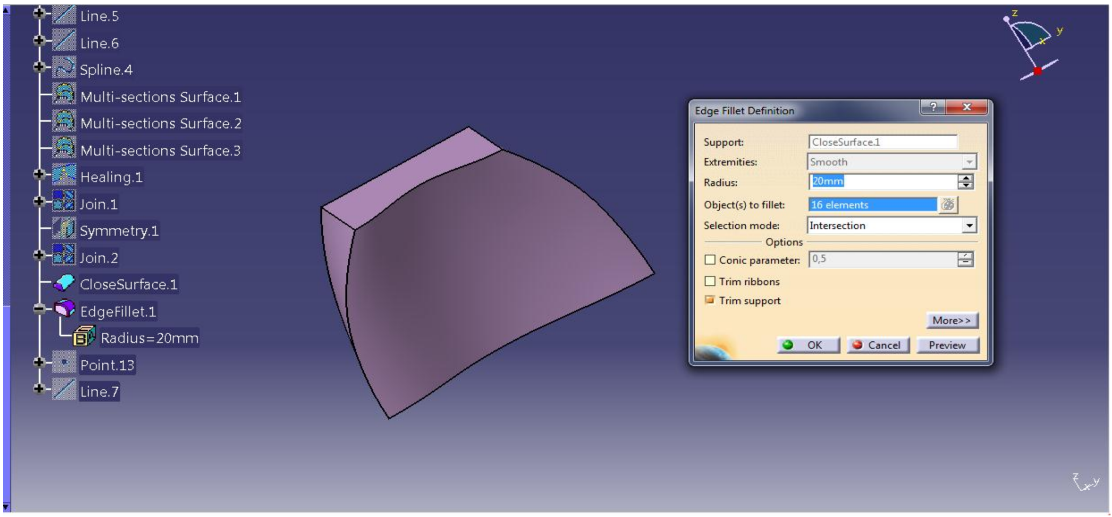
Picture 2.3.9: Definition of third multi-section surface.



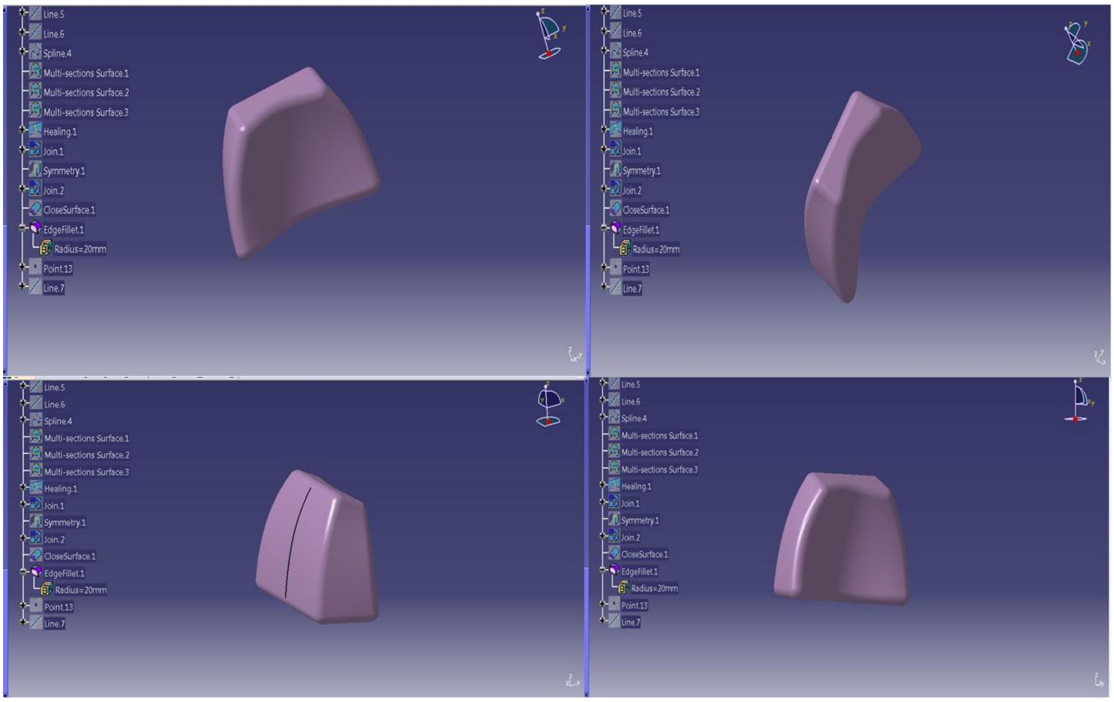
Picture 2.3.10: Healing of the headrest surface.



Picture 2.3.11: Headrest solid volume construction.



Picture 2.3.12: Definition of the curvature radius for the filleting operations.



Picture 2.3.13: Final 3D volume of the headrest part.

2.4 DESIGN OF THE STEERING WHEEL PART

For the design of the steering wheel part, a significant effort was undertaken to reconcile functionality with style into an innovative solid geometry. For this purpose it was difficult to use an actual, already existing solid steering wheel as a prototype for the digital representation of geometry. As a result, the source for the external shape was derived from pictures of conceptual steering wheel designs. Moreover, in order to satisfy the criteria of implementabilty and manufacturabilty of the steering wheel digital mock up, the dimensions used were as measured in a real steering wheel inside an existing urban car design.

In order to produce the final solid geometry it was divided into several solid modules. The methodology for designing each one of these modules followed similar steps as did the design of the seat parts previously presented. These steps were the following: a) definition of the wireframe model as a design basis, b) construction of the surfaces, and finally c) production of the solid representation. The first designing module was the steering wheel. The wireframe model of this part was more sophisticated than the previously designed parts for the seats. At the first stage, a B-Spline curve was created acting as the ‘’spine’’ of the final volume (Picture 2.4.1). After this, a number of planes were defined on the control points of the B-Spline, vertical to the curve. This was implemented by choosing from the plane definition command, the ‘’normal to curve’’ option (Pictures 2.4.2 and 2.4.3). As shown in Picture 2.4.3, the volume has a symmetry with respect to the xz-plane. Then, the first two sections were created on the first two planes of the wireframe model. The geometry of the first section was a Spline while the corresponding geometry at the second section was a circular segment, which was defined by the three points belonging to the second plane (Picture 2.4.4). From these two sections the first surface of the part was constructed, using the "*Multi-Sections Surfaces*" command. Another two similar sections were created upon the fourth and fifth plane of the model. Then, by using the "*Multi-Sections Surface*" command, the second surface of the steering wheel was created. At section option inside the command definition window, the curves belonging to the second, fourth and fifth plane of the wireframe were selected as inputs. For the creation of this surface the same curve was used as a spine, as in the previous “*Multi-Sections*” command (Picture 2.4.6).

As a spine input inside the command the B-Spline curve was used, the same one that was used to connect the planes of the sections (Pictures 2.4.4 and 2.4.5). At the fifth and sixth plane of the model, two more circular segments were created, with a radius two times lower in approximation in comparison to the circle of the fifth section (Picture 2.4.7). These two curves were the basis for the construction of a steel cylindrical segment that acted as ‘’the bridge’’ between the two symmetrical steering wheel handles. Following, the lower surface of the cylinder was created as a Multi-Section Surface having as sections the two circular curves and as spine the Spline that connects the six planes of the wireframe model (Pictures 2.4.8 and 2.4.9). The next step, following the part design, was the covering of the openings, derived from the module’s wireframe model, in order to build the first symmetrical half of the steering wheel. For this purpose, on the first and the second plane of the section two more curves were drawn, a Spline and a circle on each plane respectively (Picture 2.4.10). The newly created curves tend to be similar to the already existing ones. Taking these two sections as input, an additional multi sections surface was created (Picture 2.4.11). With the same technique, symmetrical curves were designed on the fourth, fifth and sixth planes of the layout. Similarly, taking all the above curves as a single input for the Multi Sections command, the remaining part of the cover surface was defined (Picture 2.4.12). The entire cover surface of the steering wheel was almost completed. Likewise, symmetrical curves were created on the fifth and sixth planes on the initial spine curve. These curves were both circular segments. Using these curves as a base, the symmetrical cylindrical surface was created in the same way as with the other surfaces (Picture 2.4.13). By joining the resulting surfaces together, the solid volume was created with the "*Close Surface Technique*" (Picture 2.4.14). The other half was created using the "*symmetry*" operation. The xz-plane was used as a symmetry plane (Picture 2.4.15).

The second volume of the steering wheel part was the driver side air bag, which was attached to the steering wheel with two additional support volumes. An important feature of this volume was the displayed symmetry with respect to the xz – plane. Due to symmetry, only half of the volume needed to be constructed. The first section was created by initially defining the wireframe geometry. This section was designed slightly protruding from the steering wheel. The exact distance of protrusion of the first section was 25 mm from the steering wheel. The section is depicted in Pictures 2.4.16 and 2.4.17. This section consisted of three curves, one B-Spline that connected five of the peripheral points together, and two more lines that connected three distinct points in order to close the section. Then, the second section was designed to have a similar shape as the first section with the difference that the second is even more extensive in terms of length and height in comparison to the first. The distance between these two sections was 70 mm, towards the negative direction of x-axis (Picture 2.4.18). Then, a unification of the section was a mandatory task that should be performed to the section in order for it to be eligible for the solid creation process. The final modification into the wireframe model was the designation of the guide curves. These curves were used in the solid creation phase, in order to approach the conceptual geometry more accurately (Picture 2.4.19). A notable detail in the wireframe's design is that each Spline at the anchor points of the second section is tangent to a virtual vector whose direction is parallel to the line appearing next to the steering wheel (as shown in Picture 2.4.20). This happened because this certain line was used as input for the tangency condition option, during the execution of the definition command of each Spline. Due to the convenient geometry of the sections, it was made easy to approach the desired solid geometry by using a single solid creation command. By using the "*Multi Sections Solid Volume*" command, the desired volume emerged by inserting the two unified curves as inputs inside the sections option and the four Spline curves as Guides (Picture 2.4.21). For reasons that are imposed by decorative criteria, a rounding was imported at the edge of the front surface of the solid volume of the driver side air bag with a curvature radius of 15 mm (Picture 2.4.22). To conclude this module design, the second half of the volume was formed with the symmetry operation command, taking the xz-plane as the symmetry plane (Picture 2.4.23).

Next, the steering column cover was designed, that is the volume which will comprise the steering wheel with the dashboard cockpit during the assembly. The wireframe model was determined first. This certain wireframe is a lot more sophisticated than the previous ones. It contains an inner section apart from the primary external one. The inner one forms a recess that serves only styling and aesthetic purposes (Picture 2.4.24). The second section was designed at a distance of 200 mm away from the first one, in the negative direction of x-axis and is 47 mm lower as compared to the first one. Unlike the first one, this section is planar and similar to the first in terms of length and height (Picture 2.4.25). After the design of the wireframe model, the construction of the surfaces followed. At first, the surface that covers the front view was created. Due to the coexistence of two wireframes the surface that covers the front view had to be divided in two segments for the total surface to be created. The first surface of section number one was a multi section surface, which was created by using the two smaller lines, part of the curve that closed the section, as inputs for the command's "section" option (depicted in Picture 2.4.26). The two Spline curves of the total wireframe were used as guides, both the external non-planar and the internal planar one. The resulting surface is shown in Picture 2.4.27. The inner surface of the section was created by using the "*Fill*" surface command, inserting as inputs the inner B-Spline of the wireframe and the two remaining lines of the section (Picture 2.4.28). The reason why the "*multi section surface*" command was not used is because the three inserting curves were planar. For the purpose of constructing the peripheral surfaces of the geometry, three more Splines were created, in order to be used as generative curves for the later surfaces. These curves are represented in Picture 2.4.29. The first peripheral surface was constructed as a multi-section surface. The first two Splines were set as inputs at the "section" option while the two Splines of the first and second section were used as guide curves (Picture 2.4.30). The next surface in sequence was created in a counterclockwise direction as a multi section surface (Picture 2.4.31). The Spline curve, which was used as a second generative into the first multi section surface, was used as the first generative curve here. The second generative was assigned to the last Spline at the same direction. The guide curves were the same as in the previous surface. Then the lateral planar surface of the volume was created using "*Fill Surface*". All the layout curves of the planar surface were imported as the boundaries of the newly created surface (Picture 2.4.32). In the same way, the planar surface of the second section was created. The closed layout which consisted of one B-Spline and two lines was imported inside the boundary ''frame'' of the *"Fill"* command (Picture 2.4.33). With the creation of the surrounding surfaces of the volume, the construction of the solid volume was almost ready to be realized. Before this realization, the set of surfaces had to be unified with the "*Join*" operation. After the integration of the surrounding surfaces into a single one, the solid volume was generated using "*Close Surface*" command (Picture 2.4.34). The second symmetrical part of the volume was made by using the symmetry operation while the front sharp edges were rounded by the "*edge fillet*" command, with a radius of curvature of 0.5 mm as shown in Picture 2.4.35.

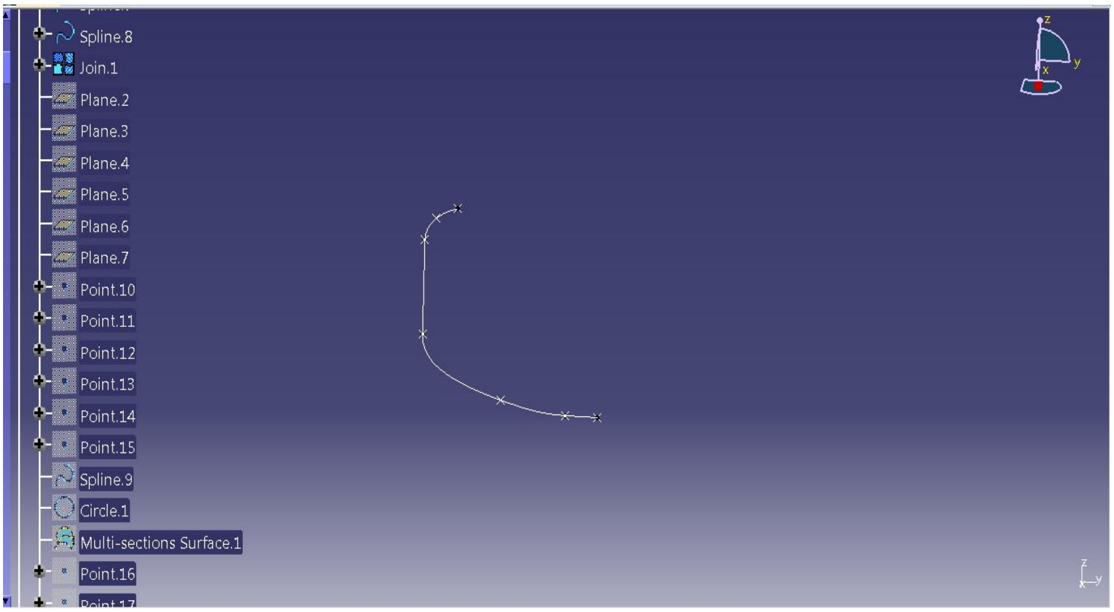
The next volume created was a connective one between the driver side airbag and the steering column cover, defined as steering wheel hub. The steering wheel's hub volume was a lot simpler in its creation than all the previous ones. By using the unified wireframe of the second section of the driver's side airbag an extruded volume was created. The length of the new volume was 60 mm (Picture 2.4.36). The second symmetrical half of the extruded volume was created using the symmetry operation, taking as symmetry plane the xz-plane (Picture 2.4.37).

The next volumes that were designed were the steering's wheel pillars. These pillars function as a cohesive device between the driver side air bag and the steering wheel. The first stage was the creation of the wireframe model (Picture 2.4.39) while the surface creation followed. In order to finalize the volume, a surface should be created on every hydra of the wireframe model. Firstly, the adjacent surface to the steering wheel was created. The creation of the adjacent surface was made using the *"Fill"* command (Picture 2.4.40). The other peripheral surfaces were constructed as multi-section surfaces. As generative curves, the connective Splines between the driver side airbag and the steering wheel were used, while the adjacent curves upon the two volumes were used as guides (Picture 2.4.41). Once the surfaces had been created, the next phase involved the unification of the set of surfaces. The solid volume was produced with the "*close surface*" command (Picture 2.4.42) and it was completed by inserting some edge fillets and performing a symmetry operation for the definition of the symmetrical second half of it (Picture 2.4.43).

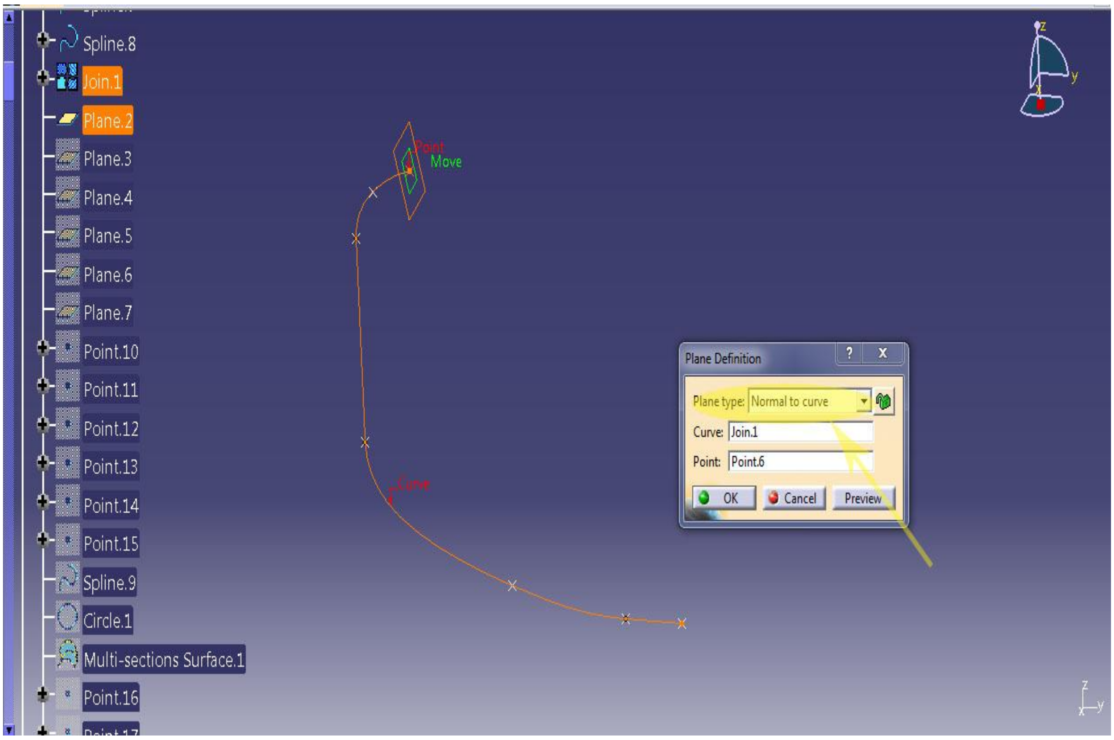
Next in sequence was the creation of the volumes where the indicator lights levers were attached. First the wireframes of the volume were created. For the construction of the volume, two wireframe sections were needed. The first section was a circle and was created within a distance of 20 mm from the steering column cover. The circle was created taking the equidistant plane as support, as shown in Picture 2.4.44. Additionally, the section number two was created as a rectangular wireframe model by defining the edge points of the geometry as points on the steering column cover's peripheral surface (Picture 2.4.45). For the stage of the surface creation, the existence of a unified curve of the second section was mandatory. For this reason, the four lines that the second section consisted of were unified by the "*Join*" command. Further, the surrounding surface was created using the "*Multi sections surface*" command. Inside the command window the circle curve and the previously unified rectangular curve were imported as inputs in the *''sections''* option. For the realization of the current operation no guide curve was inserted (Picture 2.4.46). In order to construct the solid volume two more surfaces were created, filling the two sections' curves. The resulting surfaces were created by using the "*Fill surface*" operation (Picture 2.4.47). The solid volume was created with the same "*Close surface*" command as with the other volumes (Picture 2.4.48). From the current volume geometry, it is shown that a recess needed to be created where the indicator light could be attached. The Catia software does not include any operation for direct material removal. As a result, the removal should be defined implicitly. This means that the recess resulted from two separate volumes intersecting each other. The idea of creating the desired final volume was to remove one volume from the other, so that the recess could be formed. The operation which performs such a task is the

“*Remove Volume” Boolean* operation. This operation required two volumes for the removal to be executed. The first was the base volume, part of which was removed while the second was the auxiliary volume, which determined the area to be removed. As base volume, the one that protruded from the steering column cover was used. As auxiliary volume an extruded one was used, whose wireframe was a circle, created on the external surface of the steering column cover (see Picture 2.4.49 below). The resulting volume is depicted in Picture 2.4.50. For aesthetic purposes two filletings were inserted in the front sharp edges of the volume (Picture 2.4.51). For the solid model to be completed, a second symmetrical volume was needed at the other side of the steering column cover. The second mirrored part was created by using the Symmetry operation. The result is depicted in Picture 2.4.52 below.

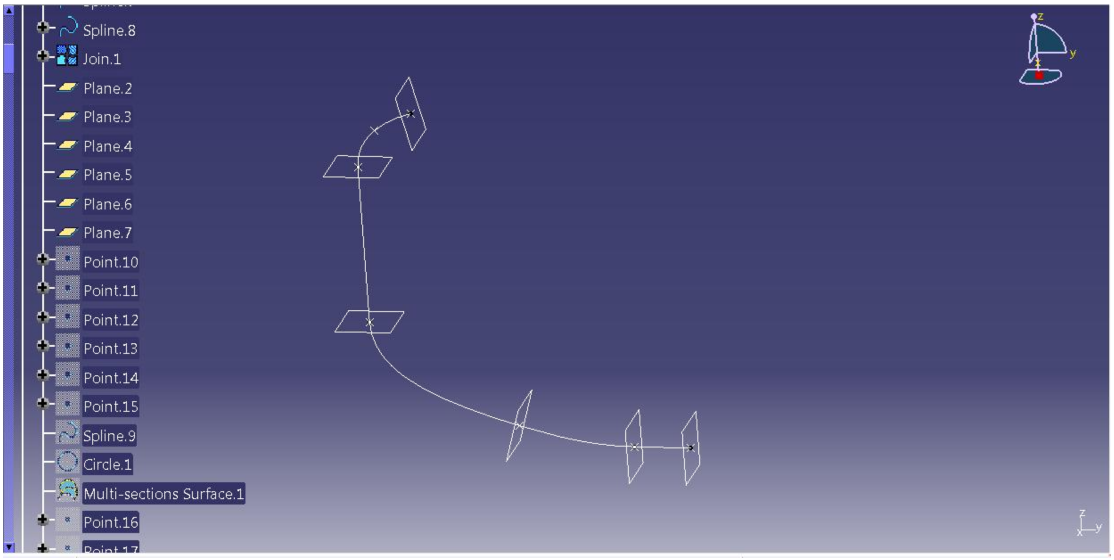
The final module that was integrated into the design, mainly for purposes of completion, was the ignition. In order to create this volume it was necessary to define the wireframe model first. Two circles were created, on different planes with a distance of 14 mm between them. The first circle had a larger radius than the second one (Picture 2.4.53). The wider circle’s wireframe was projected on the steering column’s cover external surface (Picture 2.4.54), which will be further explained at the creation of the surface’s stage. Next, the two circle curves were used to create two fill surfaces inside each section by using fill surface operation (Picture 2.4.55). Then, the peripheral surface was created with the use of the “*multi section surface*” command. As generative curves for the creation process the two layouts of the two filled surfaces were used, while for the creation of the multi section surface the definition of guide curves wasn’t needed (Picture 2.4.56). The solid volume at its primary form was created by joining together the three surfaces, with the help of the “*Close Surface*” command (Picture 2.4.57). For functional reasons a hole should be formed on the solid geometry. The technique used for the design of the hole of the ignition was similar with the one used on the steering column cover for the indicator light. More specifically, on the external surface a circle was designed using as reference the plane’s definition point where the second section was created. The radius of the circle was 6 mm (Picture 2.4.58). Then, by using the circular profile as a generative curve, the auxiliary extruded surface was created. The “*Extrude Volume*” command was used for the direct creation of the solid volume (Picture 2.4.59). Afterwards, the hole was created by using the “*remove volume*” operation, by defining the initial ignition volume as a base object and the extruded intersecting volume as the one to be removed (Picture 2.4.60). In conclusion, the external and internal circular layouts of the front face of the ignition volume were rounded, using the “*edge fillet*” operation. The steering wheel’s final part is depicted in Picture 2.4.61.



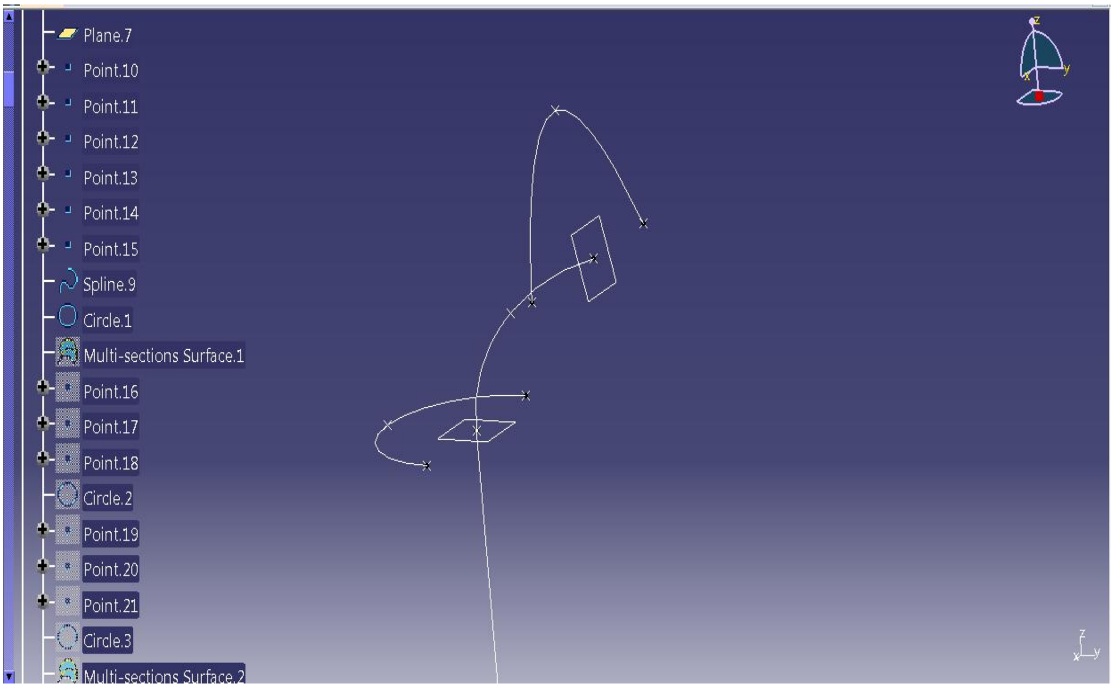
Picture 2.4.1: The construction of the steering wheel’s spine.



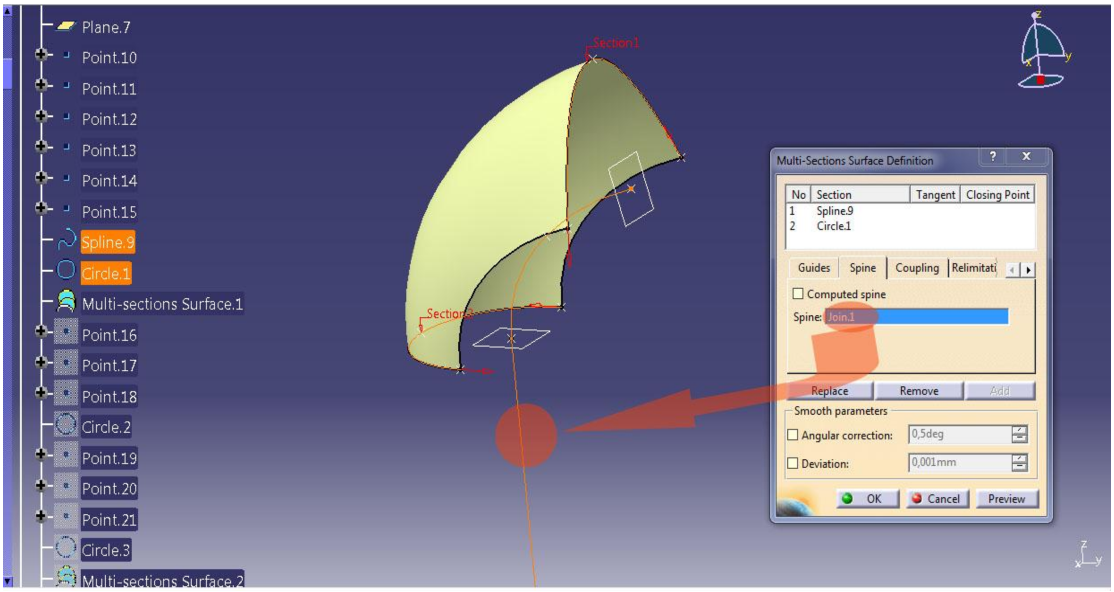
Picture 2.4.2: Plane definition normal to the steering wheel handle's spine.



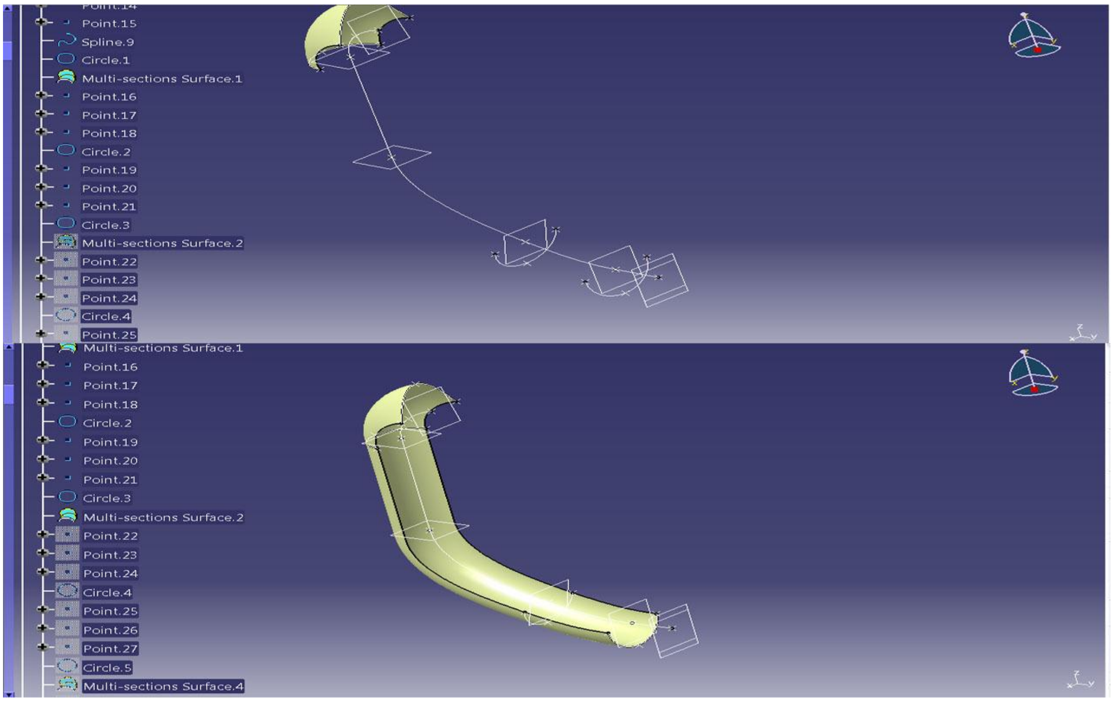
Picture 2.4.3: Planes normal to the spine of the handle.



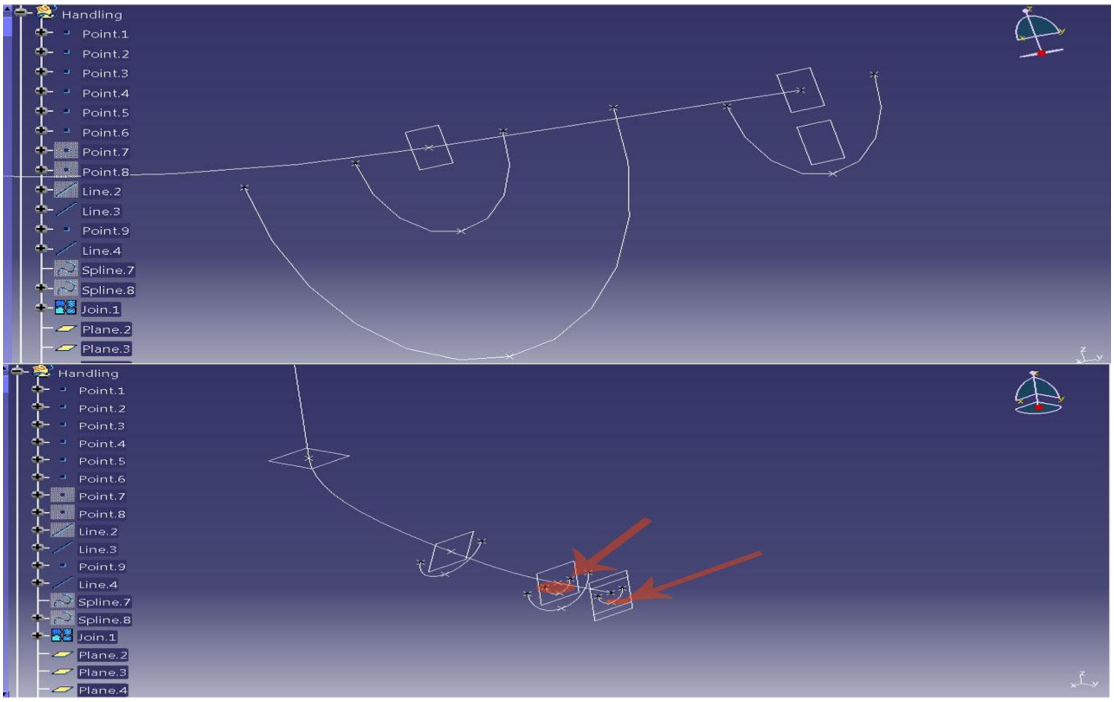
Picture 2.4.4: First two sections of the handle module.



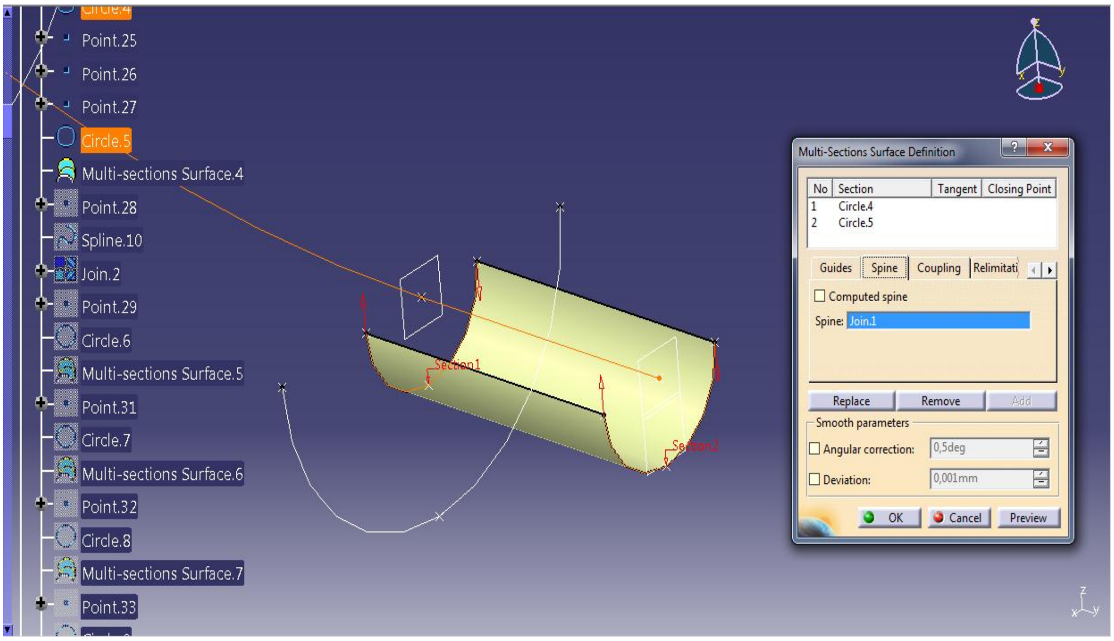
Picture 2.4.5: Initial multi-sections surface for the handle of the steering wheel.



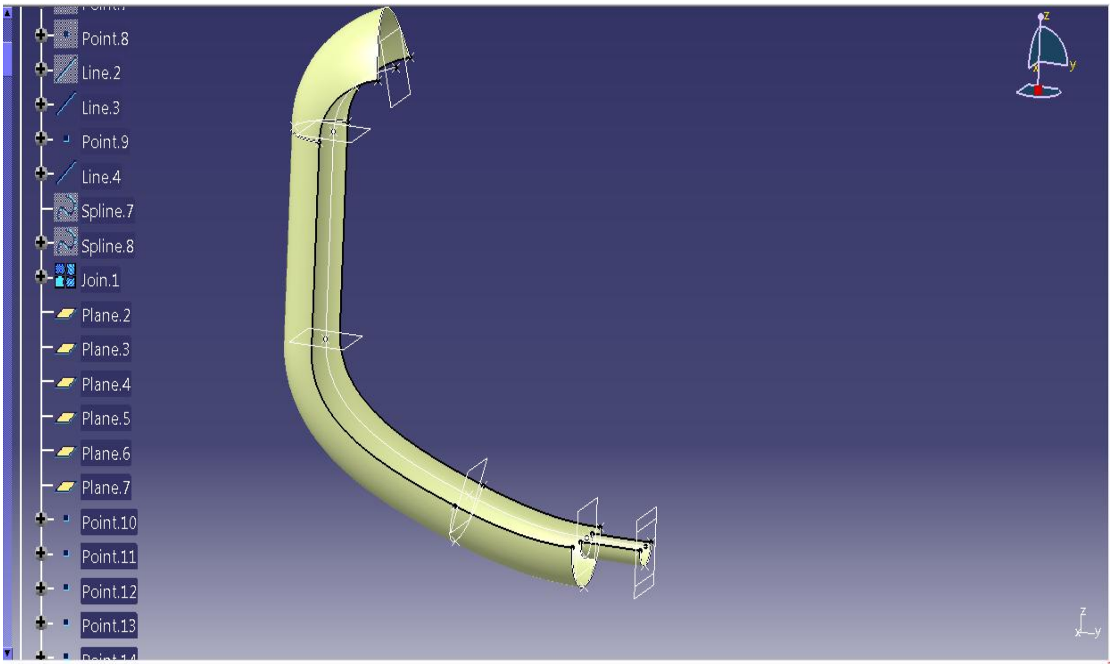
Picture 2.4.6: Multi-sections surface of the steering wheel module.



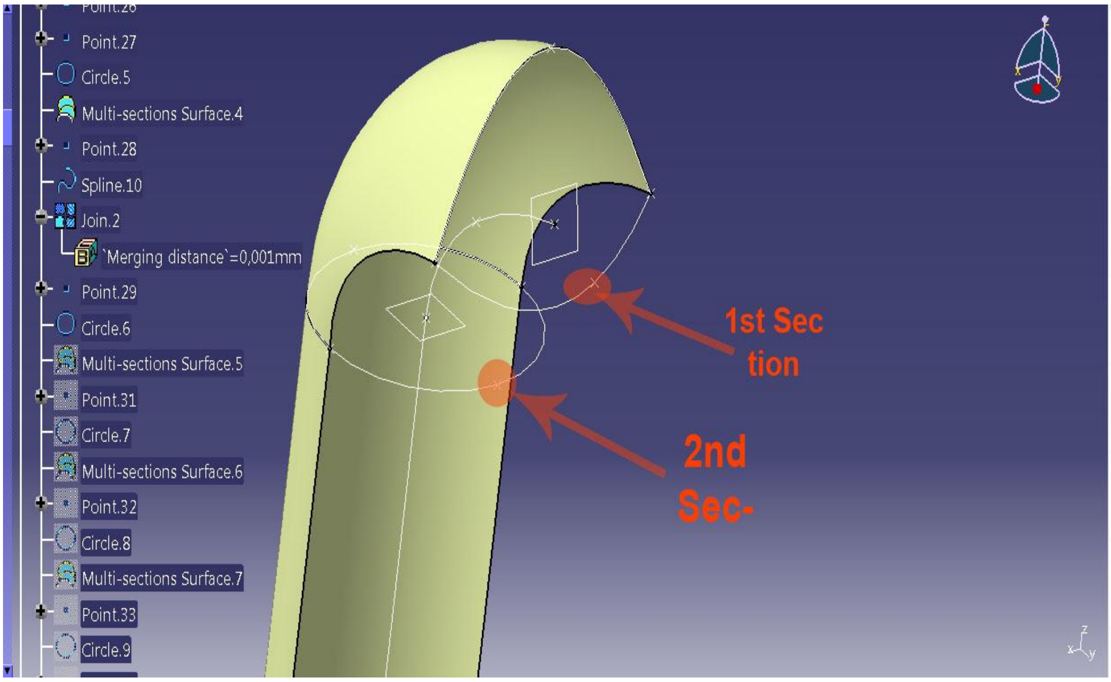
Picture 2.4.7: The wireframe of the connective steel cylinder.



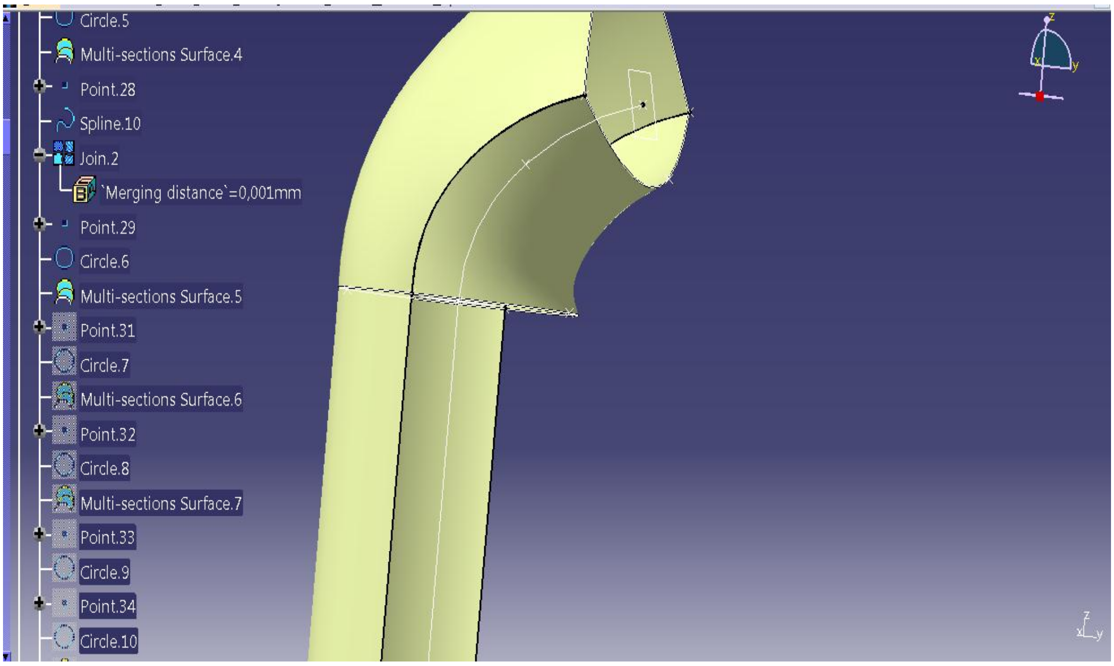
Picture 2.4.8: Cylindrical steel surface.



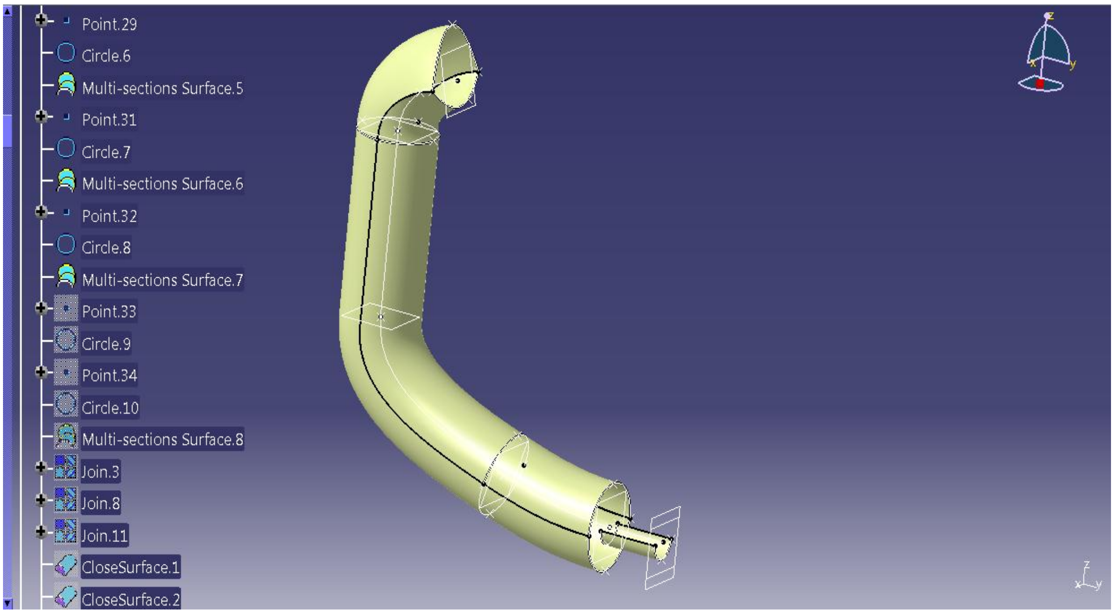
Picture 2.4.9: Steering wheel surfaces.



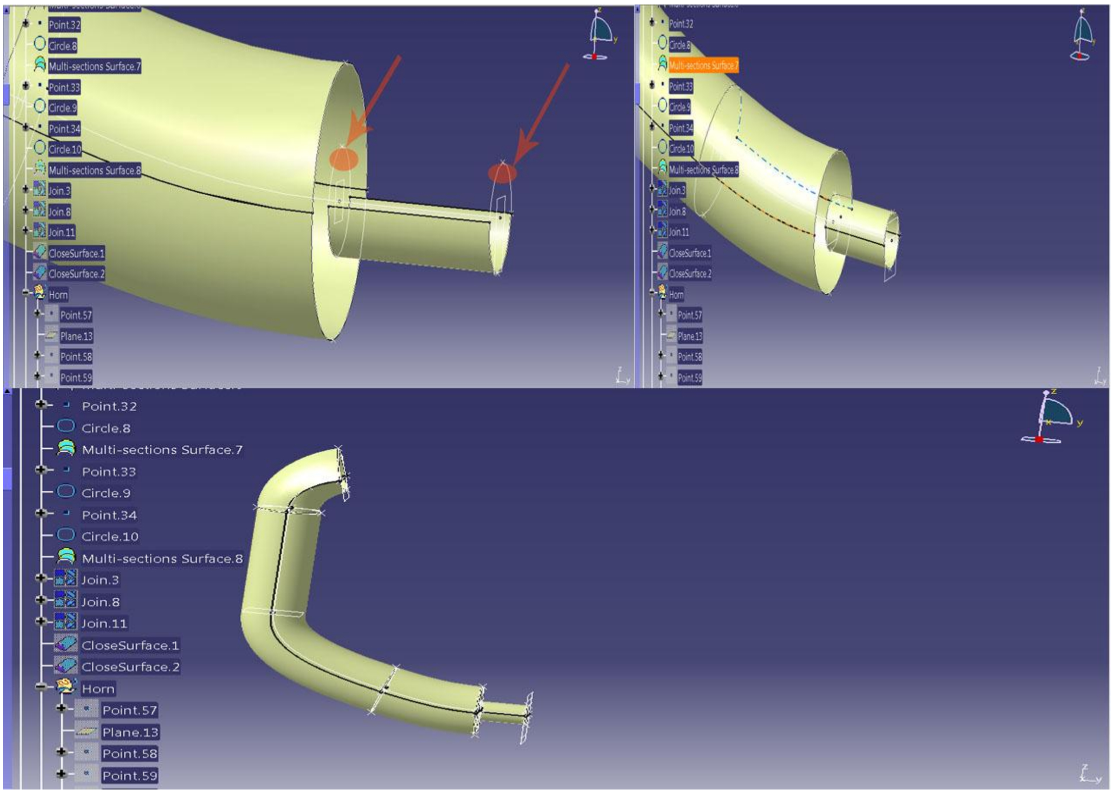
Picture 2.4.10: Cover surface for the steering wheel.



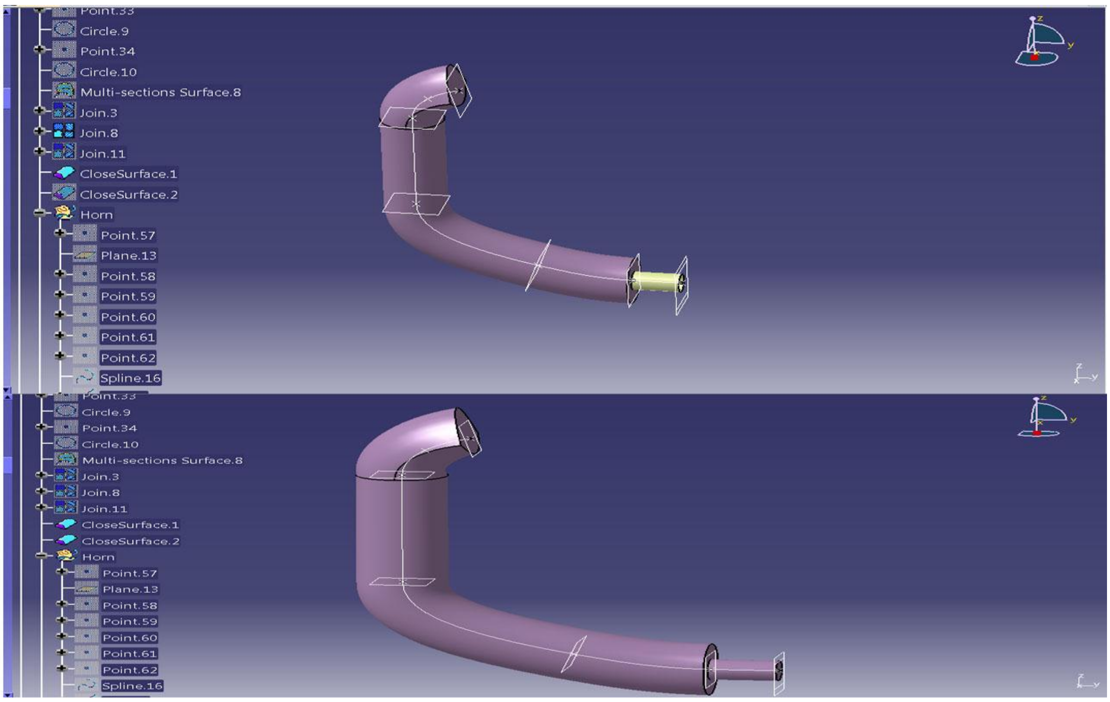
Picture 2.4.11: Steering wheel's cover surface.



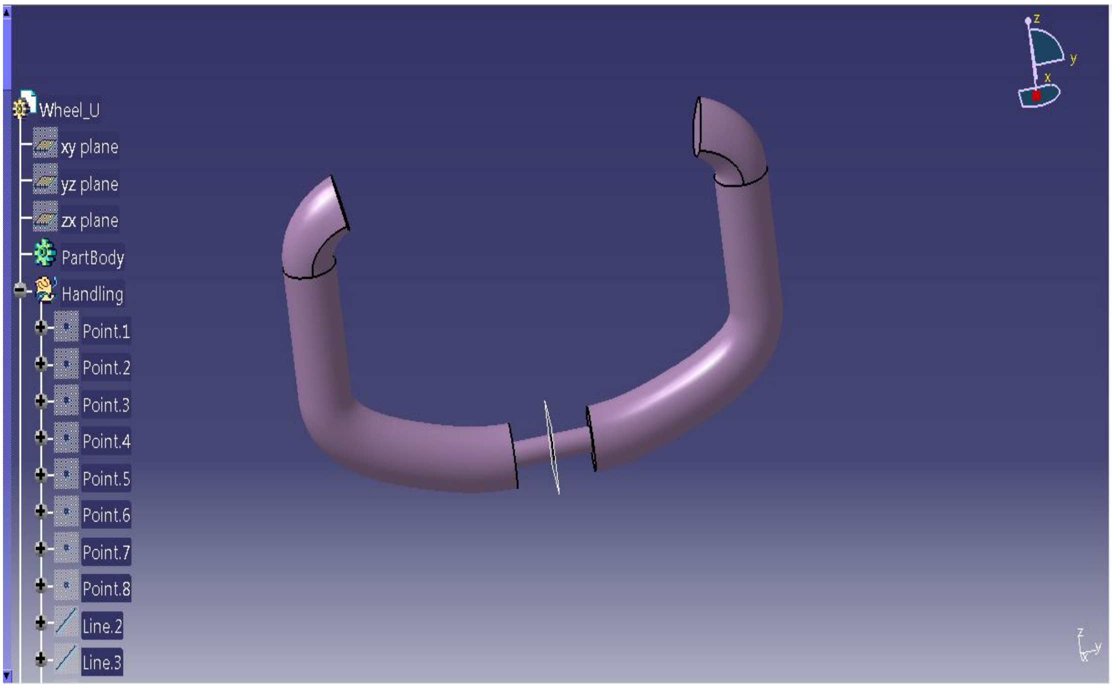
Picture 2.4.12: Steering wheel's cover surface.



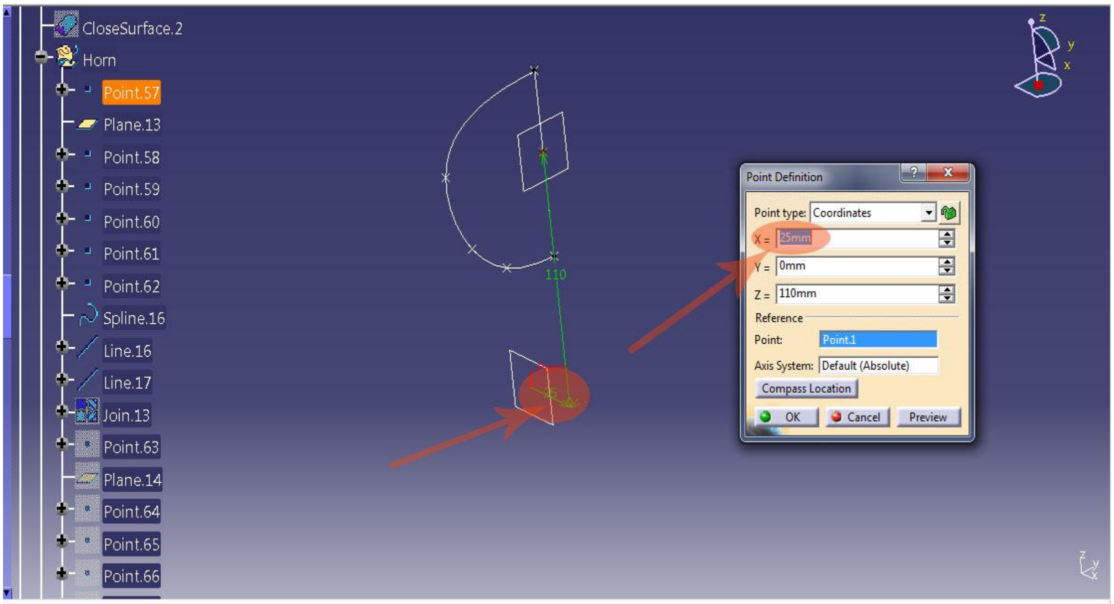
Picture 2.4.13: First symmetrical external surface for the steering wheel.



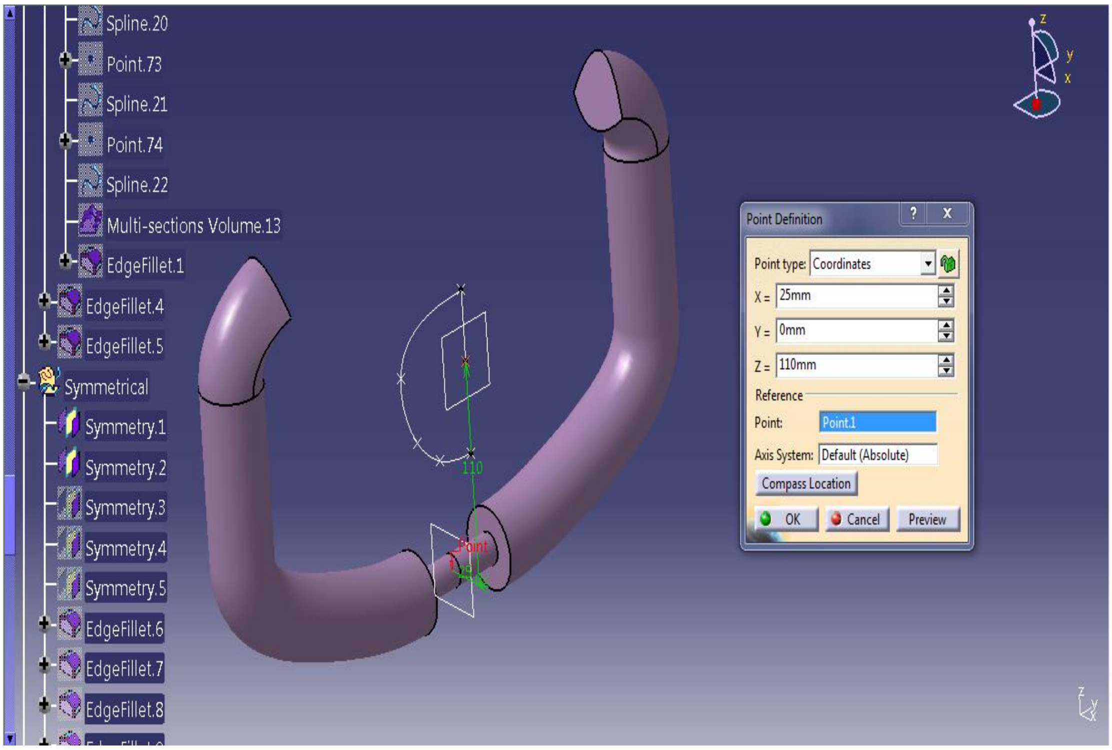
Picture 2.4.14: Steering wheel's volume.



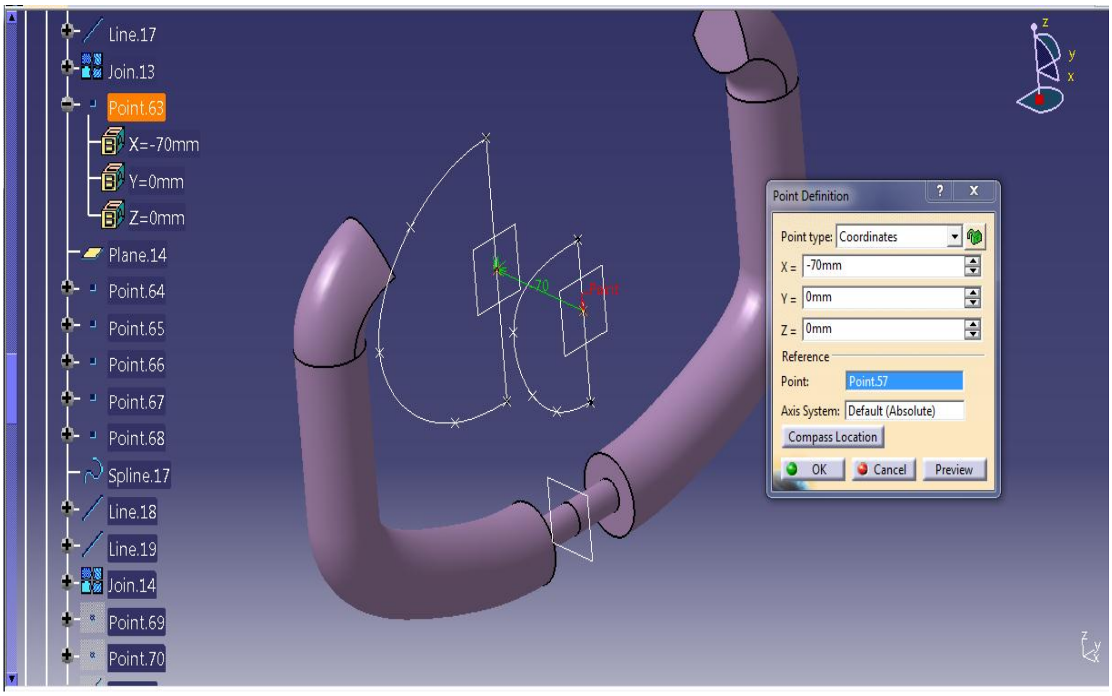
Picture 2.4.15: Steering wheel's complete solid volume.



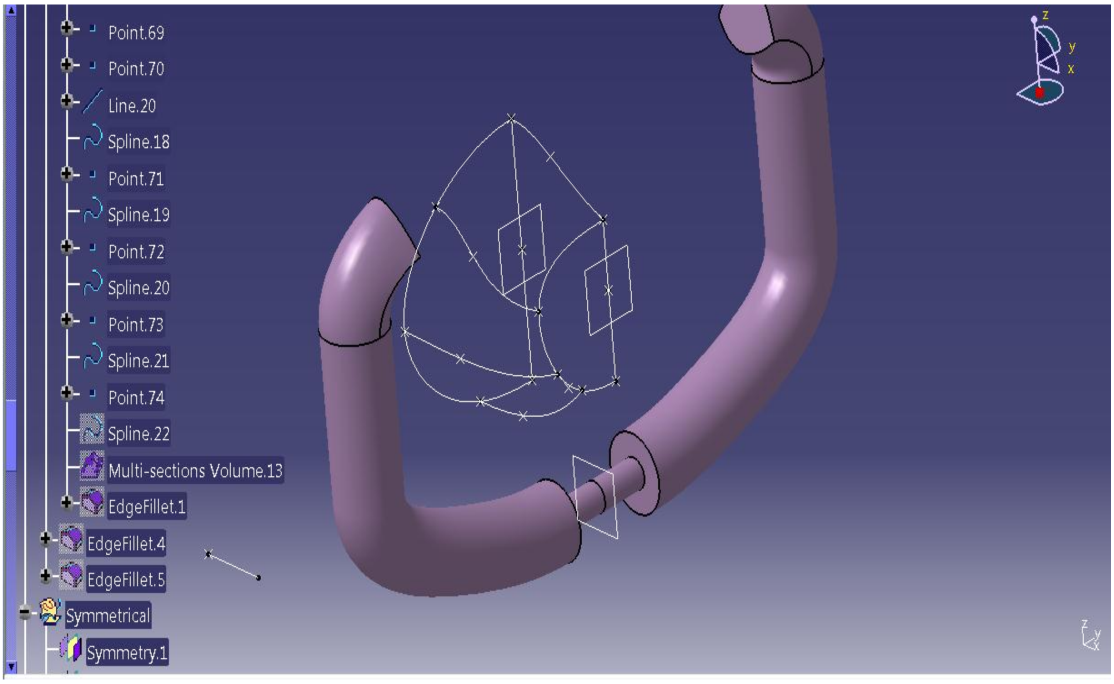
Picture 2.4.16: Driver's side air bag module 1st section wireframe.



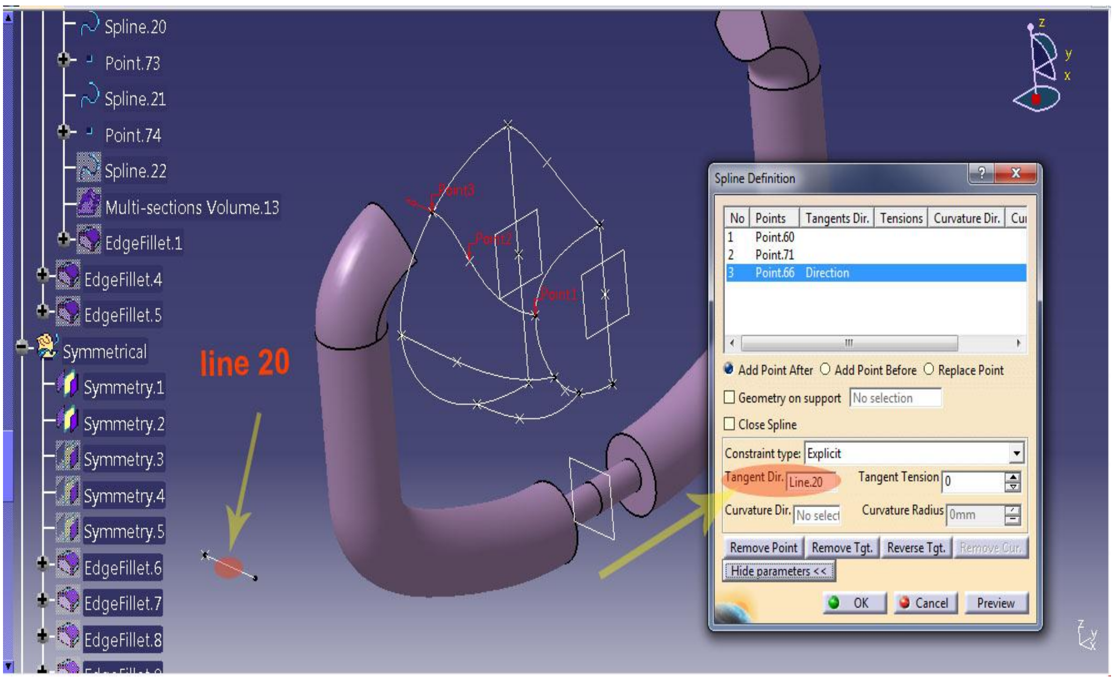
Picture 2.4.17: Driver side air bag module's 1st section wireframe with handle's volume.



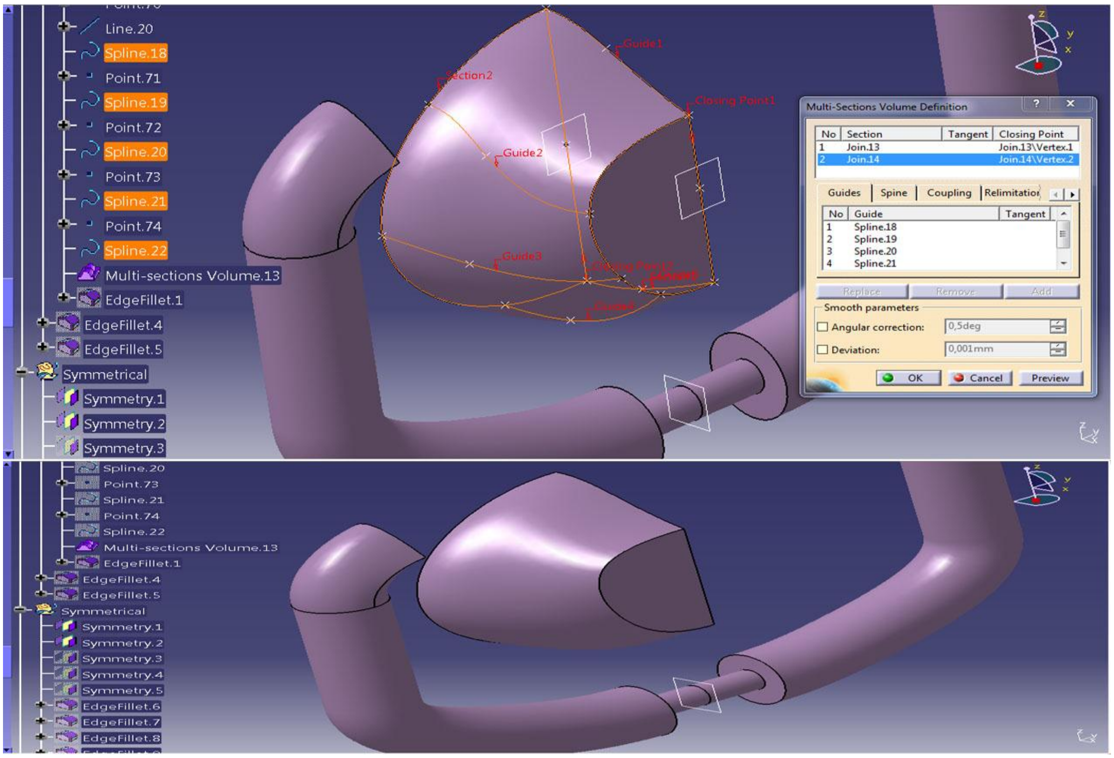
Picture 2.4.18: Distance between the driver side air bag module's two sections.



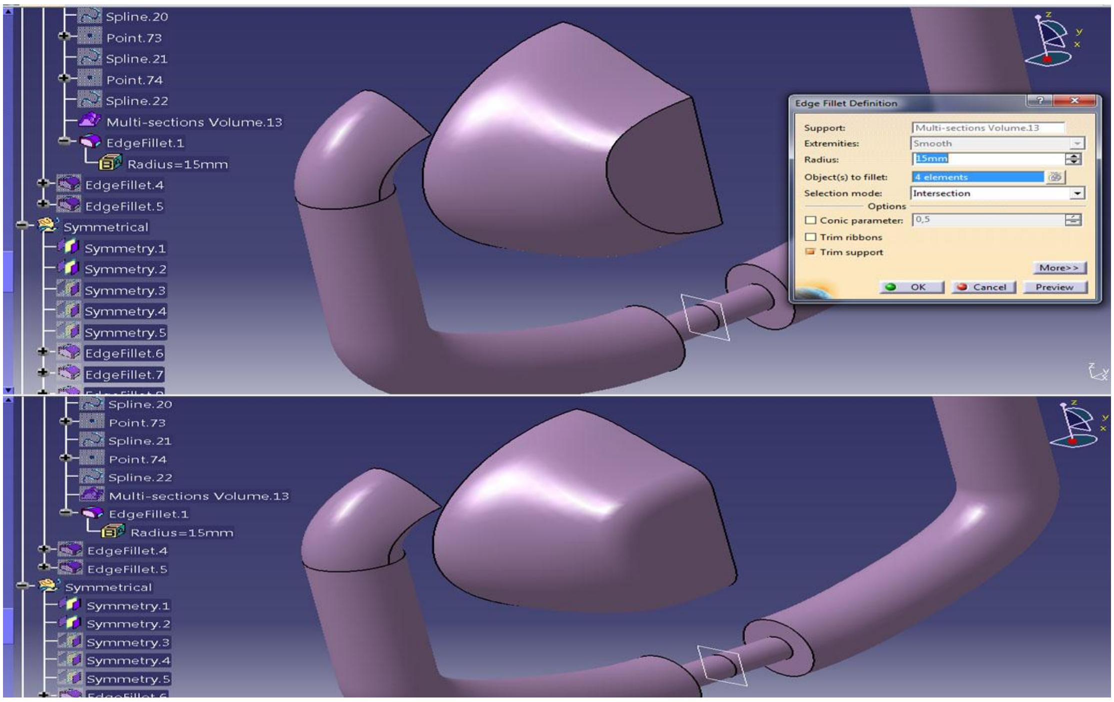
Picture 2.4.19: Guide curves for the driver's side air bag module surface creation.



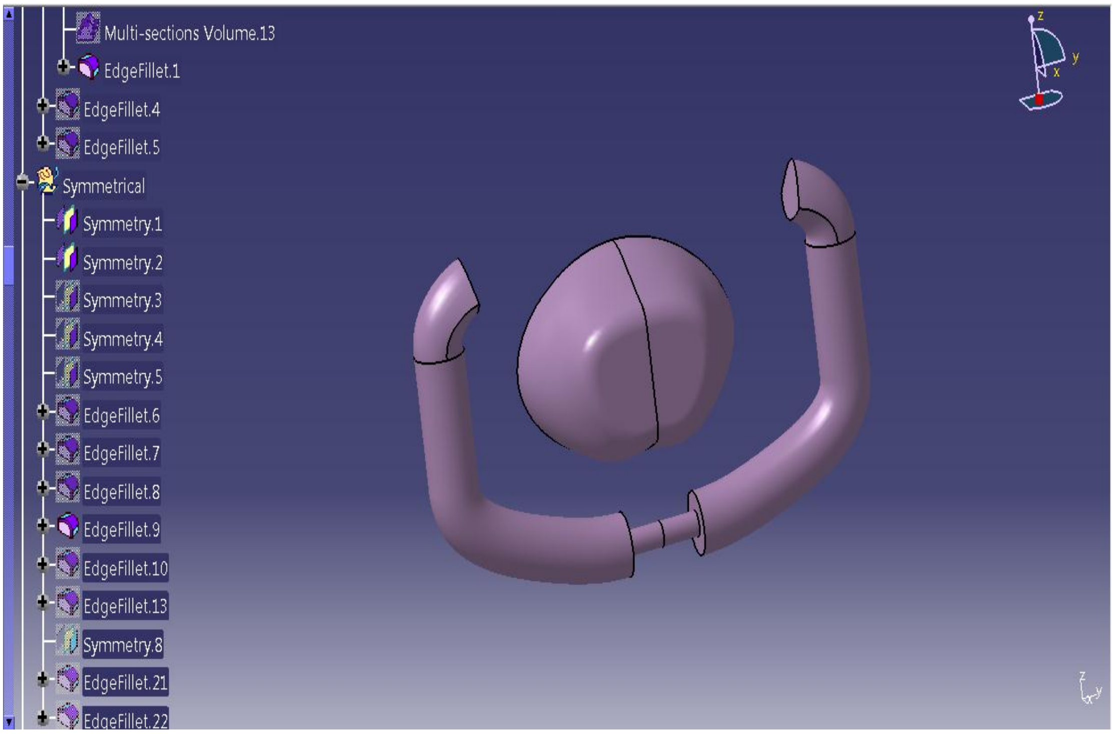
Picture 2.4.20: Guide curve definition.



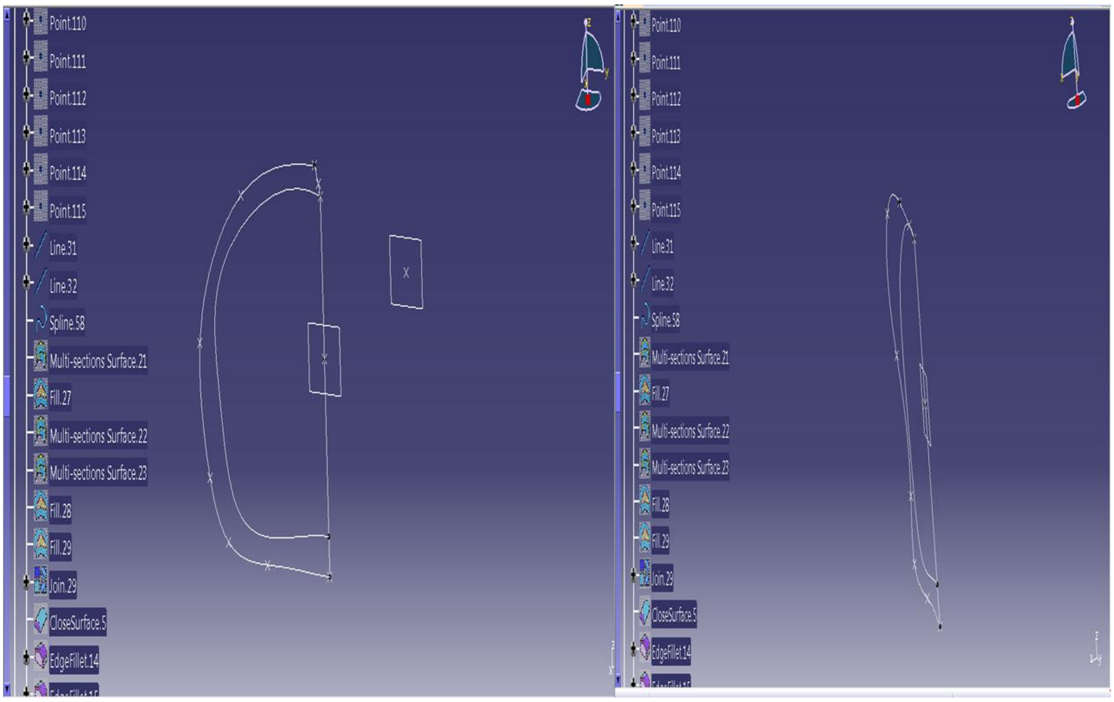
Picture 2.4.21: Solid volume of the driver's side air bag module.



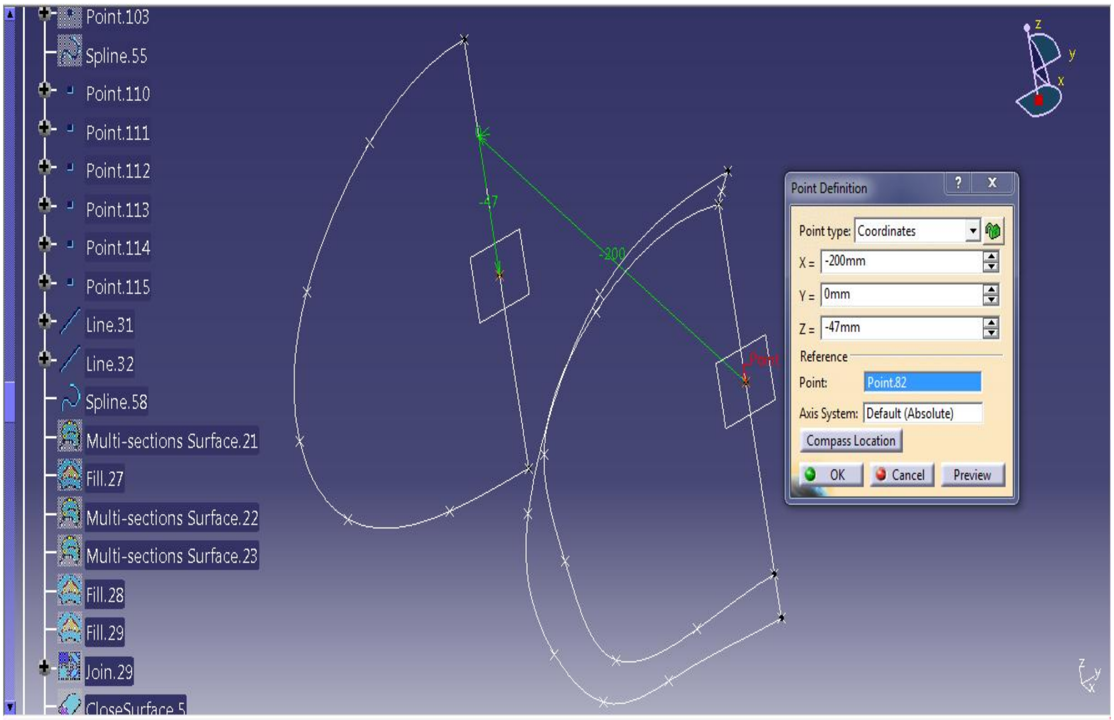
Picture 2.4.22: Driver's side air bag filleted volume.



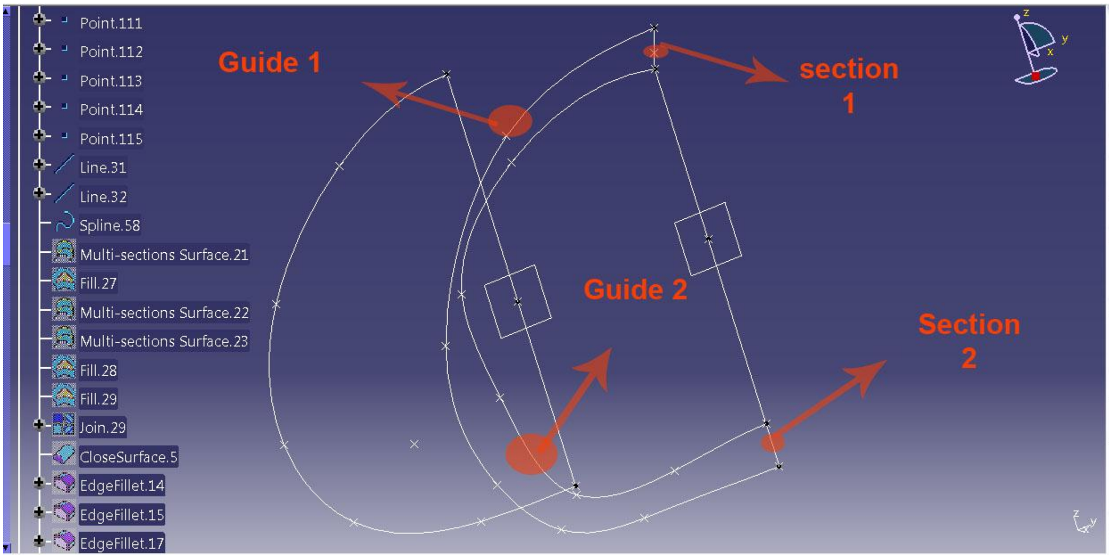
Picture 2.4.23: Driver's side air bag complete volume.



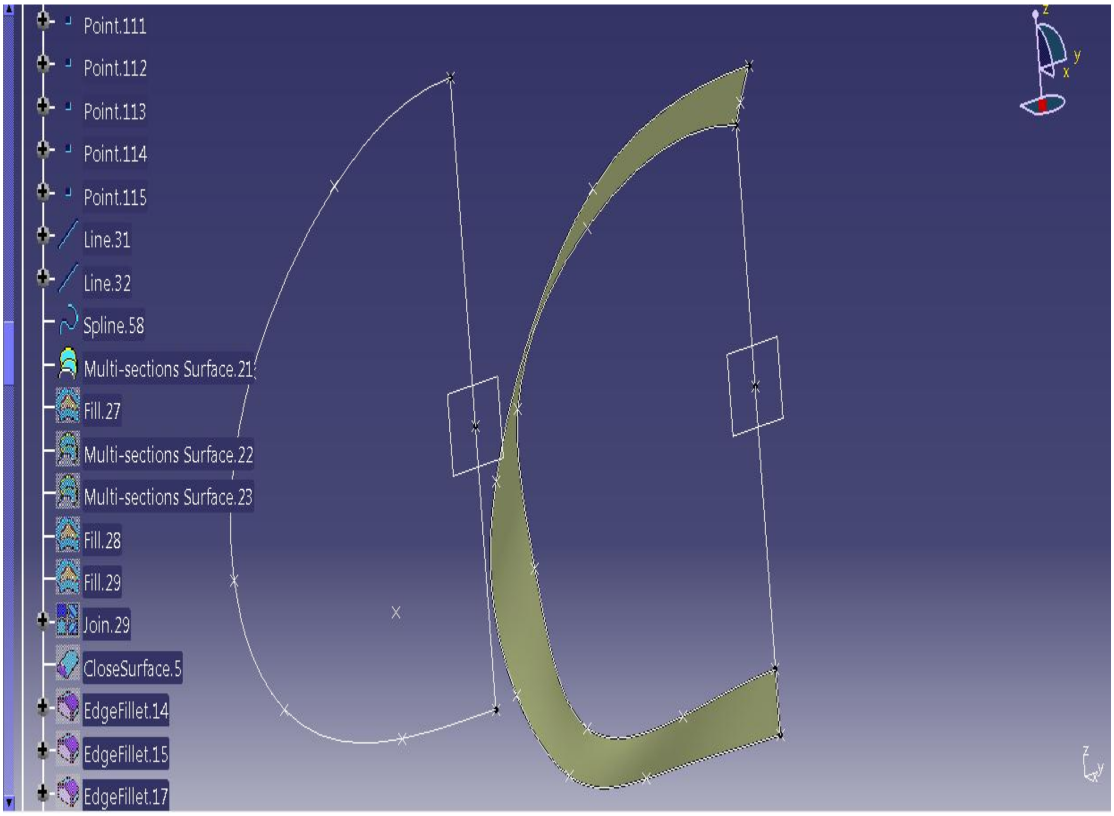
Picture 2.4.24: Steering column cover module - wireframe.



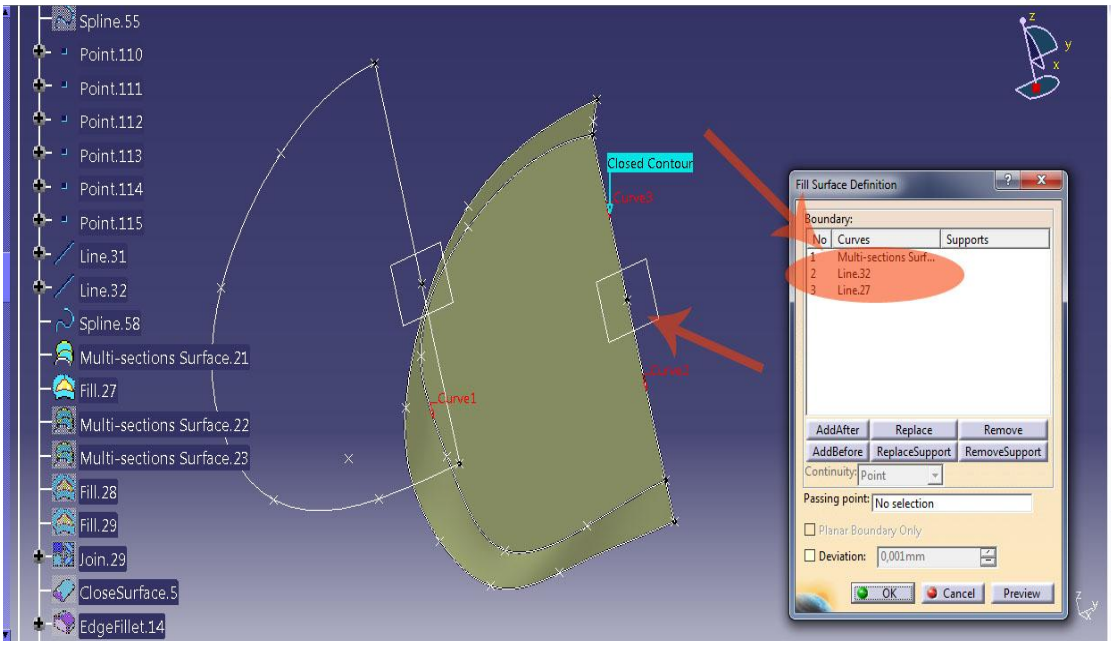
Picture 2.4.25: Steering column cover module - 2nd section.



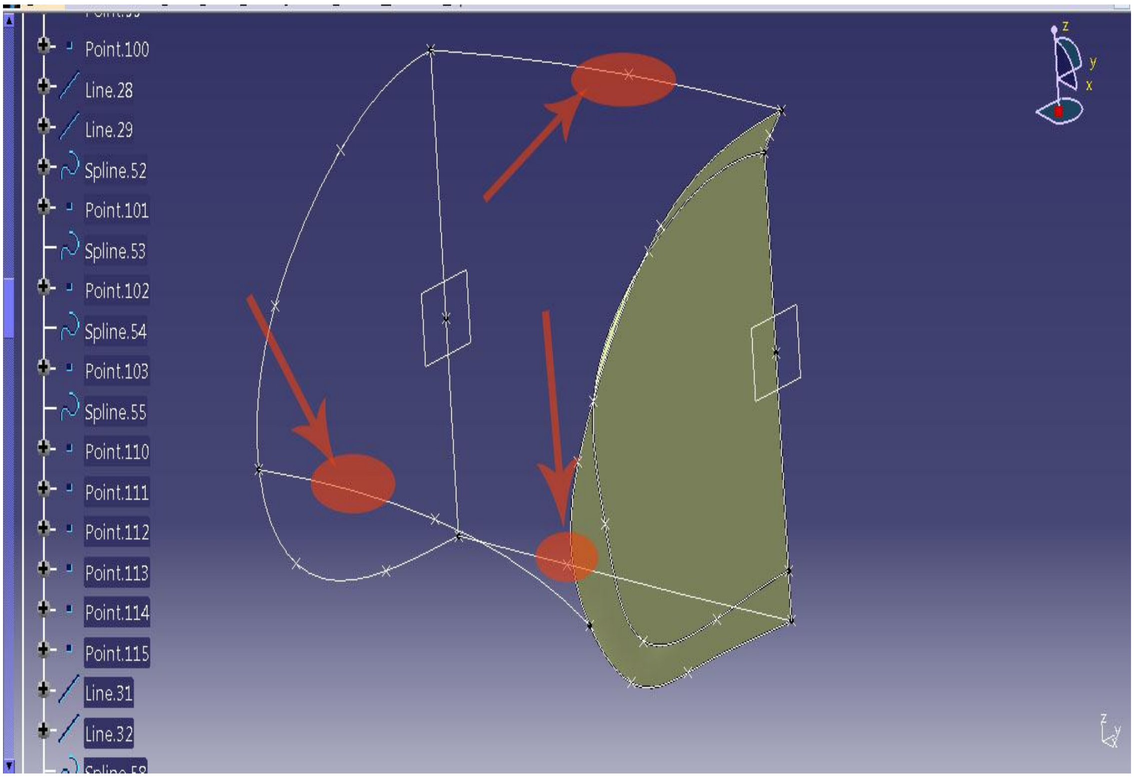
Picture 2.4.26: Sections and guide curves for the multi-section surface of section number 1 for the steering column cover.



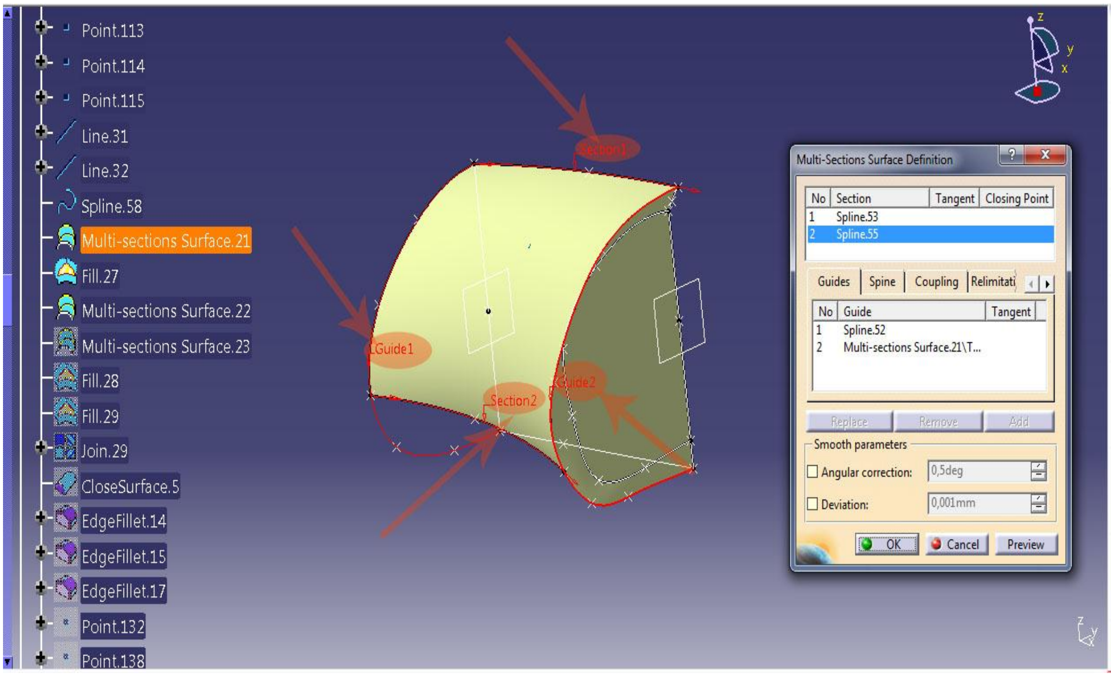
Picture 2.4.27: Multi section surface of section number 1.



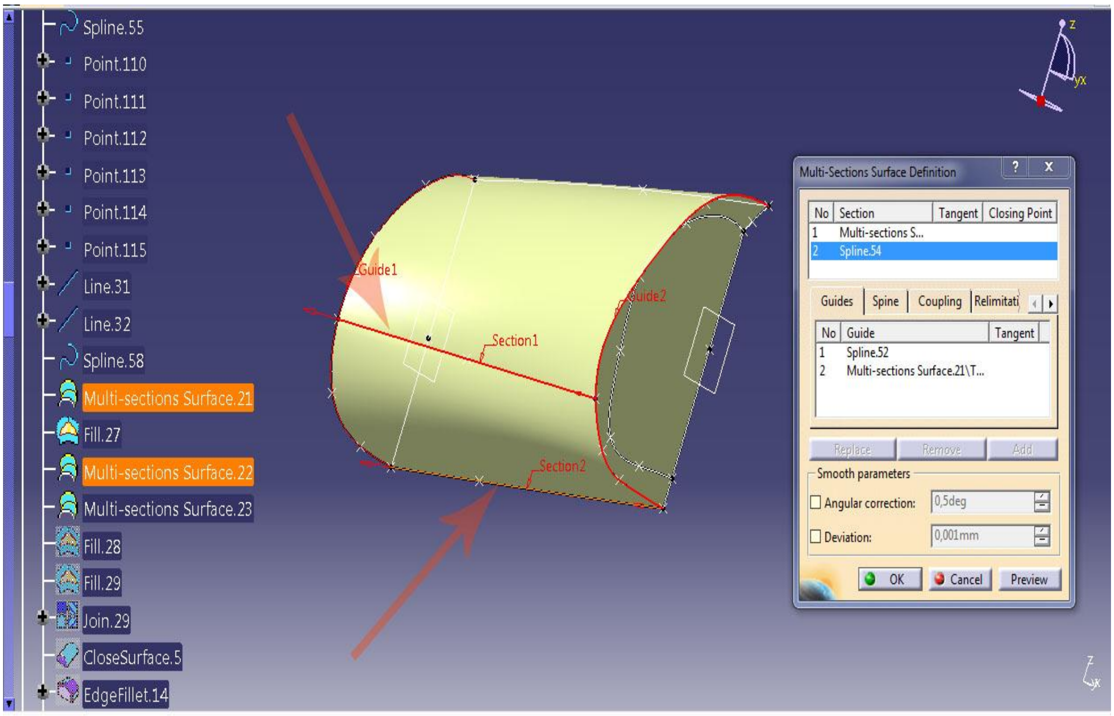
Picture 2.4.28: Filled inner surface of section number 1.



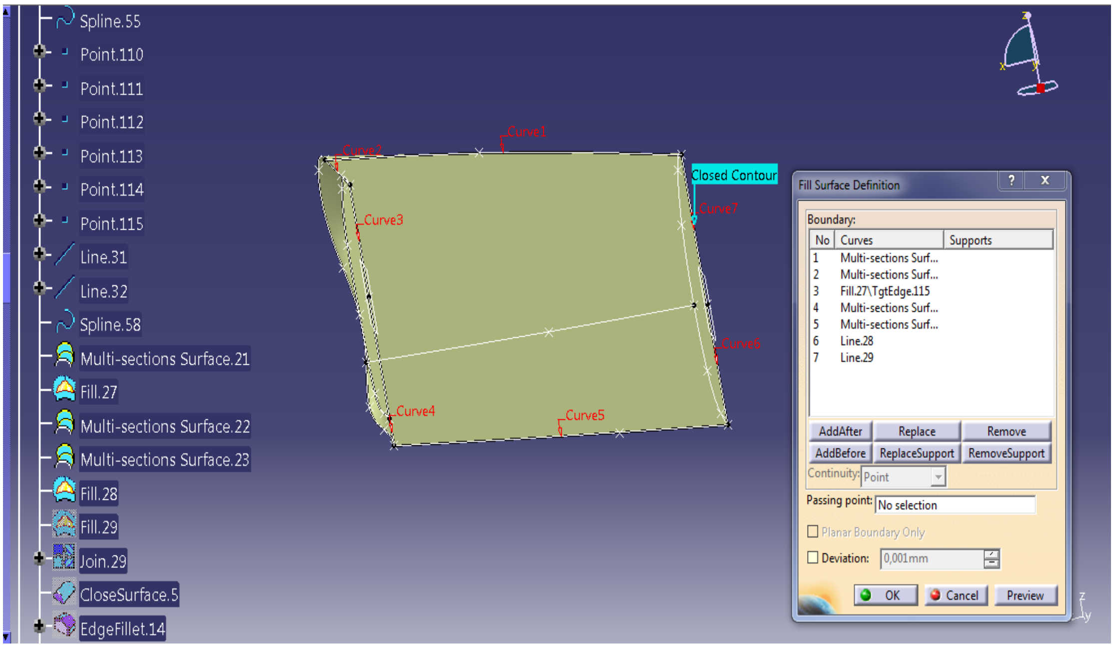
Picture 2.4.29: Generative curves for the peripheral surfaces of steering's column cover.



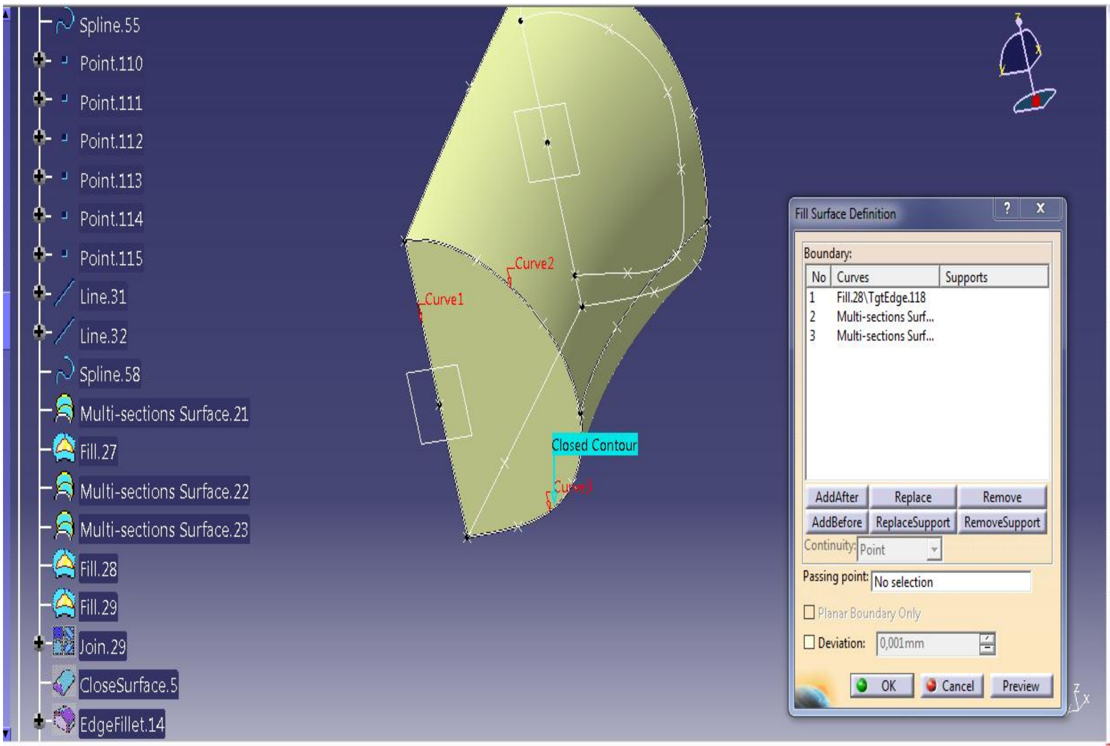
Picture 2.4.30: First peripheral multi section surface of steering's column cover.



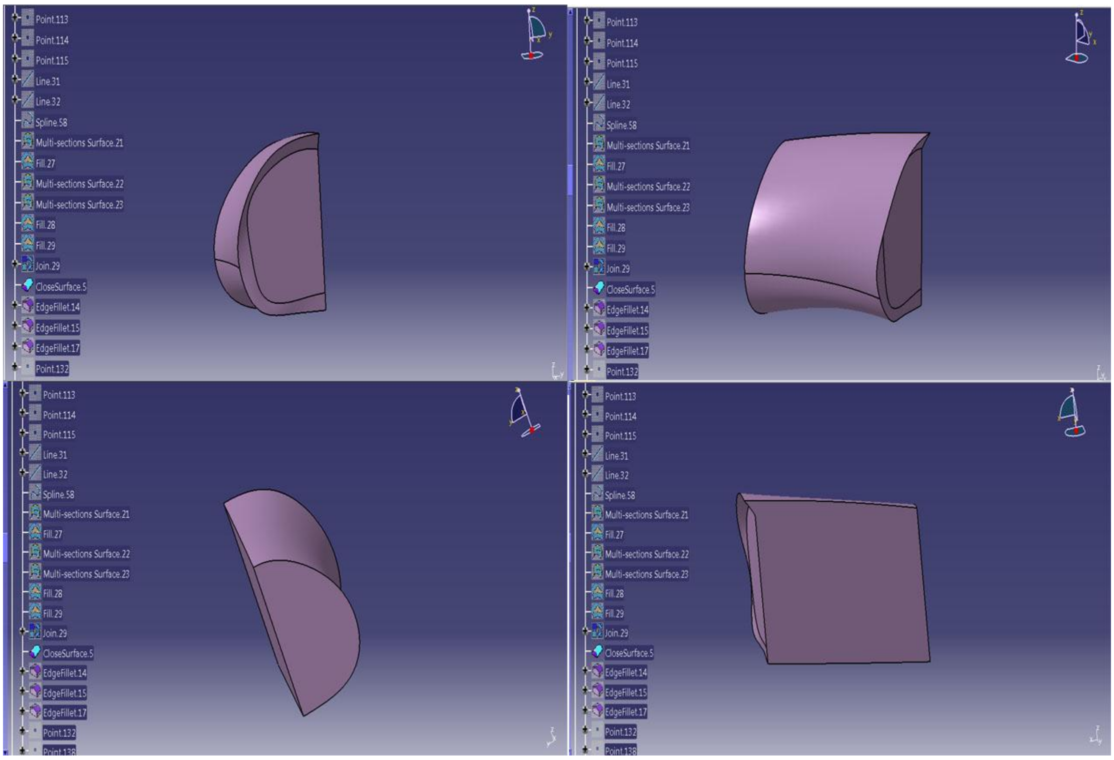
Picture 2.4.31: Second peripheral multi section surface of steering's column cover.



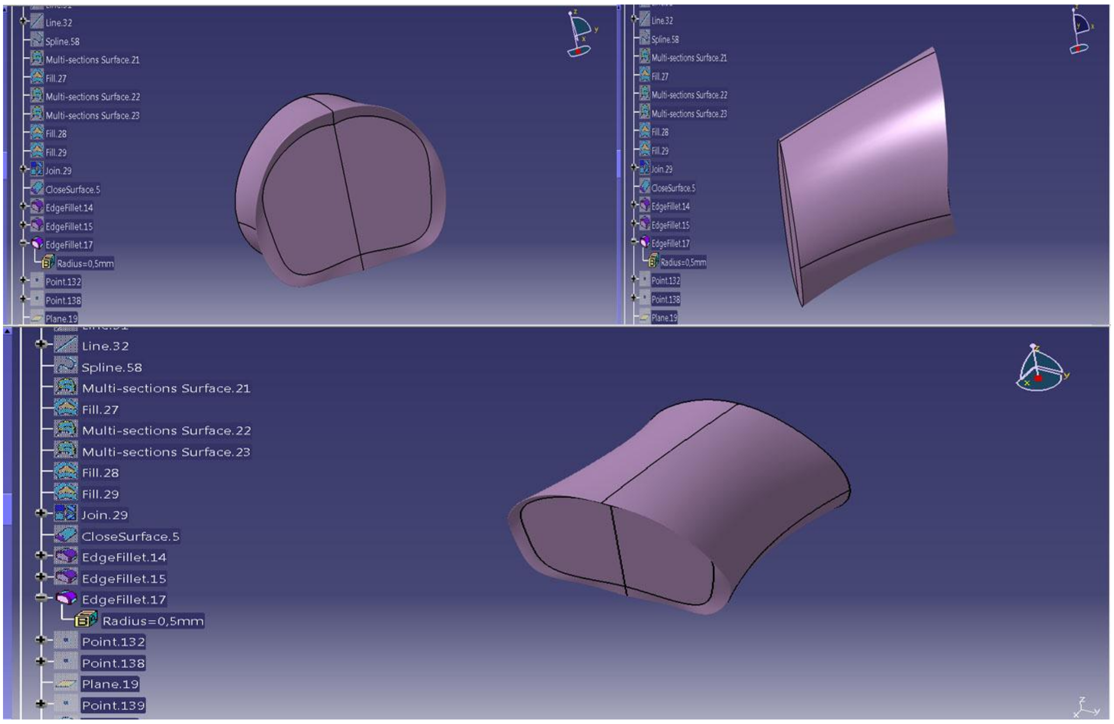
Picture 2.4.32: Lateral surface creation of steering's column cover.



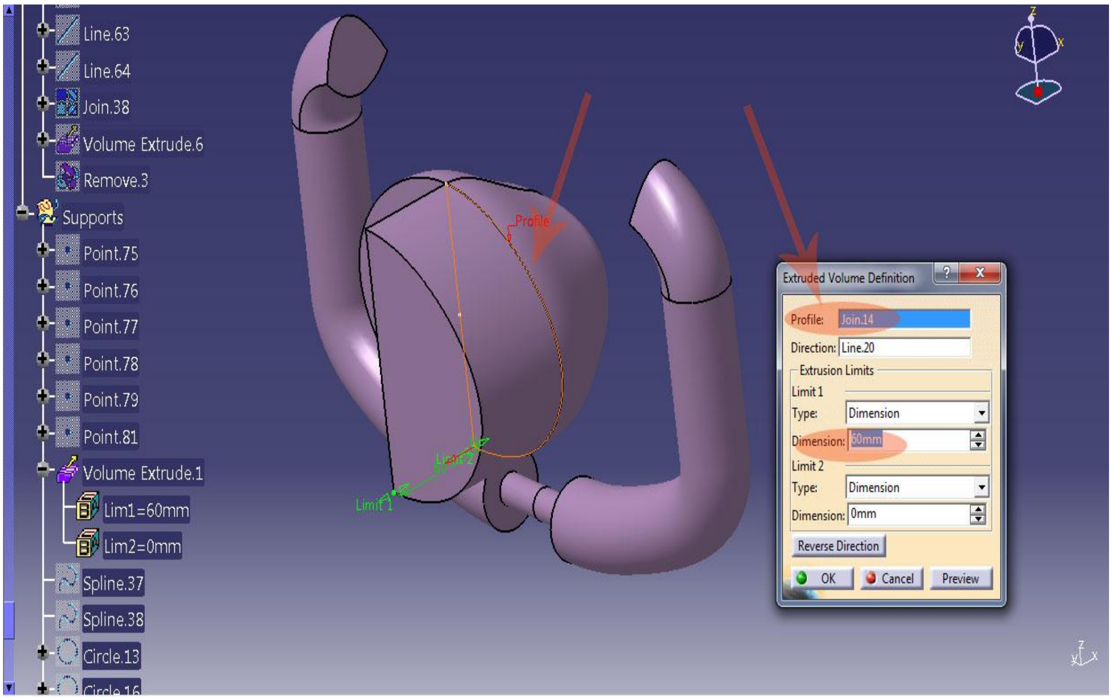
Picture 2.4.33: Steering's column cover second section's fill surface.



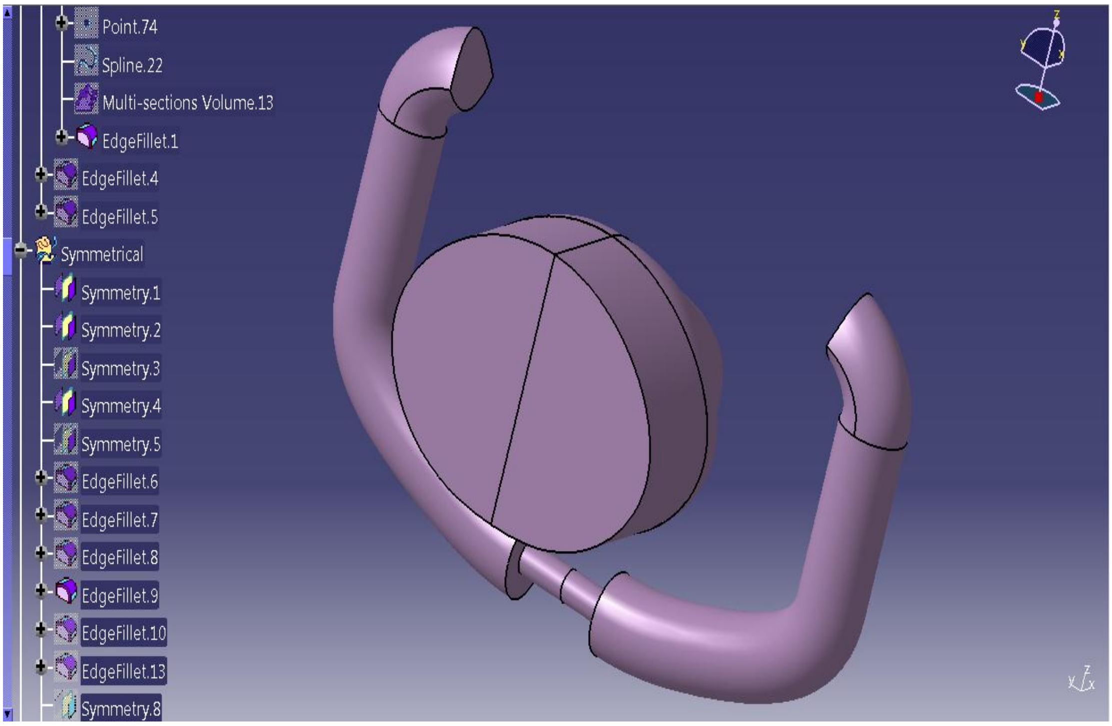
Picture 2.4.34: Creation of steering's column cover volume.



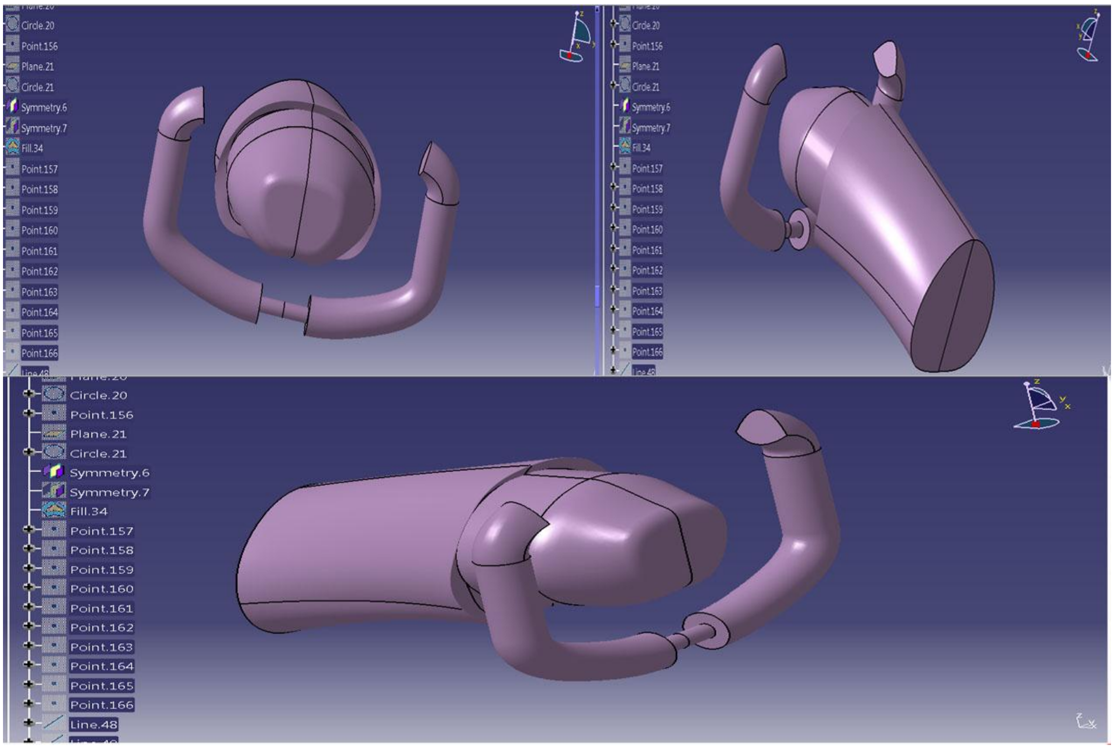
Picture 2.4.35: Steering column cover complete volume.



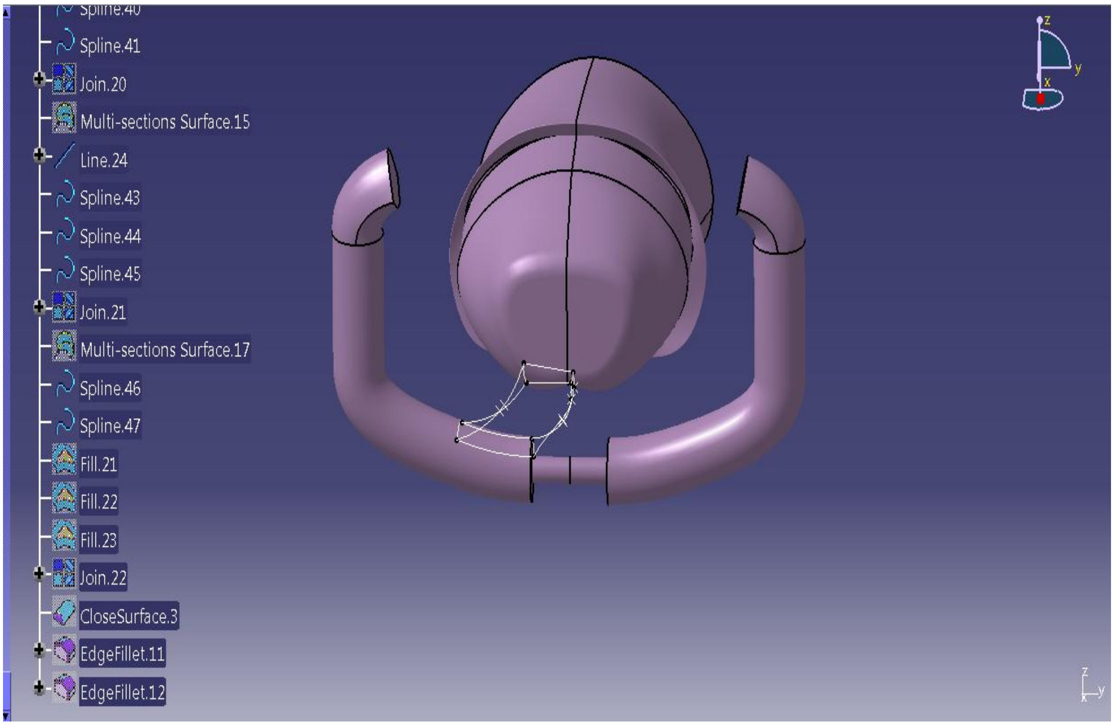
Picture 2.4.36: Creation of steering's wheel hub volume.



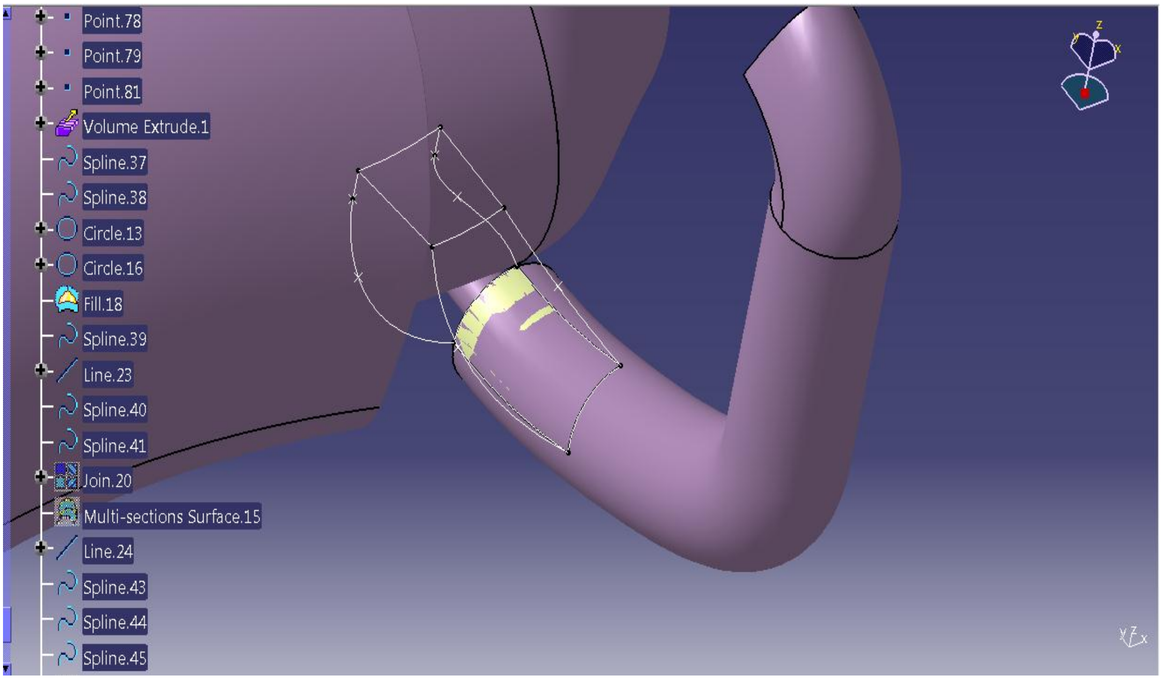
Picture 2.4.37: Complete steering wheel hub volume.



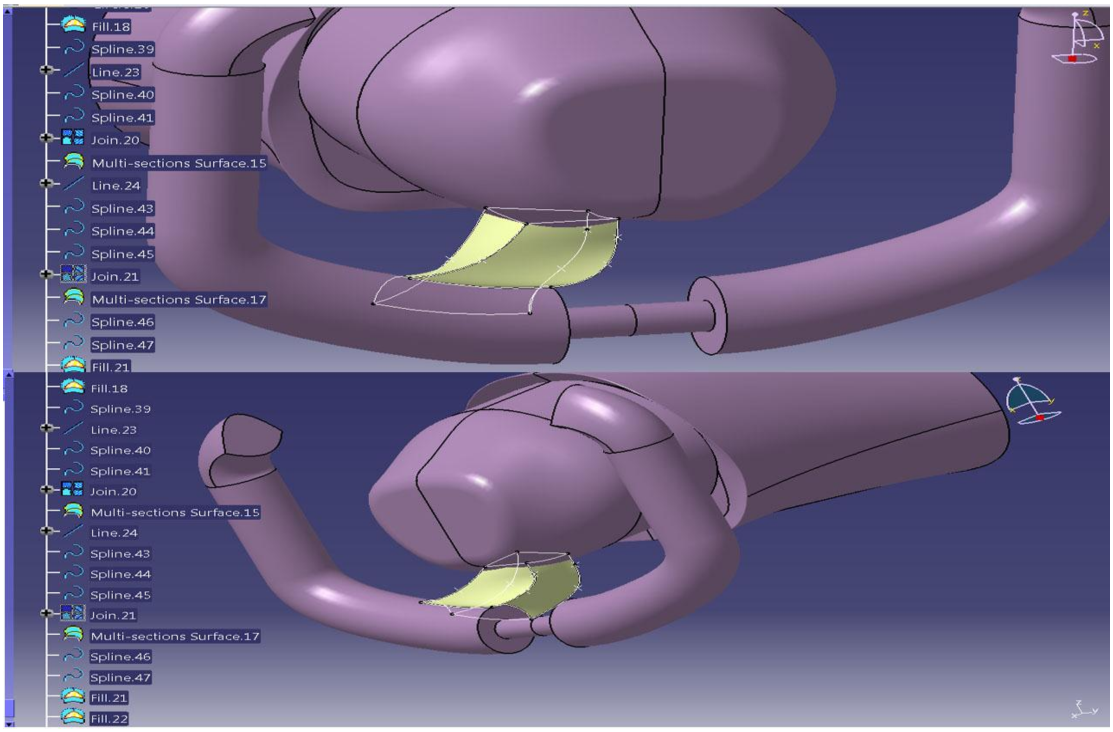
Picture 2.4.38: The three different volumes defining the part of the steering wheel.



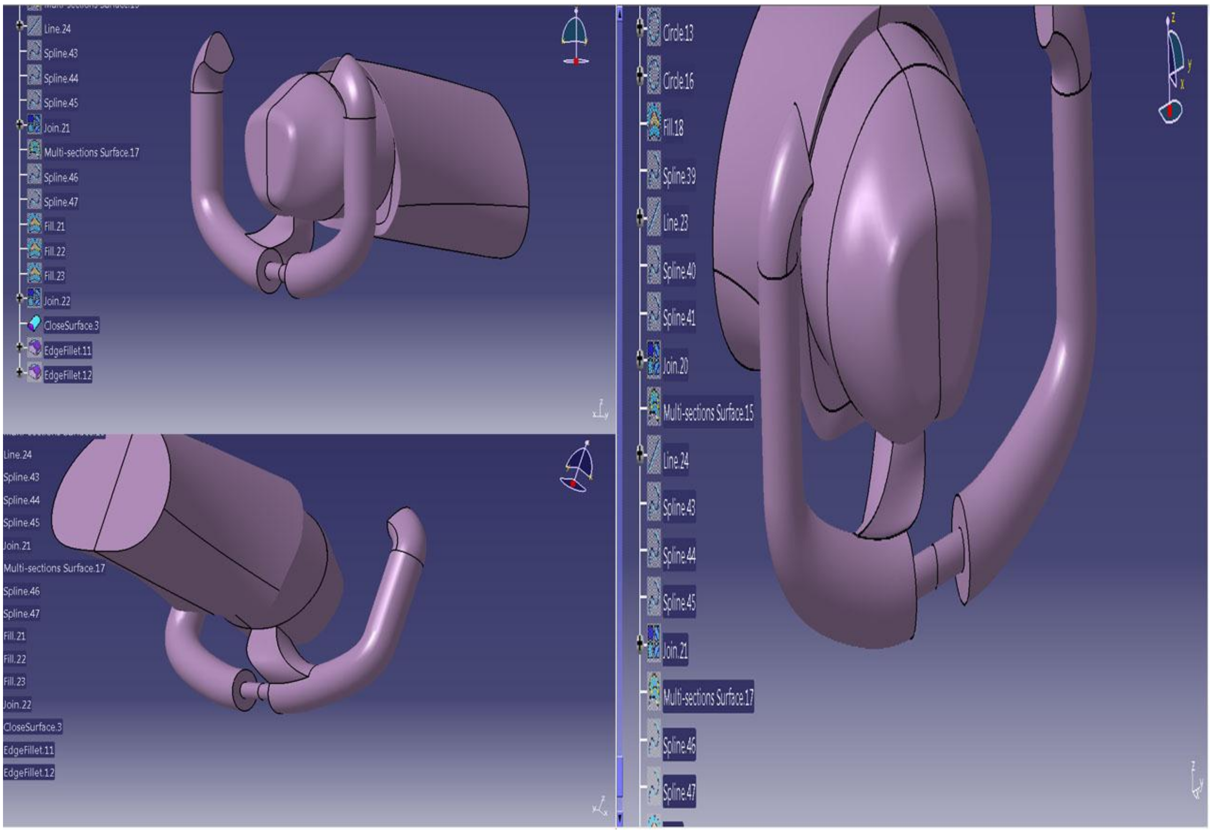
Picture 2.4.39: Steering wheel pillar wireframe.



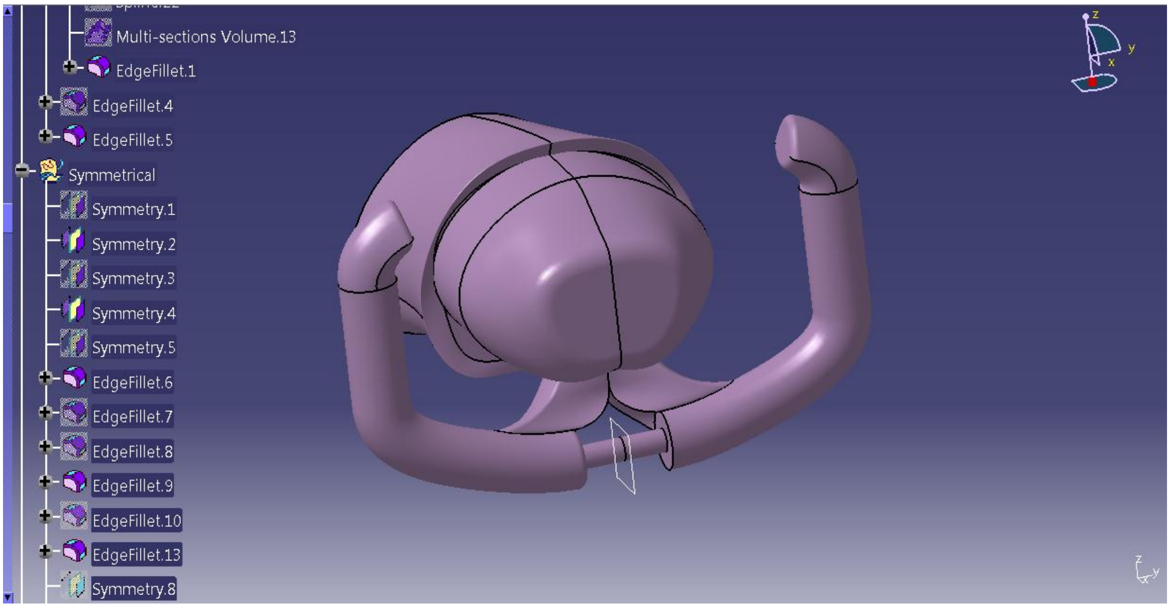
Picture 2.4.40: Adjacent to handle complete of steering wheel pillar.



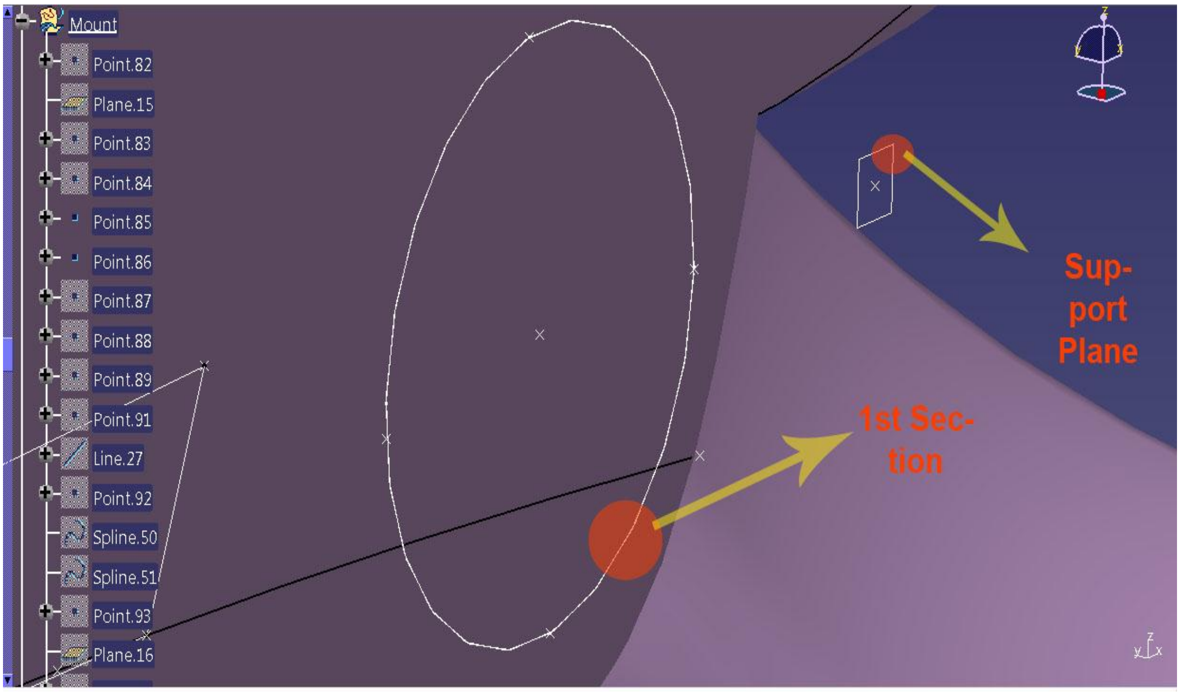
Picture 2.4.41: Regional surfaces of steering wheel pillar.



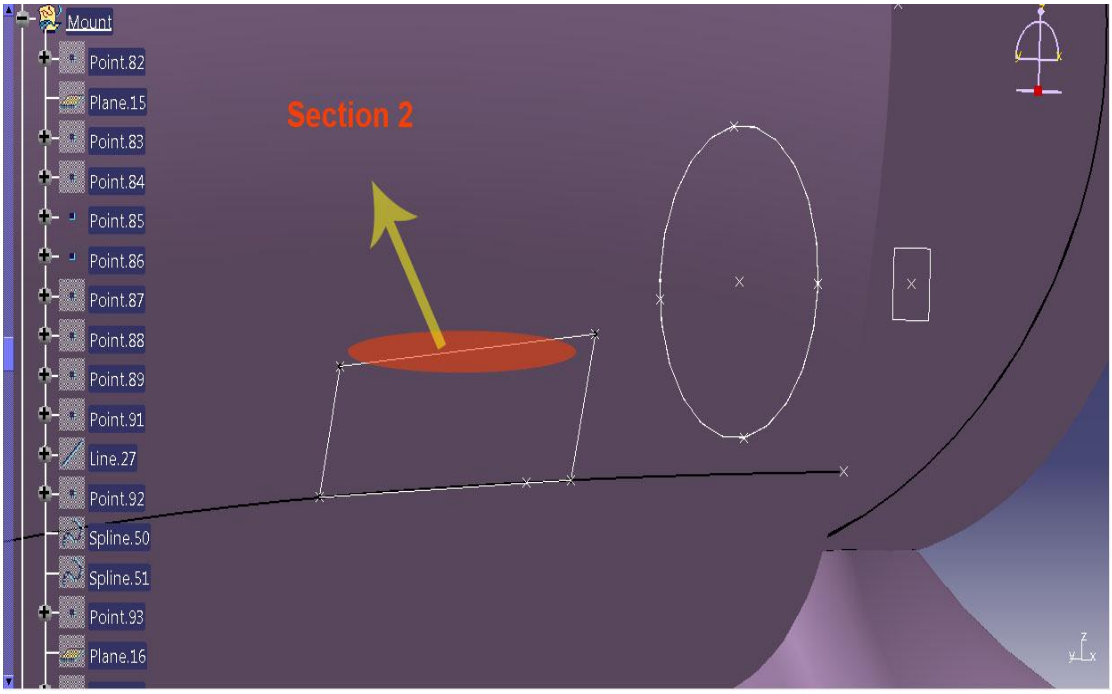
Picture 2.4.42: Steering wheel pillar solid volume.



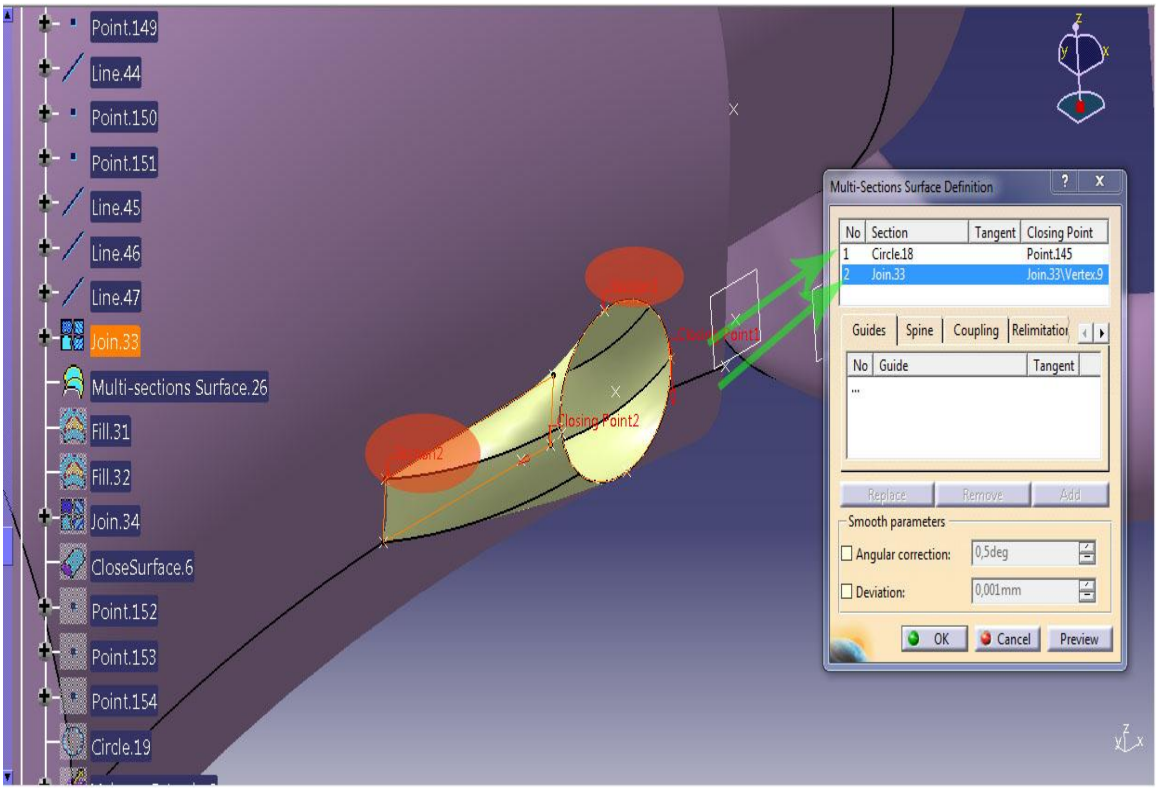
Picture 2.4.43: Filleting of steering's wheel pillar volumes.



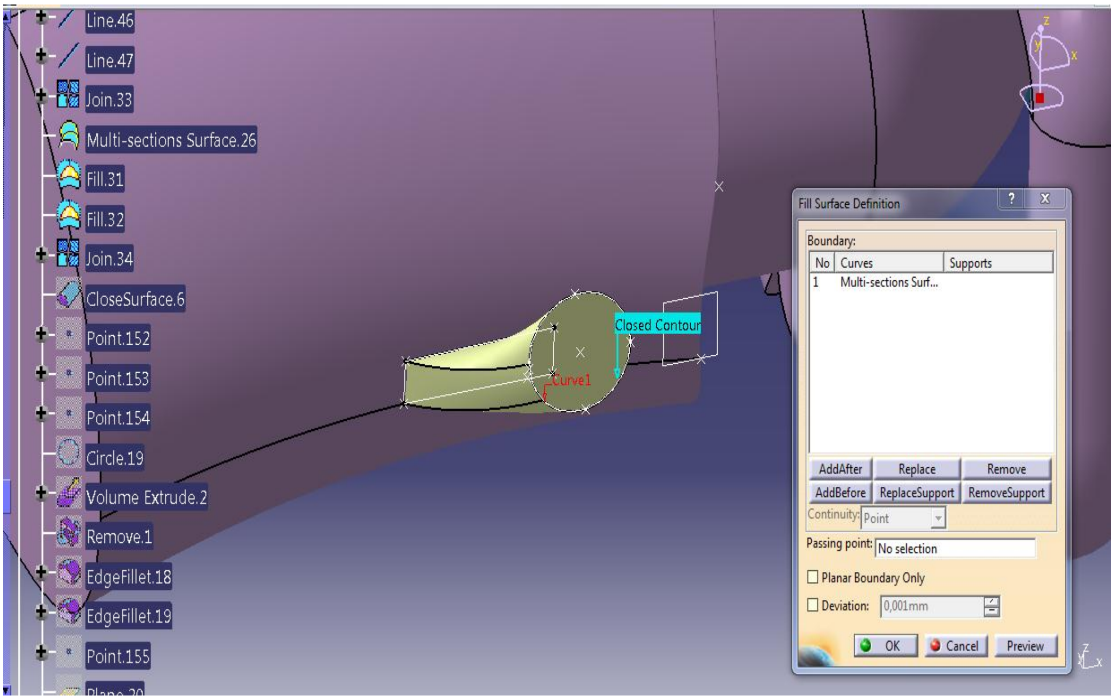
Picture 2.4.44: Circle wireframe – 1st wireframe.



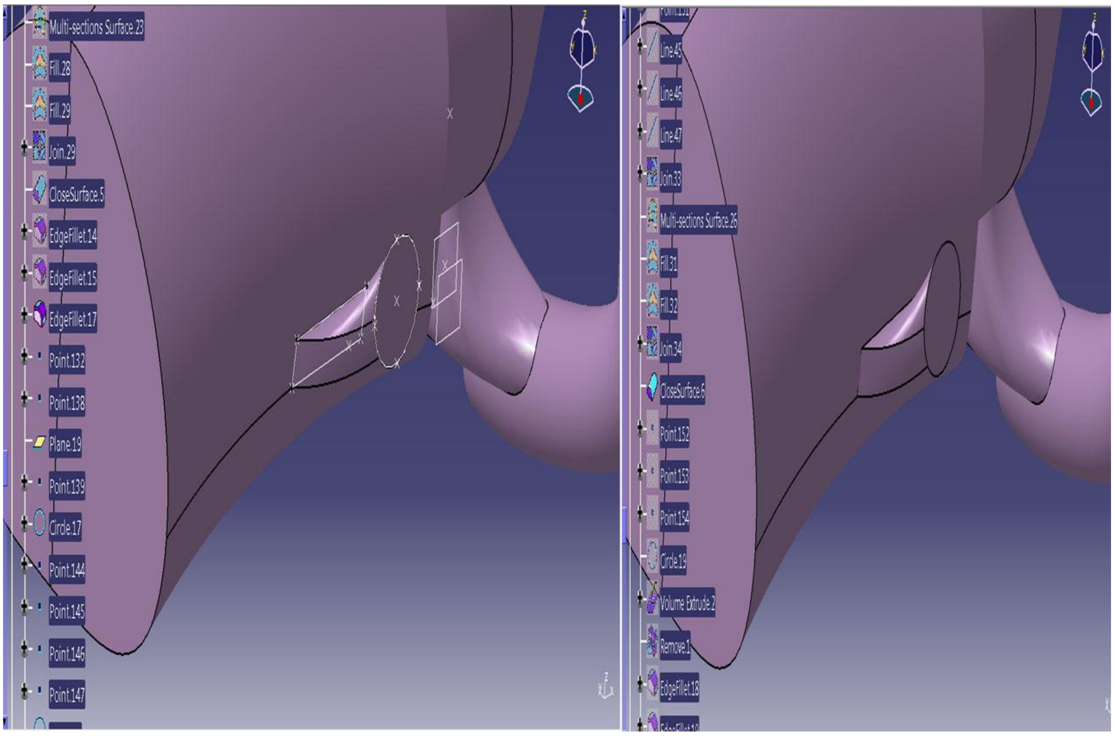
Picture 2.4.45: Indicator's lights attachment wireframe second section.



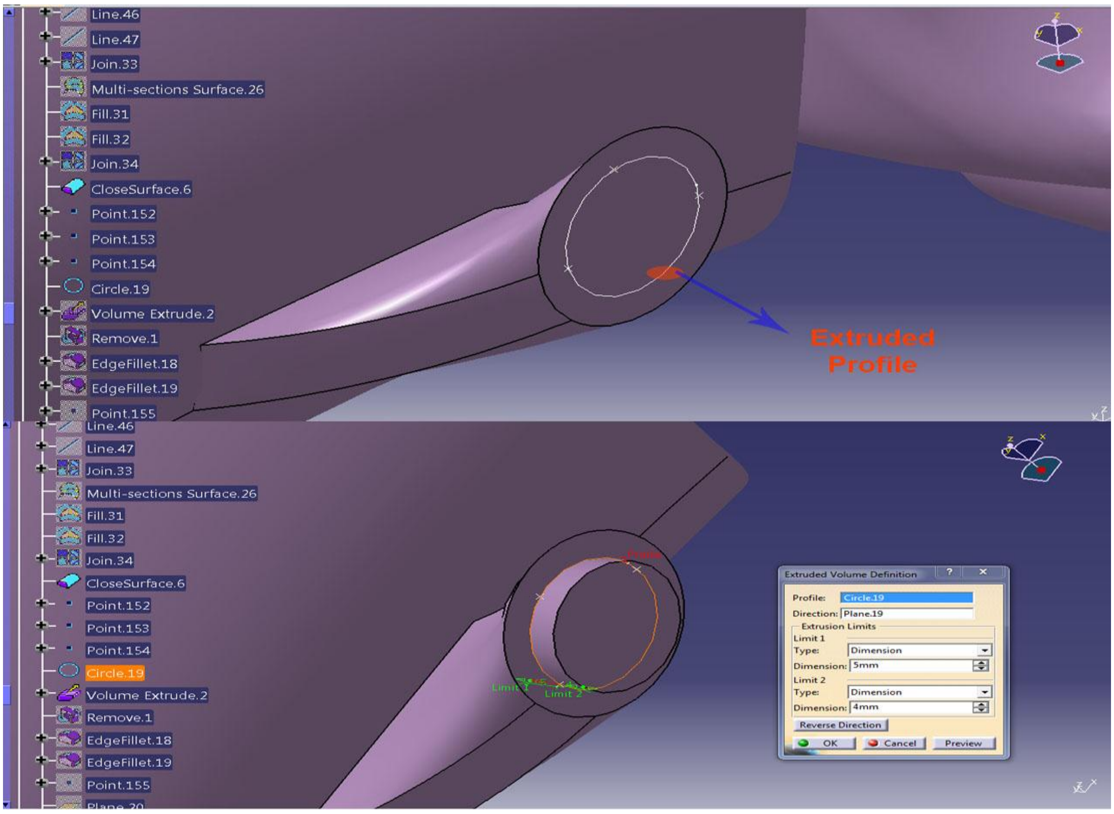
Picture 2.4.46: Indicator's lights attachment peripheral surfaces.



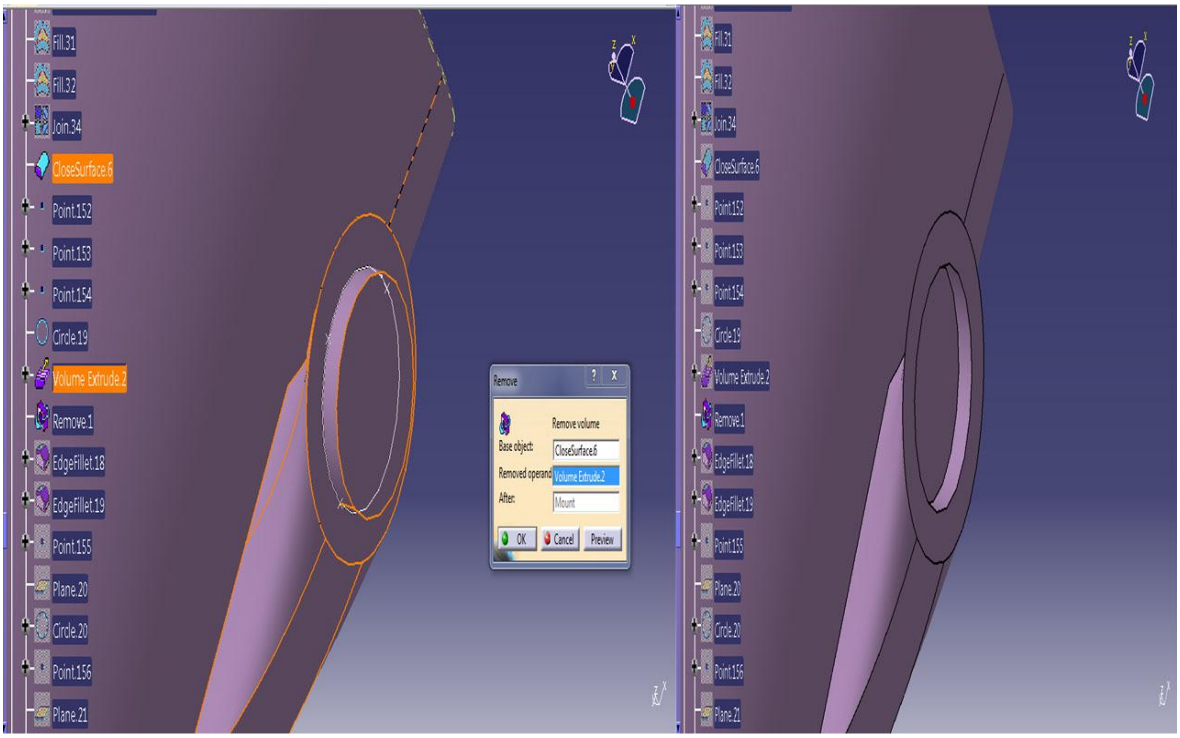
Picture 2.4.47: Complete set of peripheral surfaces.



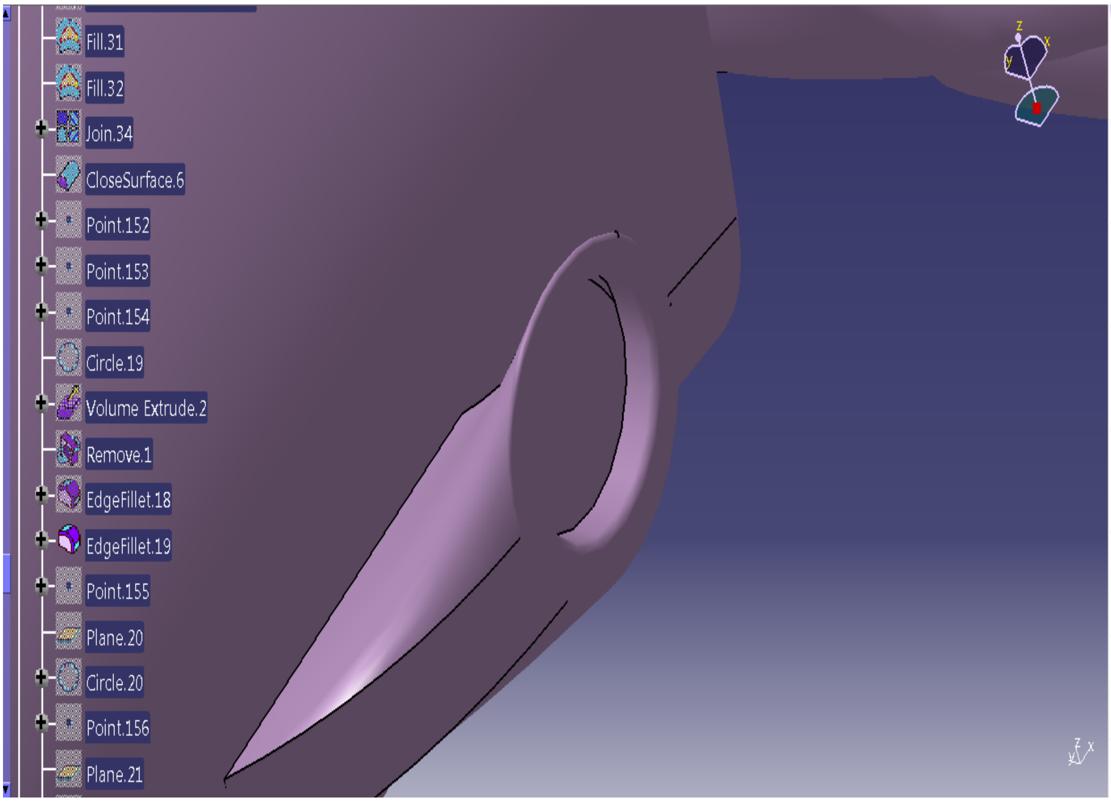
Picture 2.4.48: Creation of indicator's lights attachment volume.



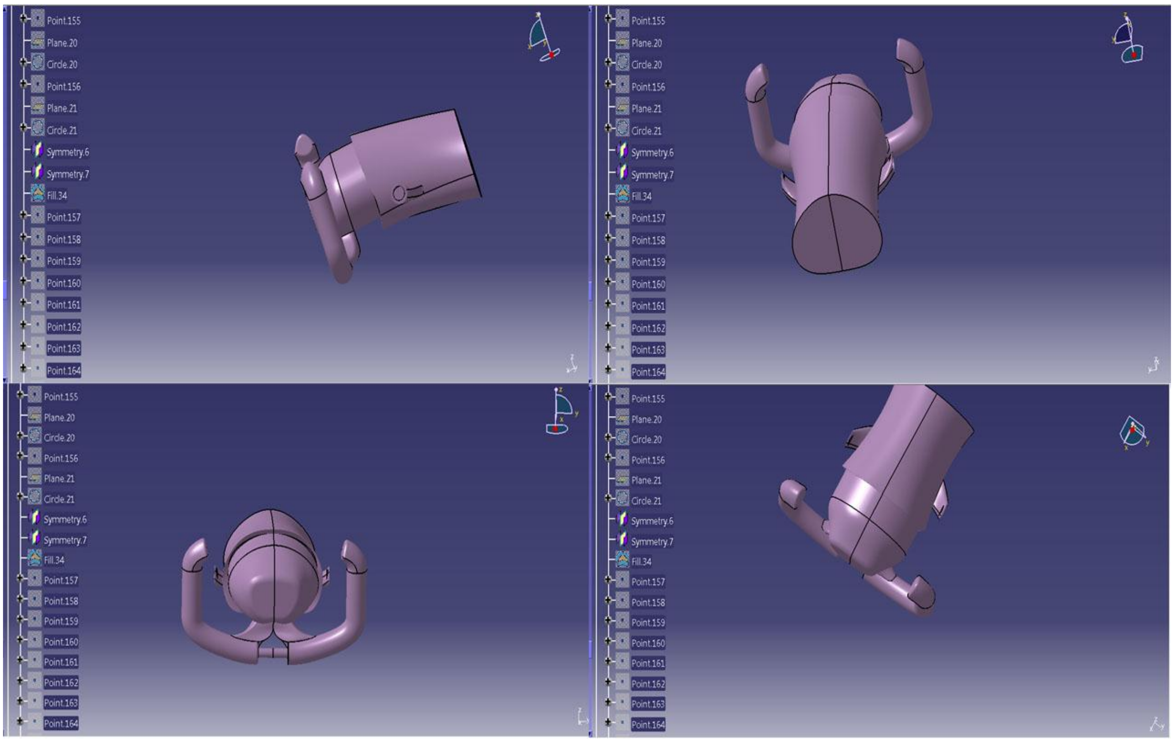
Picture 2.4.49: Creation of auxiliary extrusion volume.



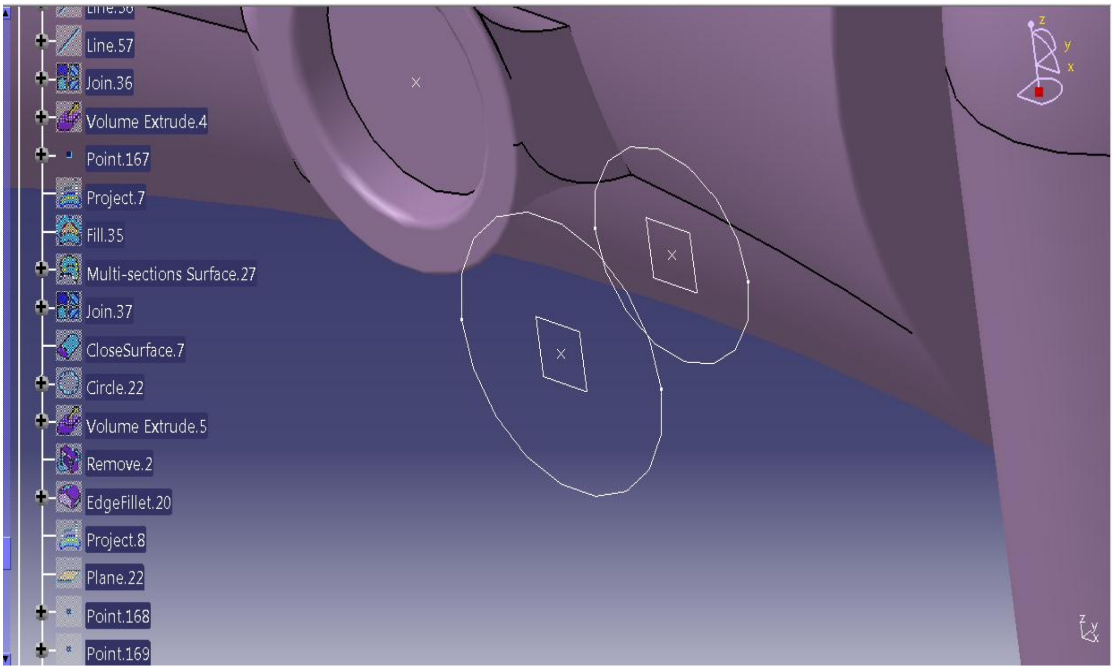
Picture 2.4.50: Resulted volume from remove operation.



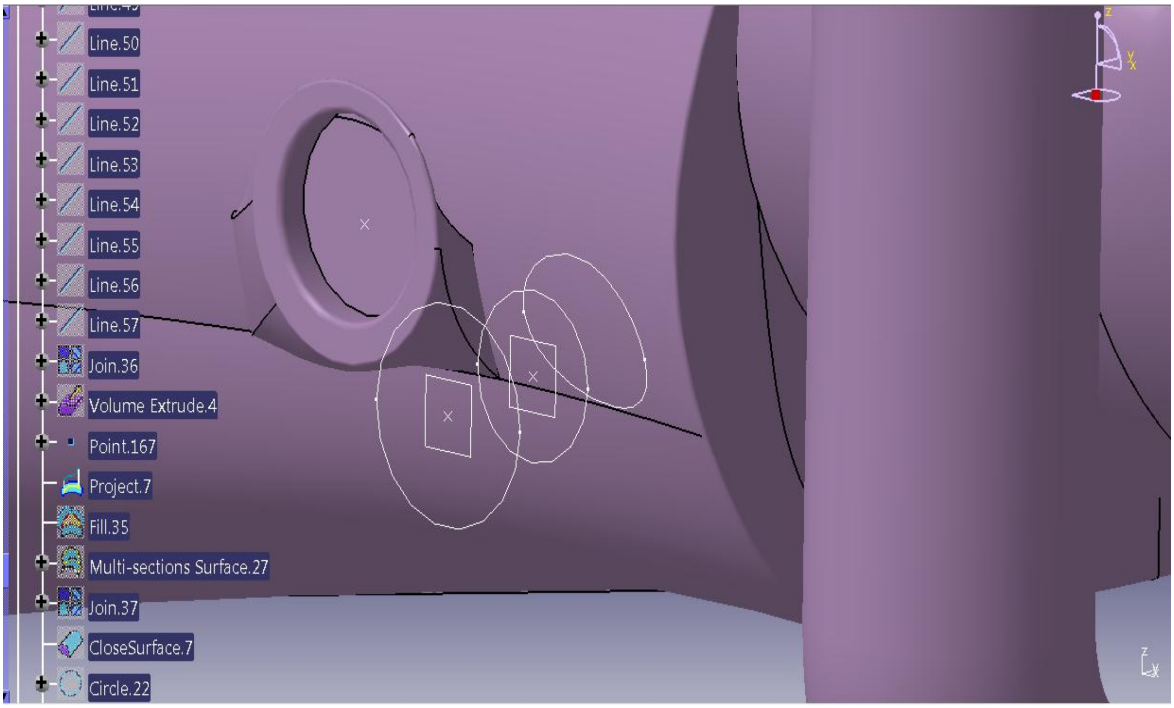
Picture 2.4.51: Final attached volume.



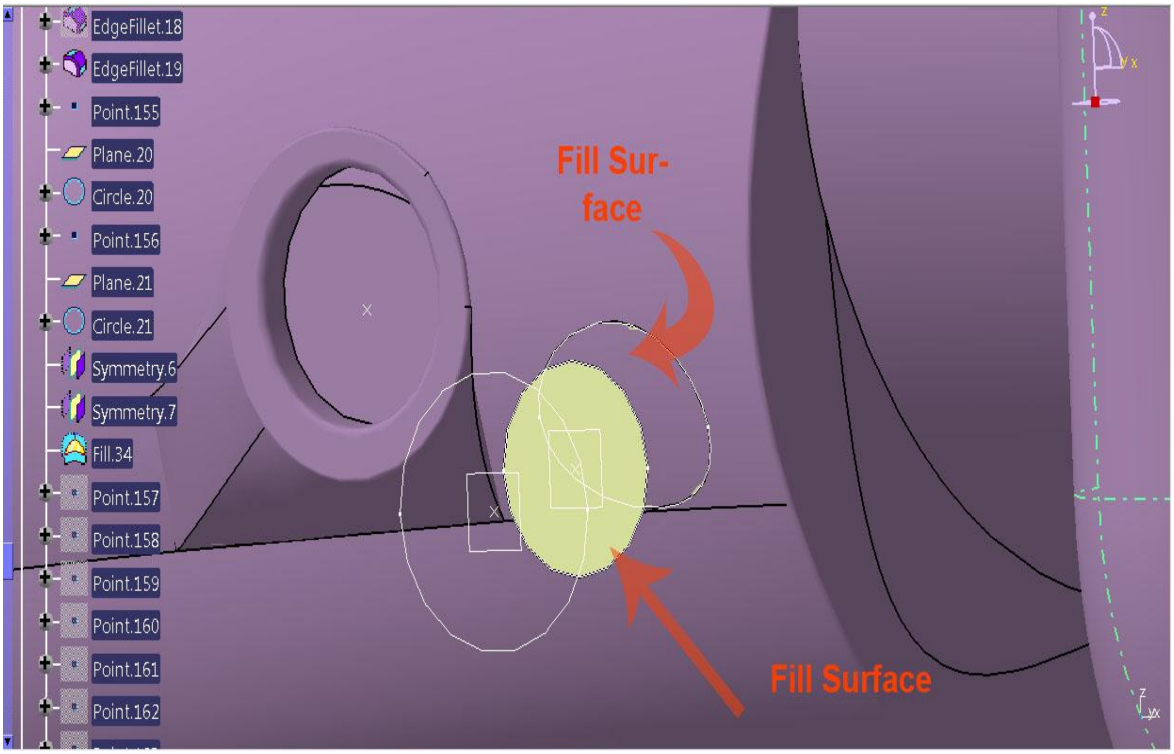
Picture 2.4.52: Steering wheel volume with integrated indicator lights lever socket volumes.



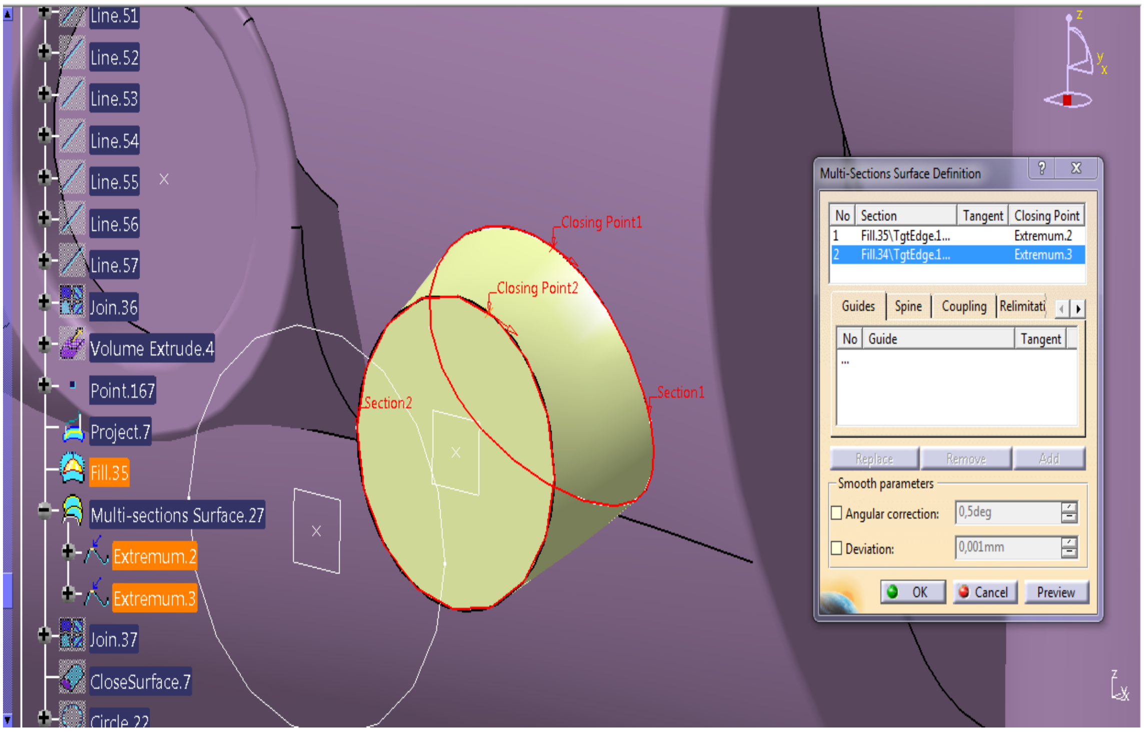
Picture 2.4.53: Ignition’s solid volume wireframe.



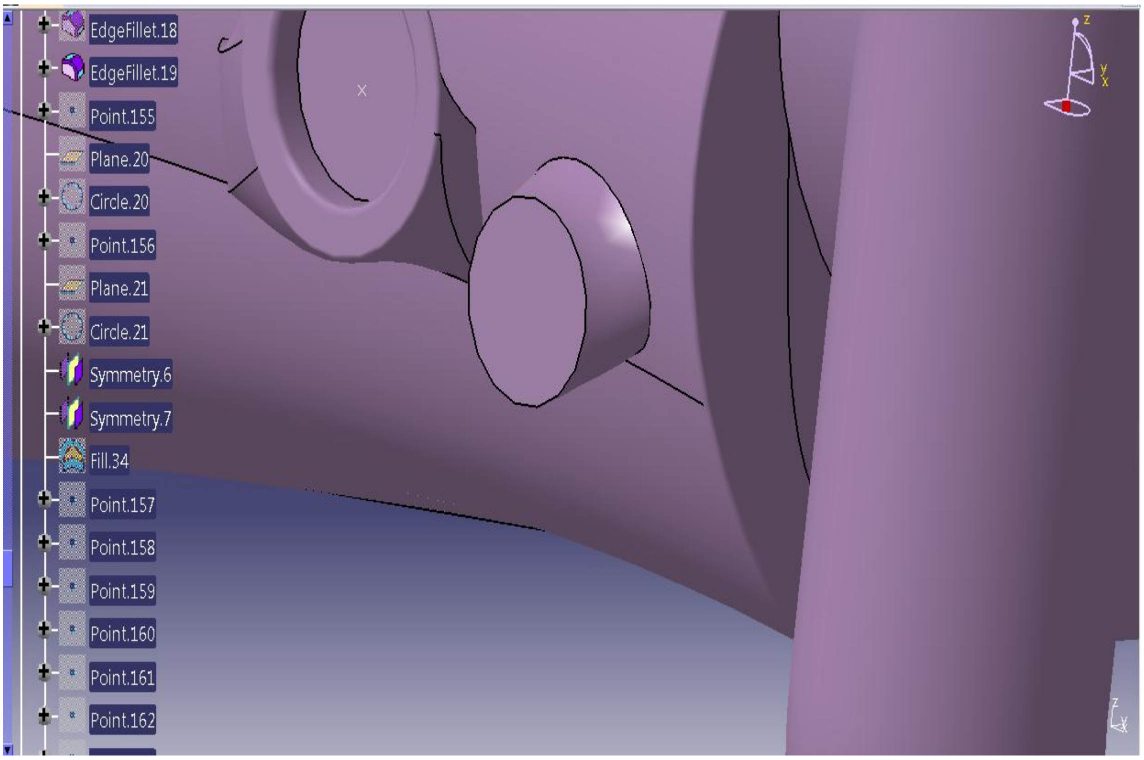
Picture 2.4.54: Ignition's section wireframe – circle projection.



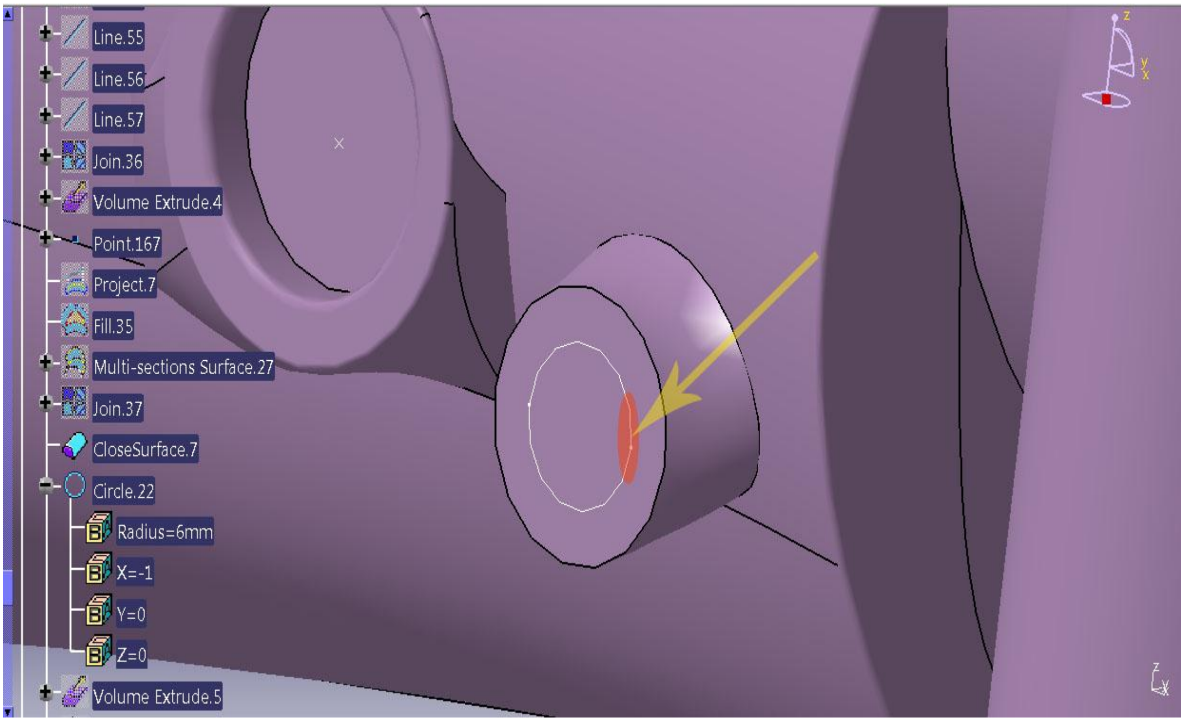
Picture 2.4.55: Ignition's fill surfaces.



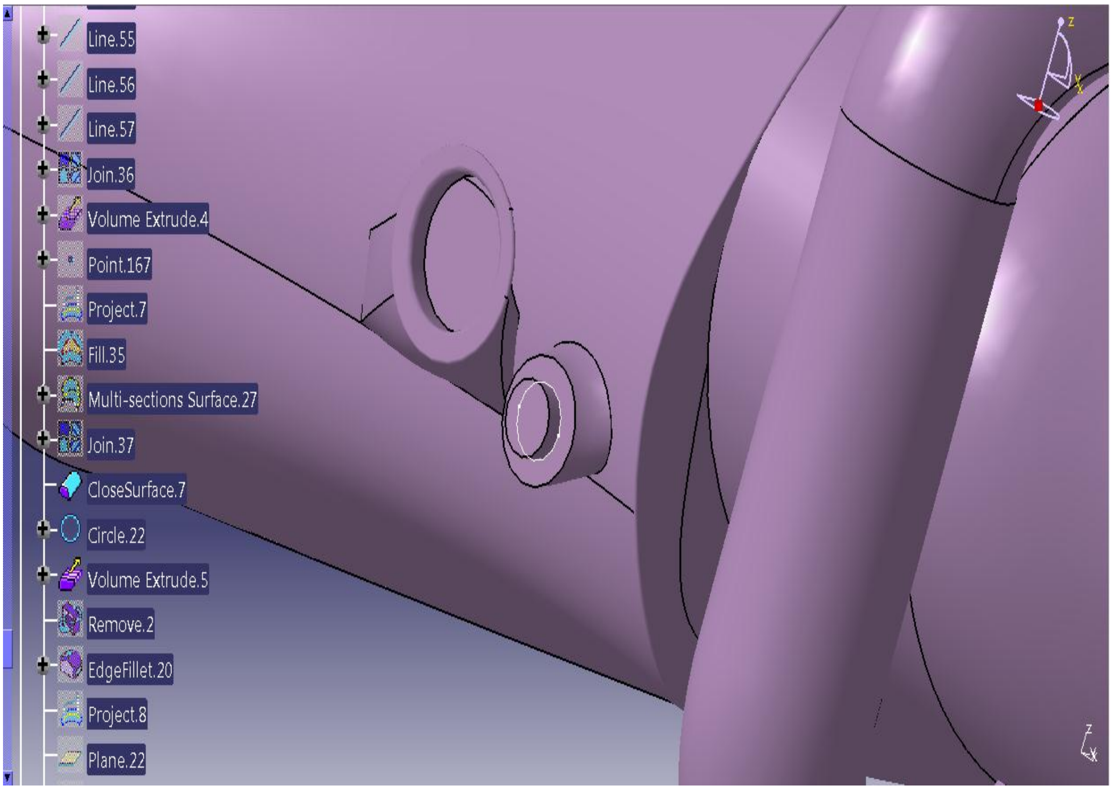
Picture 2.4.56: Ignition's peripheral surface.



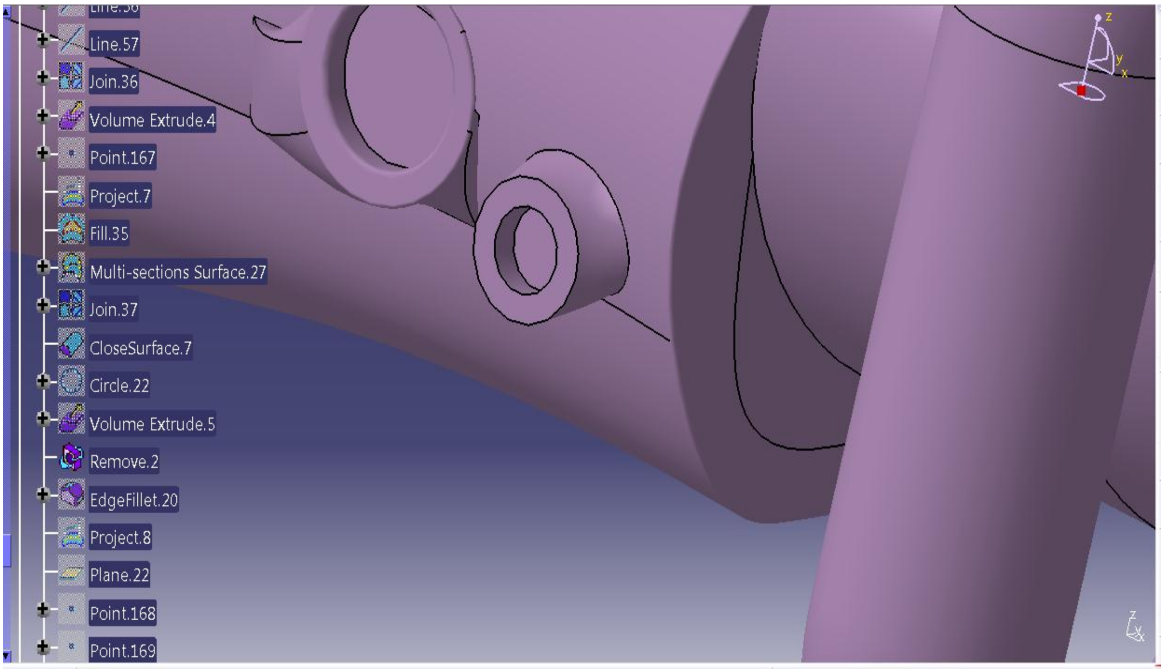
Picture 2.4.57: Ignition’s initial solid volume.



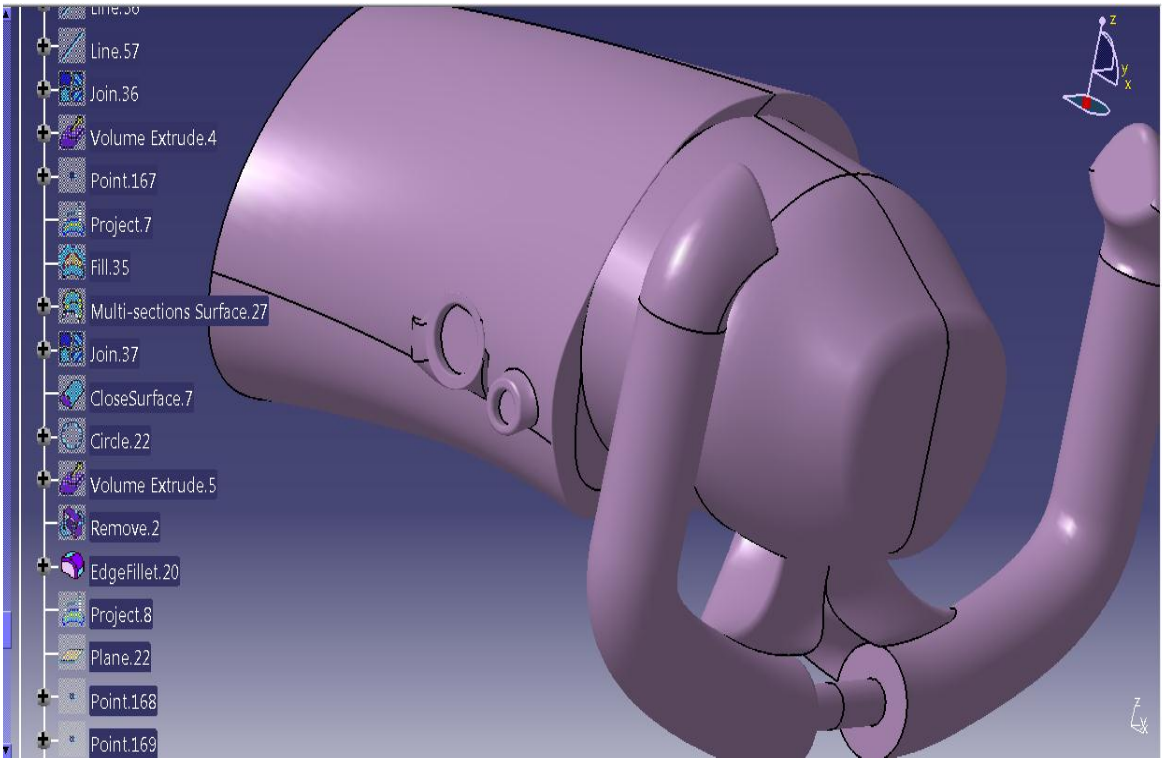
Picture 2.4.58: Auxiliary's volume wireframe creation.



Picture 2.4.59: Auxiliary volume of extrusion.



Picture 2.4.60: Ignition’s final volume.



Picture 2.4.61: Steering wheel part - complete volume.

2.5 DESIGN OF THE DASHBOARD PART

The dashboard part is the largest and most complicated volume inside the car's cockpit. This part constitutes the cockpit's control center and, consequently, integrates many functions. The primary volume contains many ''sub-elements''. Those sub-elements, which are indicator screens, air-nozzles and other similar parts, could be designed as independent parts and be attached later at the primary volume. However, this technique was avoided for reasons of simplification. For the same reasons the geometry was designed to be as simple as possible, without setting aside the style or aesthetic scope of the design. In order to create this volume, the same technique as before was used. The solid part was divided in several modules, each one of them being designed separately.

The first module was the dashboard's front face. At first, the wireframe model was constructed, which consisted of four sections. Each section resembles an ''arrowhead'' geometry (Picture 2.5.1). Each section was constructed by four Splines and one line in order to be closed. In addition, the sections of the model were designed to be equidistant to each other. Some additional lines were created in order to connect the four sections (Picture 2.5.2). This connectivity will facilitate the creation of the solid volume, as it will become clear later at the volume creation phase. Then the curves that existed inside each section were unified together with the use of "*Join*" operation command.

The solid volume was derived directly from the wireframe model by using the "*multi sections volume*" operation (Picture 2.5.3). For the execution of the command, the four joined sections were used as inputs inside the section frame, while the lines that connect the four sections were used as guides. The results are shown in Picture 2.5.4 below. Finally the symmetrical second half of the volume was created by using the "*Symmetry*" operation command. The plane that was used as input for the symmetry operation was the xz-plane, which is also the plane of symmetry of the whole solid geometry (Pictures 2.5.5 – 2.5.6).

After in the design process, the creation of the wireframe model of the cockpit followed. This wireframe model (as can be inferred from Picture 2.5.7) is a lot more sophisticated and difficult to design than the previous ones. This certain model doesn't contain separate sections that are blended together for the creation of the surface and eventually of the solid volume. It is a 3-dimension layout of the final solid geometry. Then, a surface was created on every single hydra of the geometry's wireframe. The peripheral surfaces were created as multi-section surfaces or as filled surfaces (Picture 2.5.8). Following the creation of the peripheral surfaces, a unification of all those surfaces was a mandatory task that should be performed. In order to fill the empty context of the peripheral surfaces with material, those surfaces should be used as a single entity. The unification of all those surfaces was made by using the "*Join*" operation command. Finally the solid volume was finalized with the use of "*close surface*" command (Picture 2.5.9). The second half of the symmetrical volume was created with the "*Symmetry*" operation command. Similarly, as with the front face part, the symmetry plane was the xz-plane (Picture 2.5.10). The resulting dashboard's initial solid geometry with the dashboard's face attached is shown in Picture 2.5.11. The next volume to be designed was the console's volume that separates the area between the driver's and the passenger's seat. Firstly, the wireframe model was constructed. The main characteristic of the wireframe's model was its planarity. As all the wireframes designed above, this certain wireframe was constructed by several B-Splines (Picture 2.5.12). Then, in order to construct the final solid volume, a unification of the Spline curves should follow. This was done by using the "*Join*" operation, defining as boundaries inside the command all the Spline curves of the section. After the unification of the wireframe model, the solid volume was created by the "*Sweep Volume creation*" command. As profile inside the command the wireframe's model planar section was imported, while the scanning direction that the generative curve (profile) followed was defined to be the same as in line 254 (Shown in Picture 2.5.13 below). The solid volume that derived from the use of the "*Sweep Volume*" command is depicted in Picture 2.5.14. Moreover, the second symmetrical Volume was created by using the "*Symmetry operation*" command (Picture 2.5.15). As before, the plane that was used as input for the symmetry operation was the xz-plane. Finally, to satisfy styling criteria, several filletings were added to the existing volume. These filletings were inserted with the "*edge fillet*" command (Picture 2.5.16).

A touch screen was designed to be placed on the solid volume. This screen was designed to be used for entertainment and informational purposes. The steps for the design of this geometry were similar to those already used for the design of the other modules of the part. Firstly, the wireframe model was designed. The screen's geometry contained an intrinsic symmetry with respect to xz-plane. As a result, only half of the geometry needed to be defined. The first stage for the wireframe's design was the creation of the internal edge of the screen's frame (Picture 2.5.17). The shape of the frame was similar to a rectangular geometry. The internal frame's wireframe was designed on a plane 20 mm above the external surface of the support volume. In order to approach an actual screen's geometry in as a detailed way as possible, two corners were introduced on the upper-right and lower-right corners of the geometry (Picture 2.5.19). Additionally, there was a constraint that was imposed by the solid design. The wireframe's support ought to have the same degree of inclination as the solid surface to be placed on. As a result, in order to satisfy the constraint, the plane was defined with an inclination of 43 degrees (shown in Picture 2.5.18 below). Then a similar wireframe model was constructed on a newly defined plane with 15 mm distance towards the positive direction of x-axis (Pictures 2.5.20 – 2.5.21). This intermediate distance between the two sections' wireframes was used to construct a transitional surface between the external surface of the screen's frame and its surface. Moreover, an additional wireframe model was constructed on the same plane. This new set of curves represented the external layout of the outer surface of the screen's volume (Picture 2.5.22). For the completion of the screen's wireframe, several lines were created in order to close the wireframe's geometry (Picture 2.5.23). Subsequently, the next phase of the design was the creation of the peripheral surfaces of the model. In this step, the aim was to cover the wireframe's model openings with surfaces. Once this task was completed, then it was possible to fill it with material, using the "*Close surface*" command. The first surface in the creation order was the frame's external surface. This surface was defined as a multi-section surface (Picture 2.5.24). The generative curves that were used for the surface definition were the two lines which closed the section's wireframe. As guide curves the two remaining joined curves of the plane were used that formed the curvilinear course of the generative curves. Second in creation was the lateral external surface of the screen. This surface was also created as a multi section surface (Pictures 2.5.25 – 2.5.26). As generative curves for the creation the two vertical lines to the lines that were used at the previous "*multi-section surface*" creation command were used, while as guide curve the second guide curve of the previous command was used. Then, the internal surface adjacent to the volume's external surface was created. In contrast to the previous two surfaces, this certain surface was designed as a "*fill*" surface (Picture 2.5.27). Next in sequence was the transitional surface between the screen's surface and the external frame's surface. Naturally, this surface was designed as a "*multi section*" surface. The generative and guide curves for the creation of the surface were defined as depicted in Picture 2.5.28 below. In order for the solid volume to be formed all these surfaces had to be joined. Once this was done, half of the volume was created using the "*close surface*" command (Pictures 2.5.29 - 2.5.30). The second symmetrical half was created with the "*Symmetry*" command. The symmetry plane, which was used as an input for the mirroring process, was the xz-plane (Pictures 2.5.31- 2.5.32). The integrated digital volume of the dashboard with the attachment of the previously designed touch screen is represented in Picture 2.5.33.

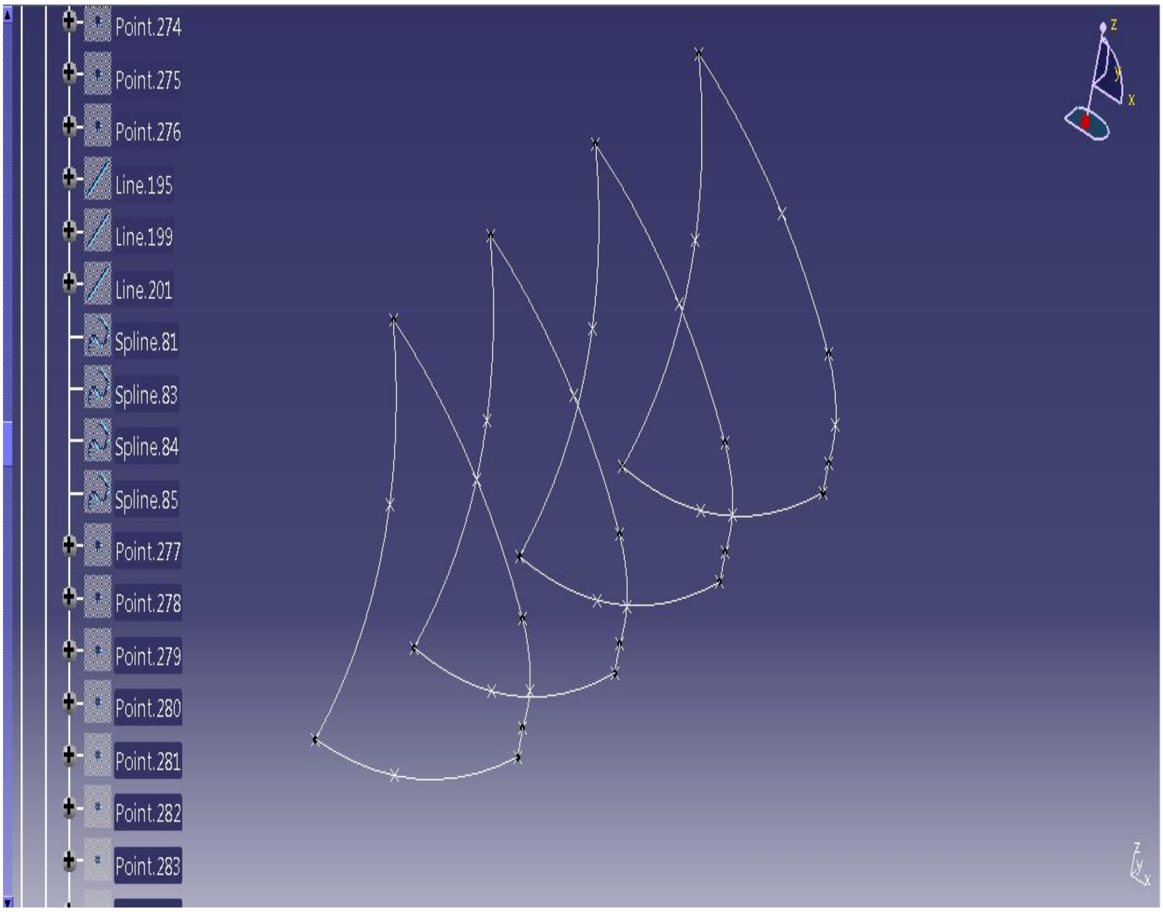
The next module to be attached on the dashboard volume, was the on board screen. The purpose of this second screen was different from that of the touch screen. Though the first screen's purpose was for entertainment, the second screen exists as a replacement of the dashboard's instrumental panel. To create the on board screen, an opening at the dashboard's volume was needed. This opening was created with the "*remove volume*" operation technique, the same way as before (at steering's column cover volume). First the volume that was subtracted from the dashboard was constructed. The wireframe of this volume was a rectangular with four curves as sides (Picture 2.5.34). The four curves were joined together for the creation of a single section. After the unification the volume derived directly from the wireframe model. The new volume was created by using the "*extrude*" volume operation. The extruded volume was expanded for 145 mm towards the positive direction of x-axis (Picture 2.5.35). The result of this operation was two intersecting solid volumes, a mandatory condition for the implementation of the "*remove volume*" operation. Then, the remove volume operation was chosen for the creation of the desired void. As base object inside the command, the dashboard's front face was inserted. The extruded volume was inserted as the volume that was deducted from the base volume (Pictures 2.5.37- 2.5.38). The board's screen volume included an intrinsic symmetry with respect to a plane parallel to xz-plane. As a result, only half of it needed to be defined. Firstly, the wireframe model of the screen was designed. The wireframe was similar to the initial's screen wireframe in terms of shape and design (Picture 2.5.39). With similar techniques as with the previous screen, the surfaces on the model were created (Picture 2.5.40). Before the creation of the solid volume all the above surfaces were joined together in order to form a single entity (Picture

2.5.41). The solid volume was defined with the "*close surface*" volume creation command (Picture 2.5.43).

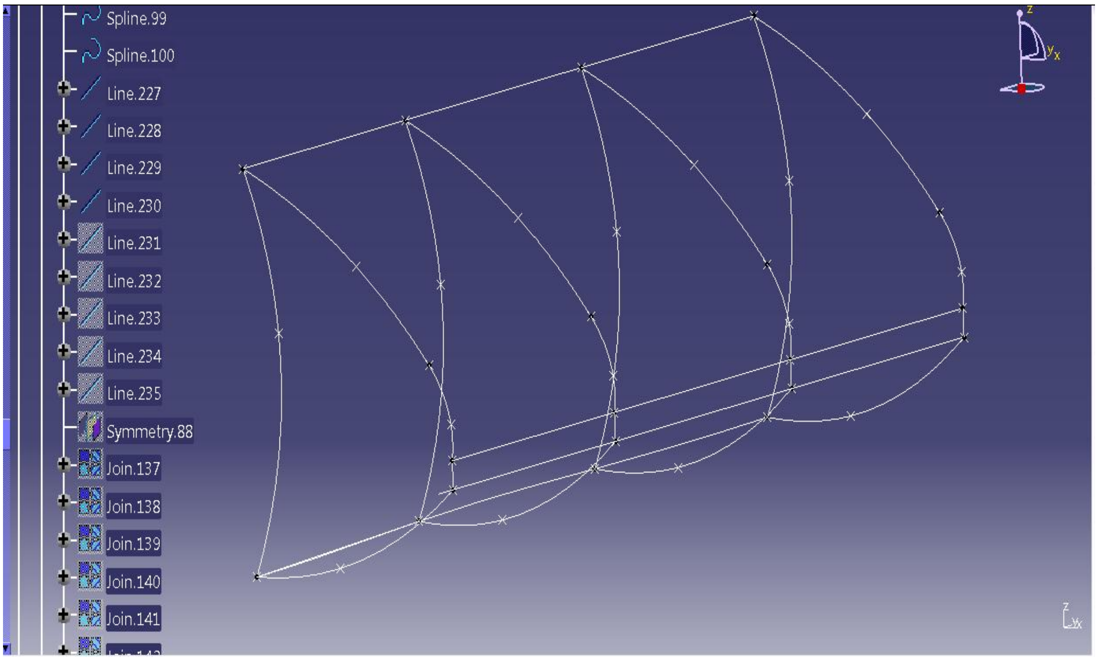
Moreover, there was a need for an added volume inside the 3-D design. This volume acted as a connection between the screen's and the dashboard's solid volume. The wireframe of the auxiliary volume was defined first (Picture 2.5.44). The wireframe contained two primary sections. Each section was designed in order to be tangent to the screen's rear surface and to the dashboard's exposed surface respectively. In addition, two more B-Spline curves and lines were defined. The functional purpose of these curves was to use them as guide curves during the solid volume creation stage. The final solid volume derived directly from the wireframe model. For its creation the "*multi-section volume*" creation operation was used. The definition of this command is quite similar to the corresponding surface creation command (Picture 2.5.55). As generative curves inside the command the two sections that unified the wireframes were used, while as guide curves the four other curves that connected the two sections were inserted. For the creation of the symmetrical volumes the "*Symmetry*" operation was used. As plane of symmetry inside the command the highlighted plane shown in picture 2.5.56 below was inserted.

The last geometrical feature that was added to the dashboard's solid volume were the air ducts. In order to create the solid volume of the air ducts, the wireframe model was designed first. Firstly, two planes parallel to zy-plane were defined. These planes had a distance of 70 mm towards the x-axis and a distance of 40 mm towards the z-axis (Picture 2.5.58). The first plane was created as part of the dashboard's solid volume. Next in sequence, the definition of the air ducts sections followed. The two sections were one circular and one elliptical respectively on each plane (Picture 2.5.59). Then, inside the wireframe model, a number of curves were imported. The functional purpose of the additional curves was their use as guide curves (Picture 2.5.60). The solid volume of the air duct was designed using the "multi sections" volume command. As generative curves the two sections of the model were imported while as guide curves the additional curves that connected the two sections together were selected (Picture 2.5.61). In order for the solid volume to be functional it was mandatory to insert some openings. These openings represented the outlet of the air coming from the air conditioning system inside the car's volume. These openings were created using the "*remove volume*" operation. The volumes that represented the amount of material removed were created as follows. On the external surface of the air-duct's solid volume two elliptical sections were designed. By extruding these two elliptical profiles by 20 mm on both directions, defined by a vector in parallel, the two solid volumes were created (Picture 2.5.62). Then by executing two "*remove volume*" operations in sequence, posing as base object the initial multi section volume and as volumes to be removed the two extruded volumes of the final functional solid volume was formed (Picture 2.5.63). Finally for aesthetic reasons, some additional edge rounding operations were performed on the sharp edges of the air-duct (2.5.64). A second air-duct volume was created on the dashboard's solid volume. The creation of the second air-duct was exactly the same as with the first air-duct (Picture 2.5.65). Two more symmetrical air-duct volumes were compulsory to exist on the other side of the dashboard's volume. The additional air-ducts were created by using the “*symmetry*” operation. As plane of symmetry the xz – plane was inserted (Picture 2.5.66).

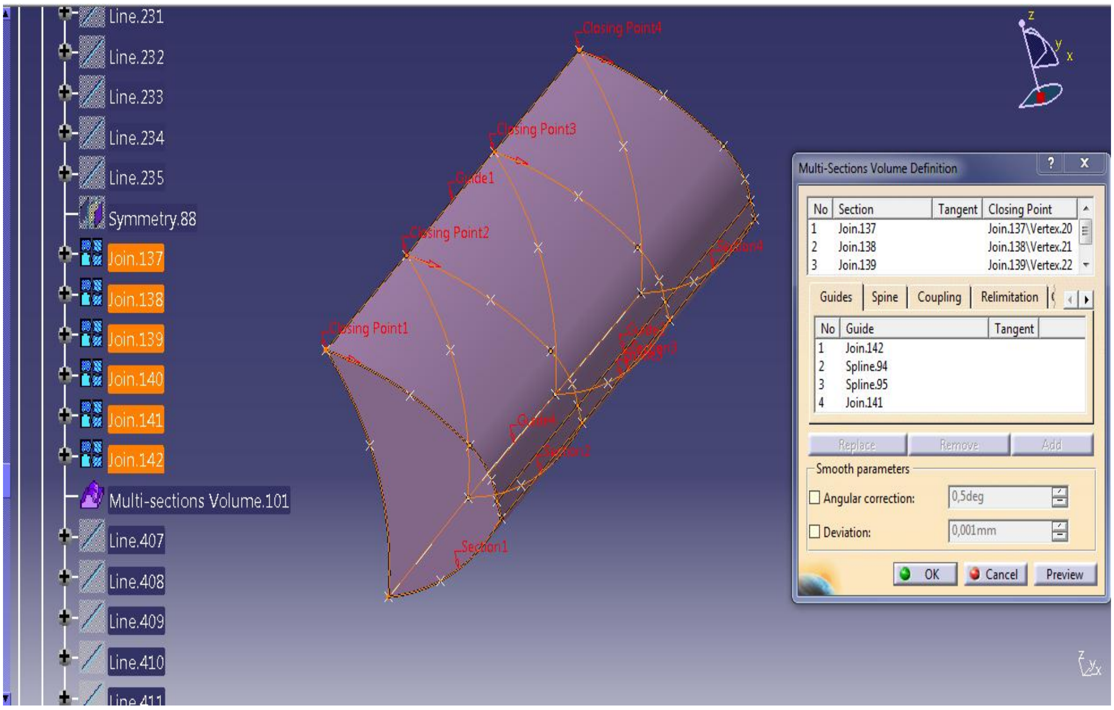
For the completion of the solid model, an additional feature should be included. This feature was the recess where the steering wheel was going to be attached on the dashboard. This recess was constructed by using the “*remove volume*” operation. Firstly, the wireframe model of the solid volume was designed. Then the area of the volume was enclosed by surrounding surfaces using the “*close surface*” command (Picture 2.5.67 – 2.5.68). With the symmetry volume operation, the symmetrical half of the desired volume was created (Picture 2.5.68). Then by executing the “*remove volume*” operation the volume to be removed was created (Picture 2.5.69).



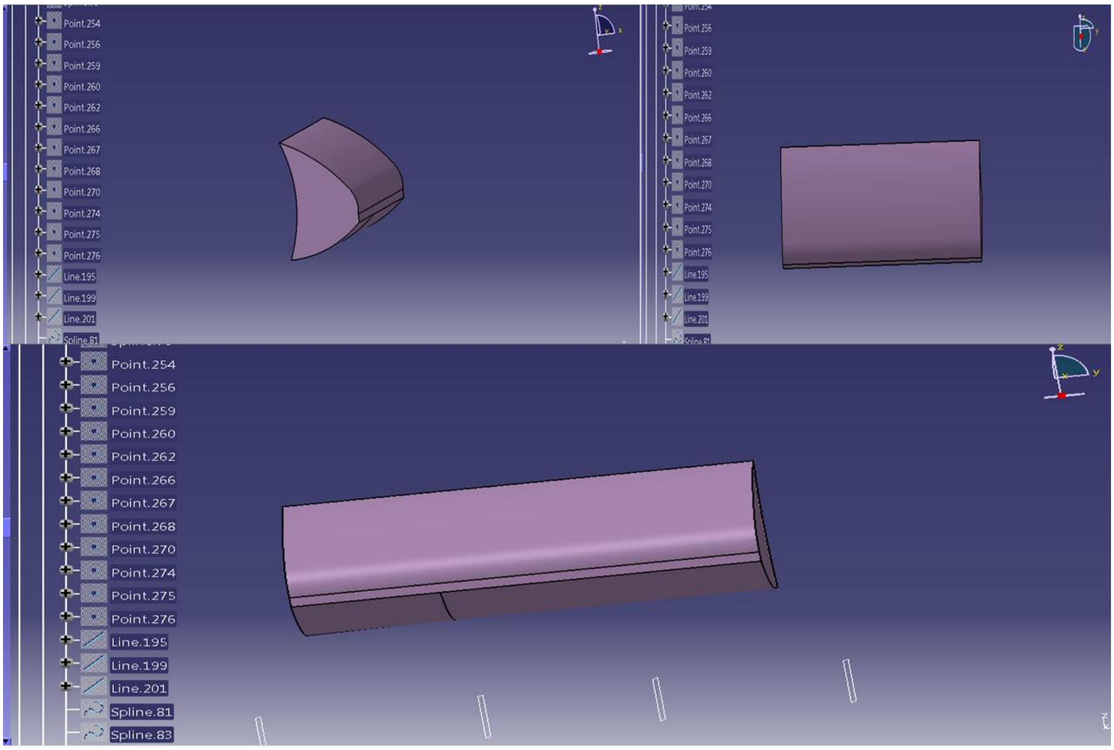
Picture 2.5.1: Wireframe model of dashboard’s front face.



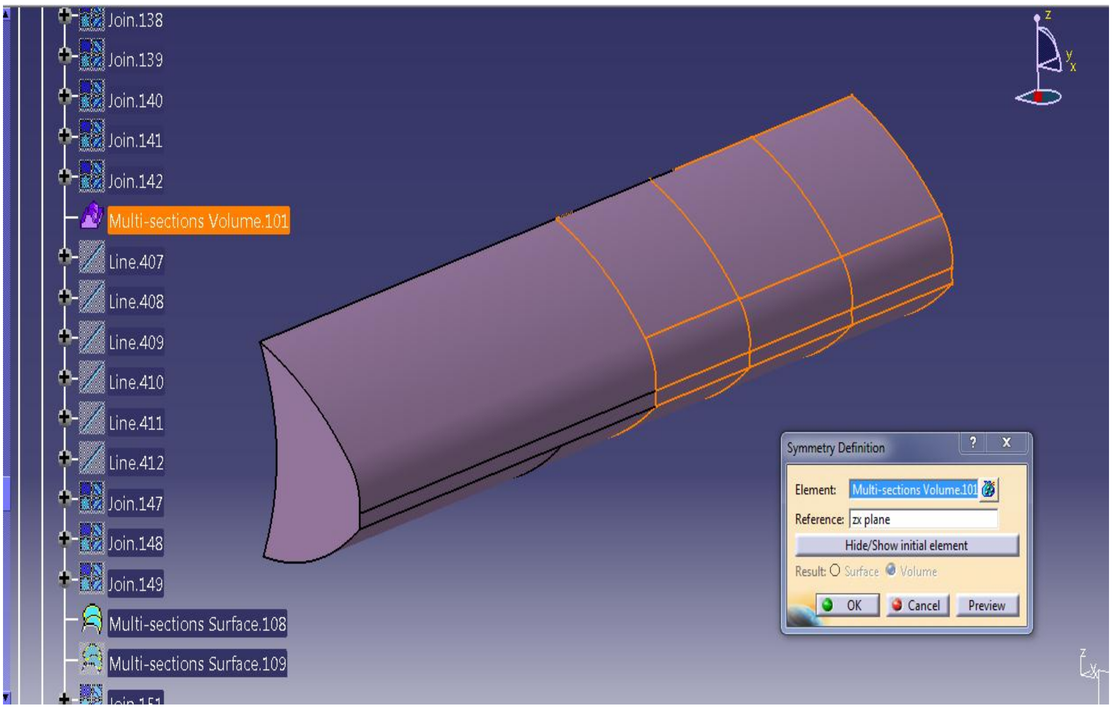
Picture 2.5.2: Wireframe of dashboard's front face – connection of the sections



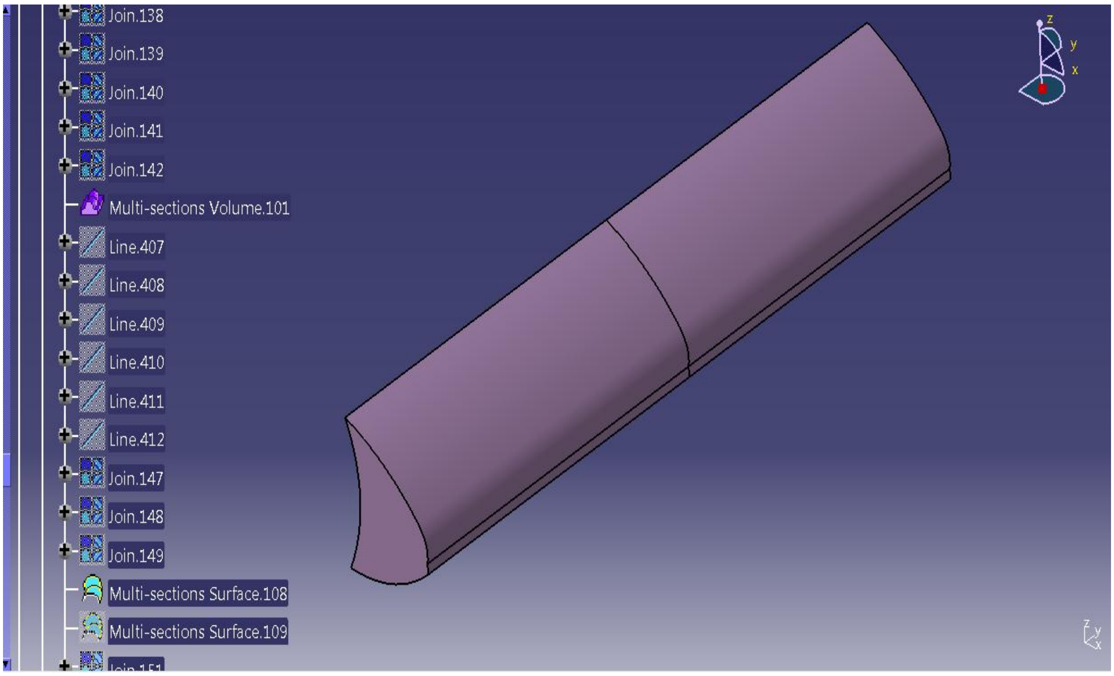
Picture 2.5.3: Creation of dashboard's front face solid geometry.



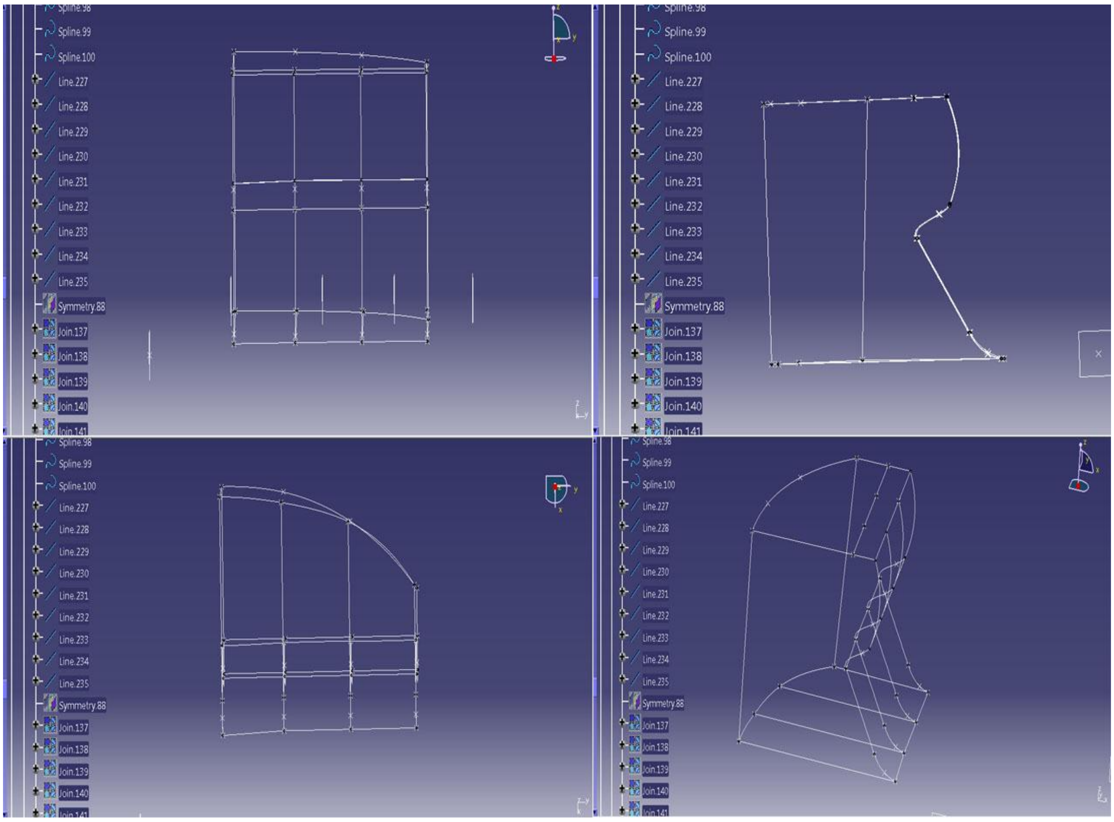
Picture 2.5.4: Dashboard's front face.



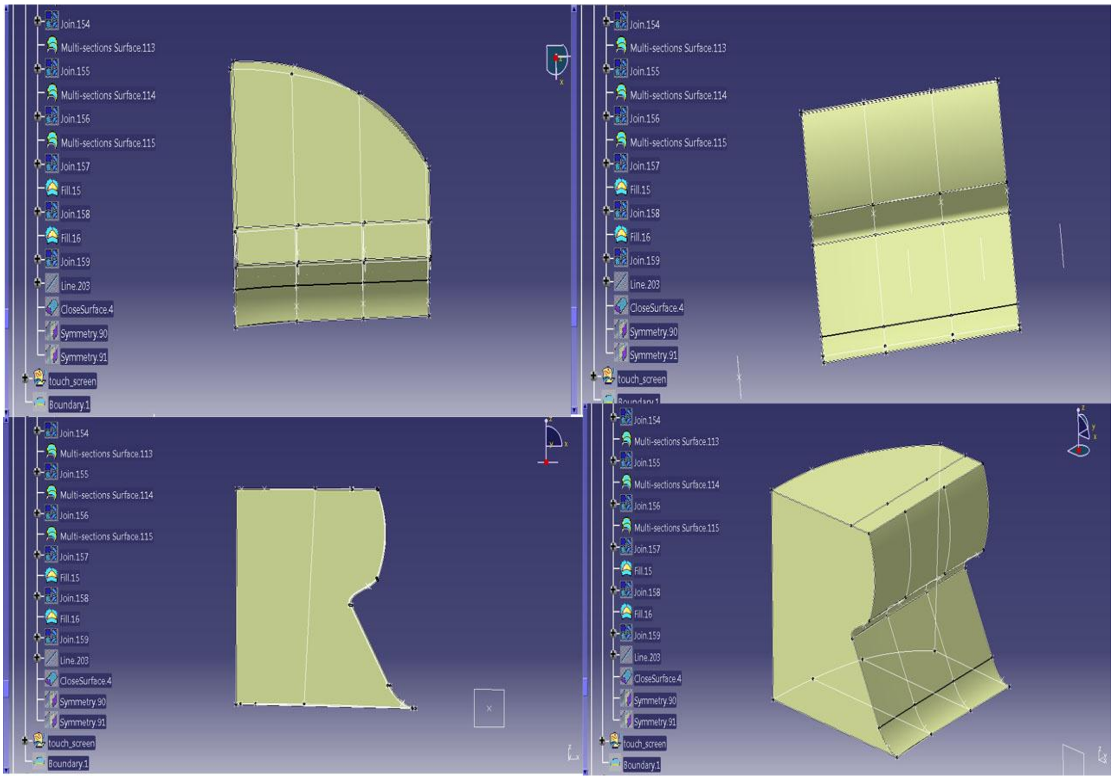
Picture 2.5.5: Symmetrical volume of dashboard's front face.



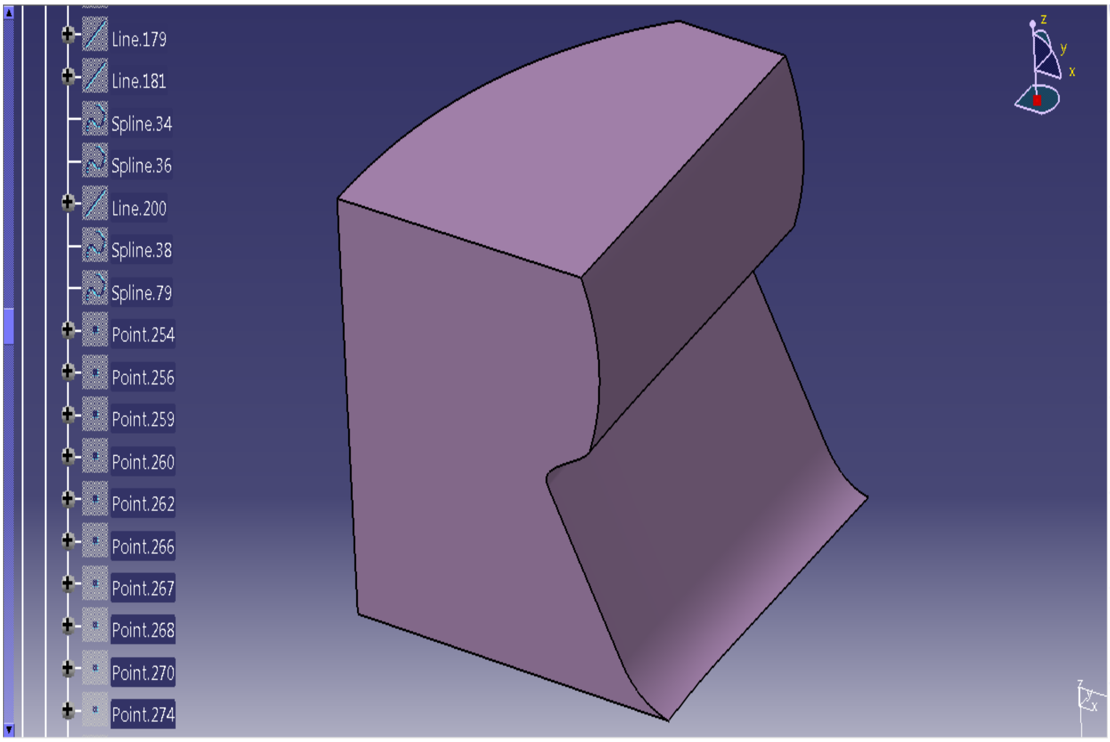
Picture 2.5.6: Symmetrical volume of dashboard’s front face.



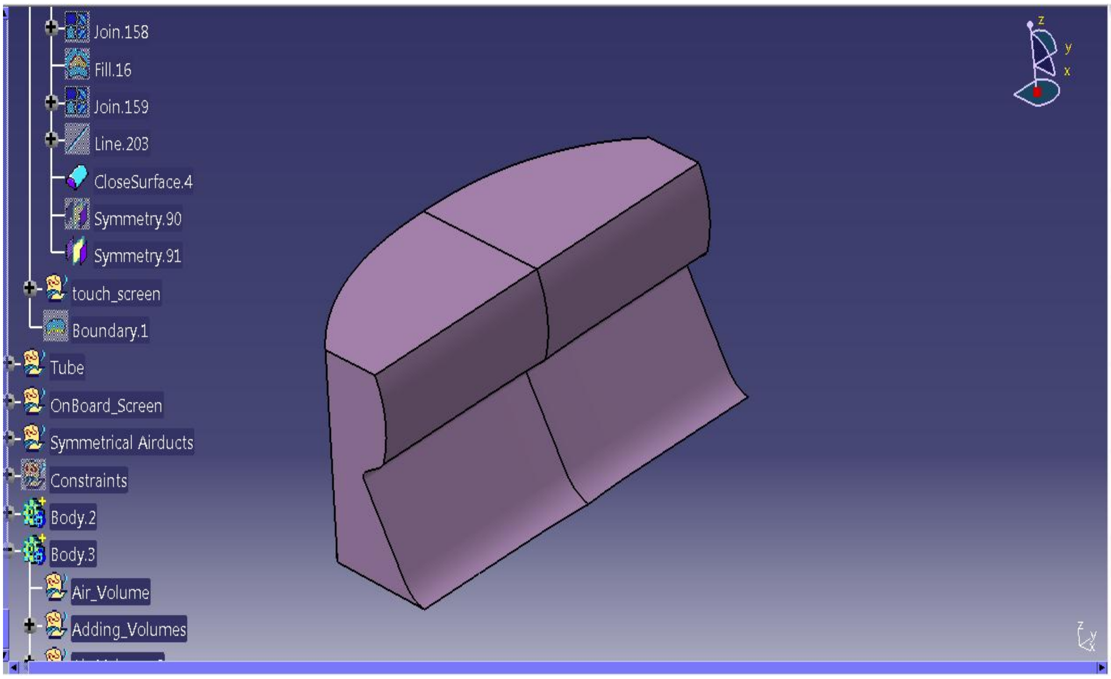
Picture 2.5.7: Dashboard wireframe design.



Picture 2.5.8: Dashboard's regional surfaces.



Picture 2.5.9: Dashboard's first symmetrical half.



Picture 2.5.10: Dashboard’s complete volume.

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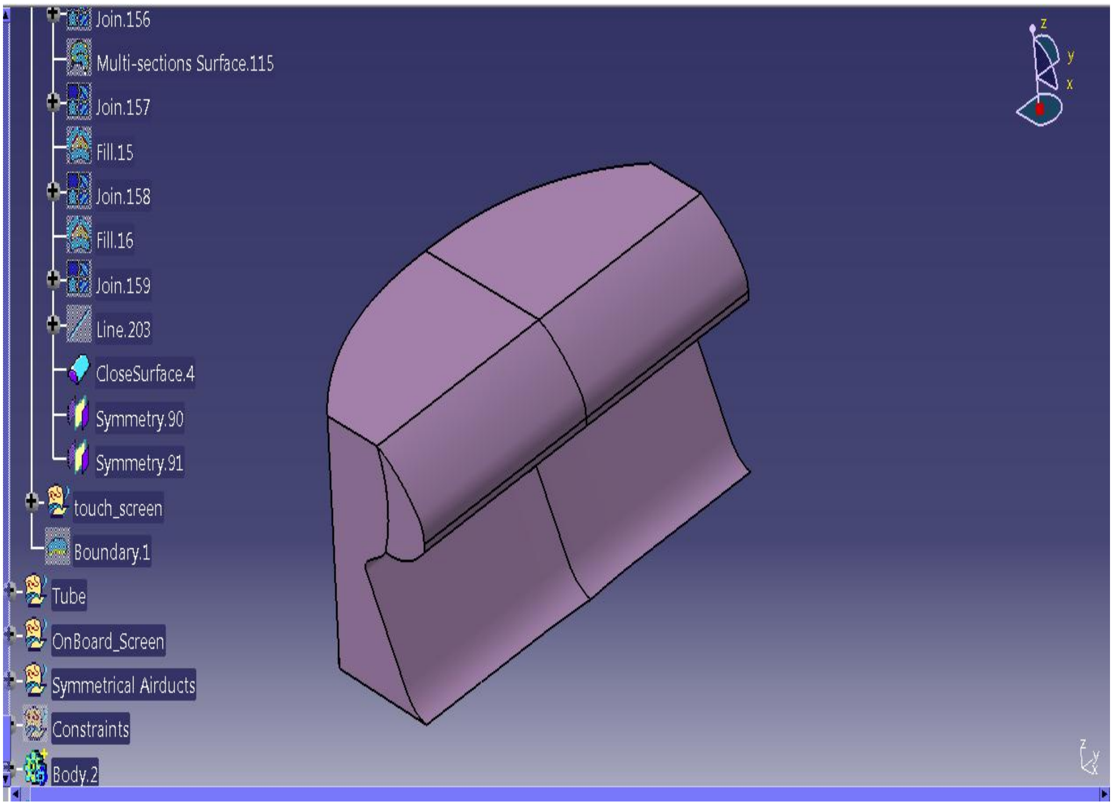
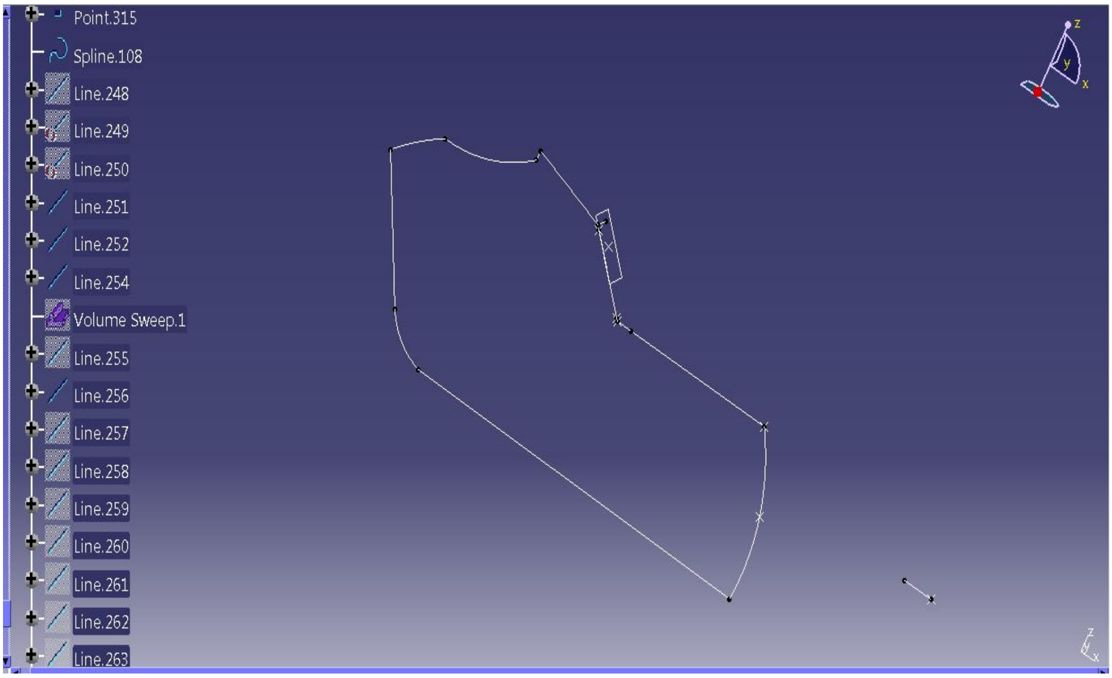
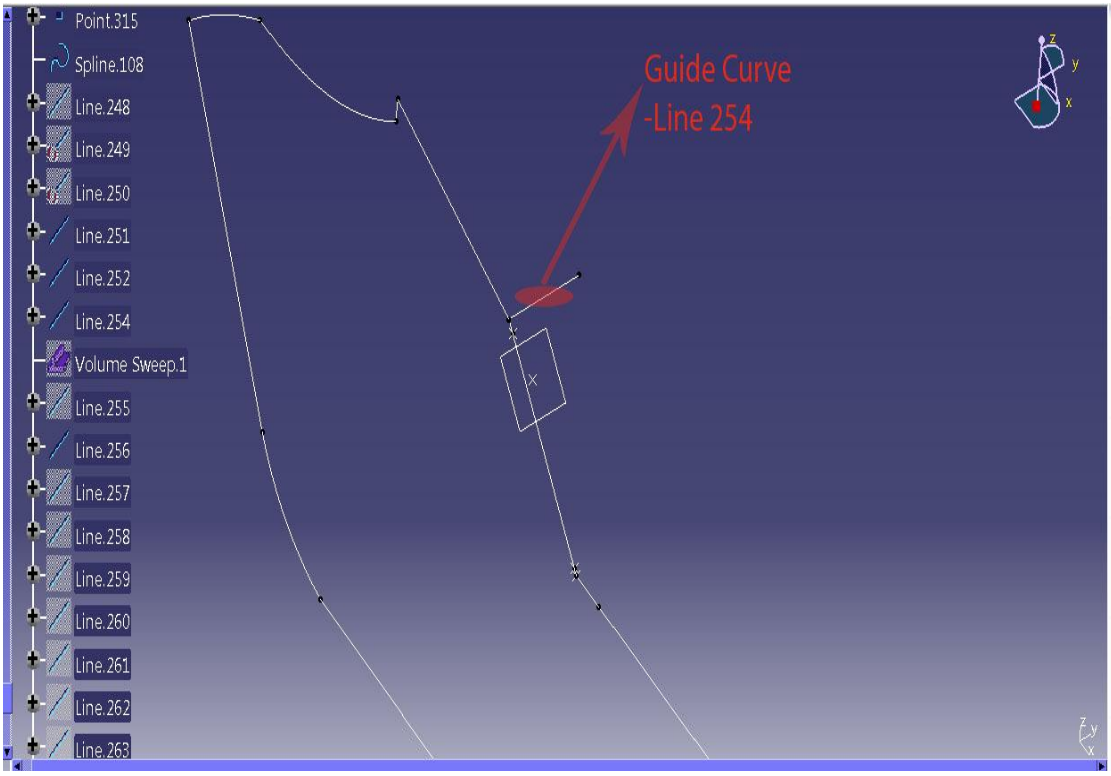


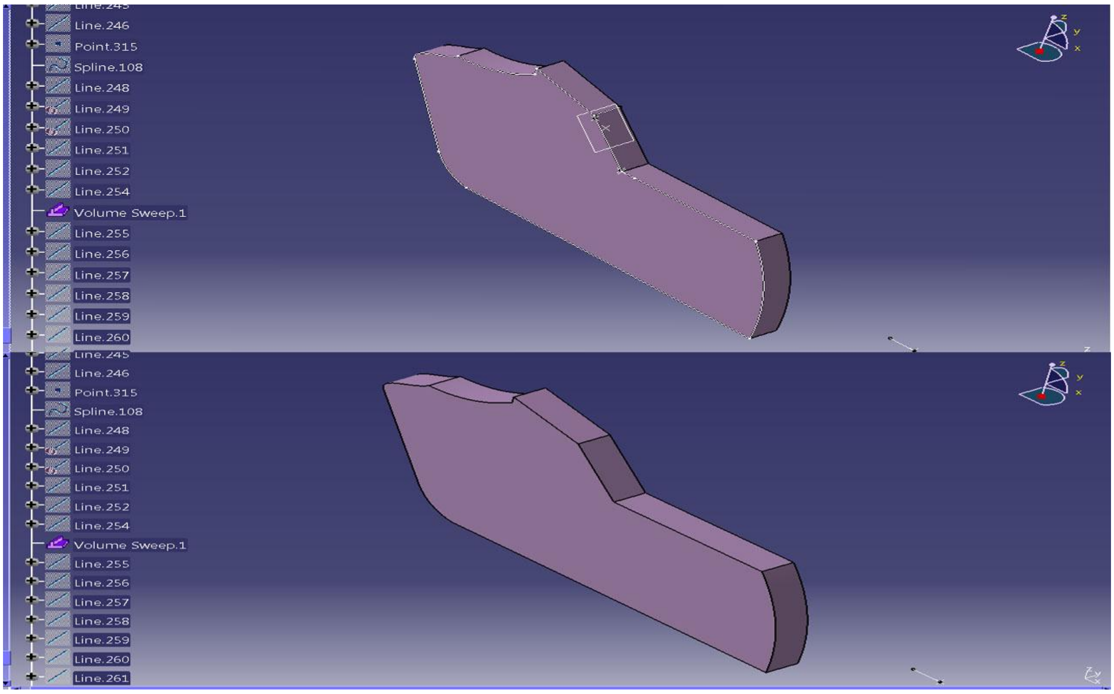
Picture 2.5.11: Dashboard’s volume with front’s face symmetrical half.



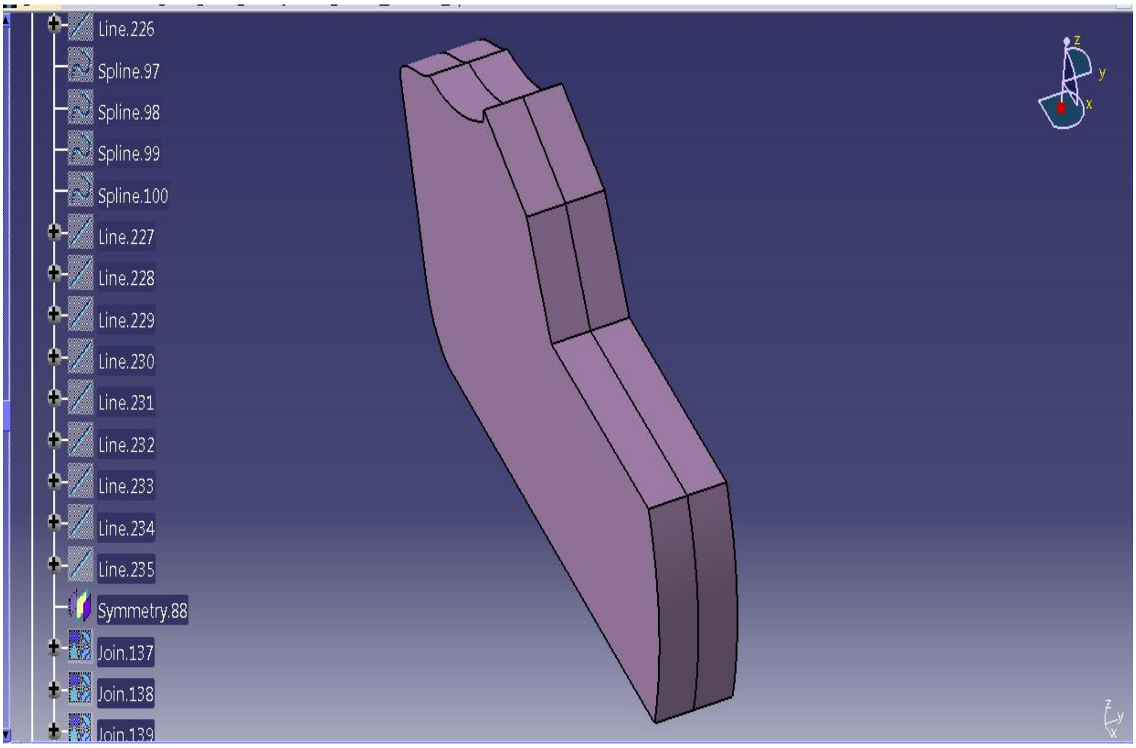
Picture 2.5.12: Dashboard's console's volume wireframe.



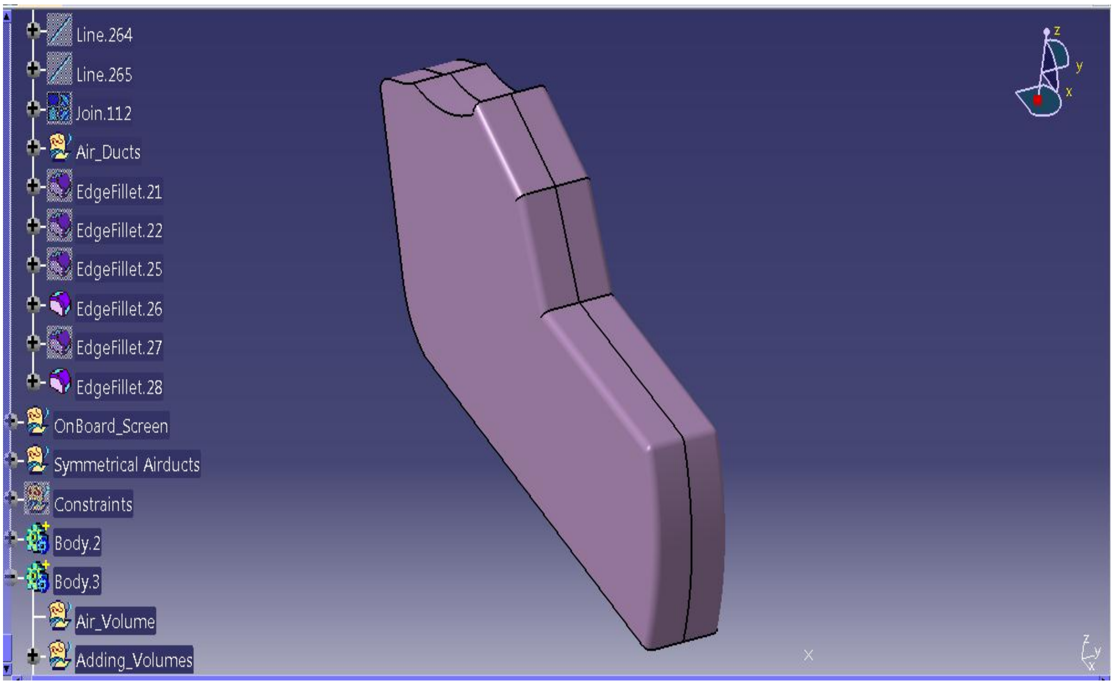
Picture 2.5.13: Sweep's operation guide curve definition.



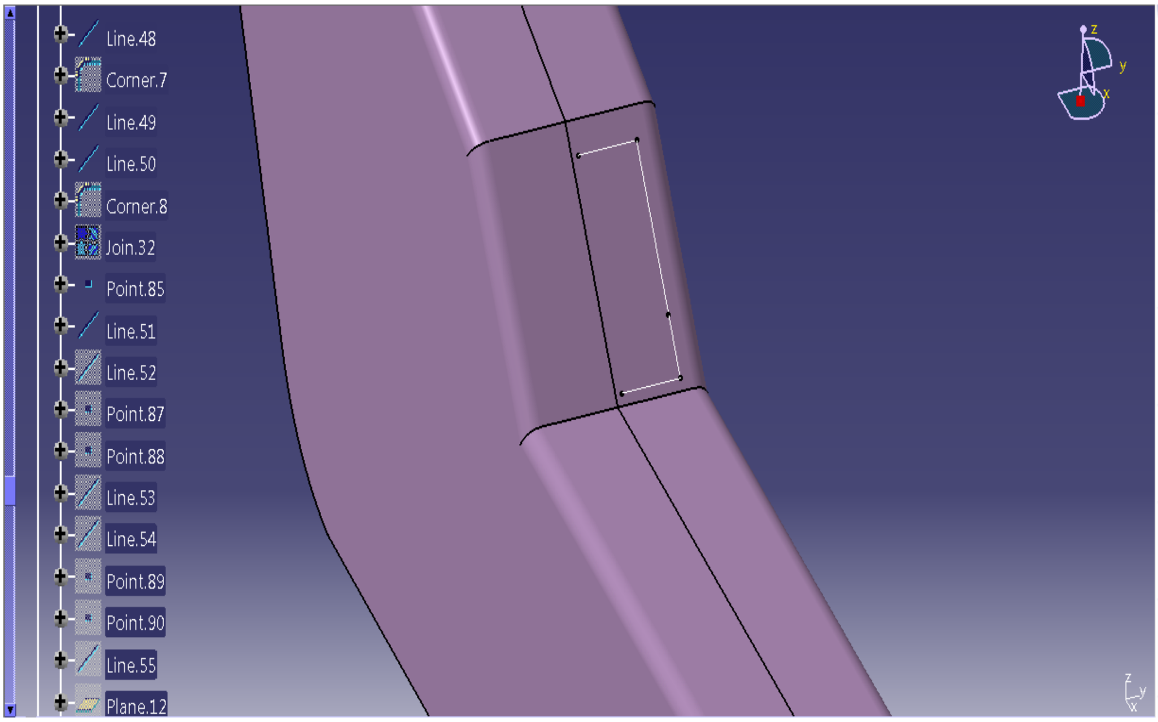
Picture 2.5.14: Console’s solid volume.



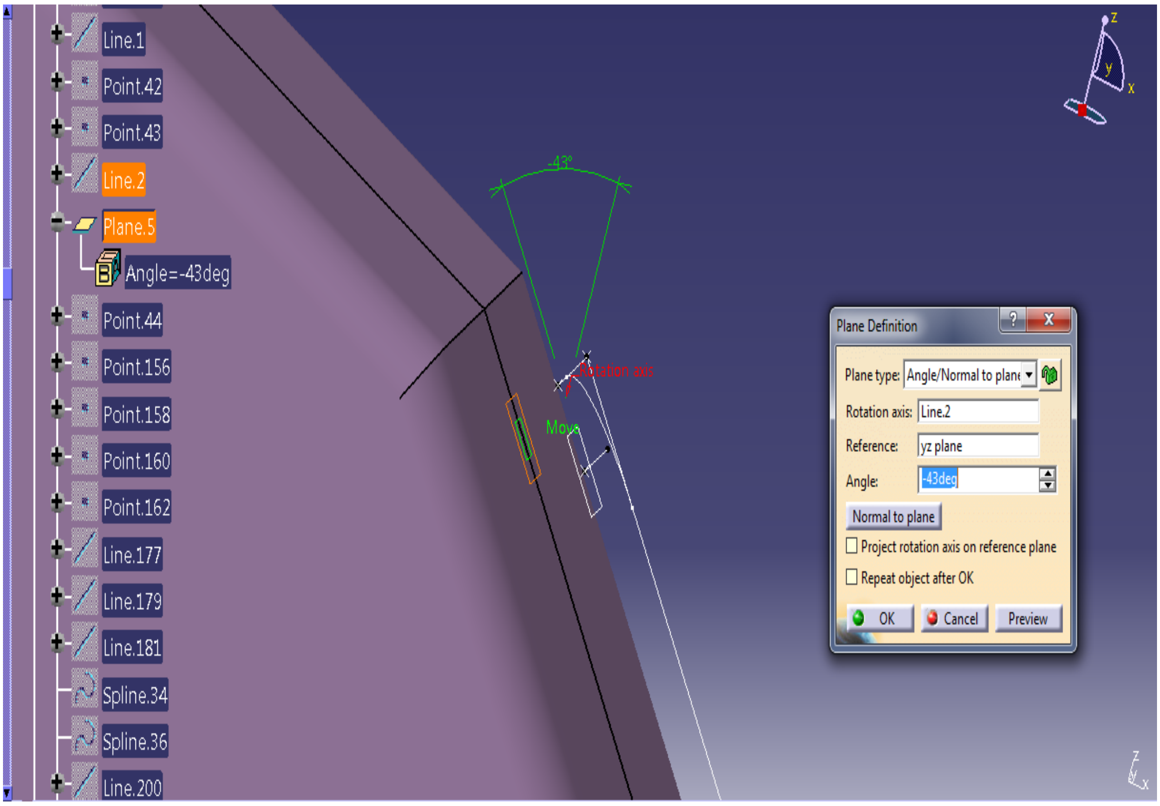
Picture 2.5.15: Symmetrical half of console’s volume.



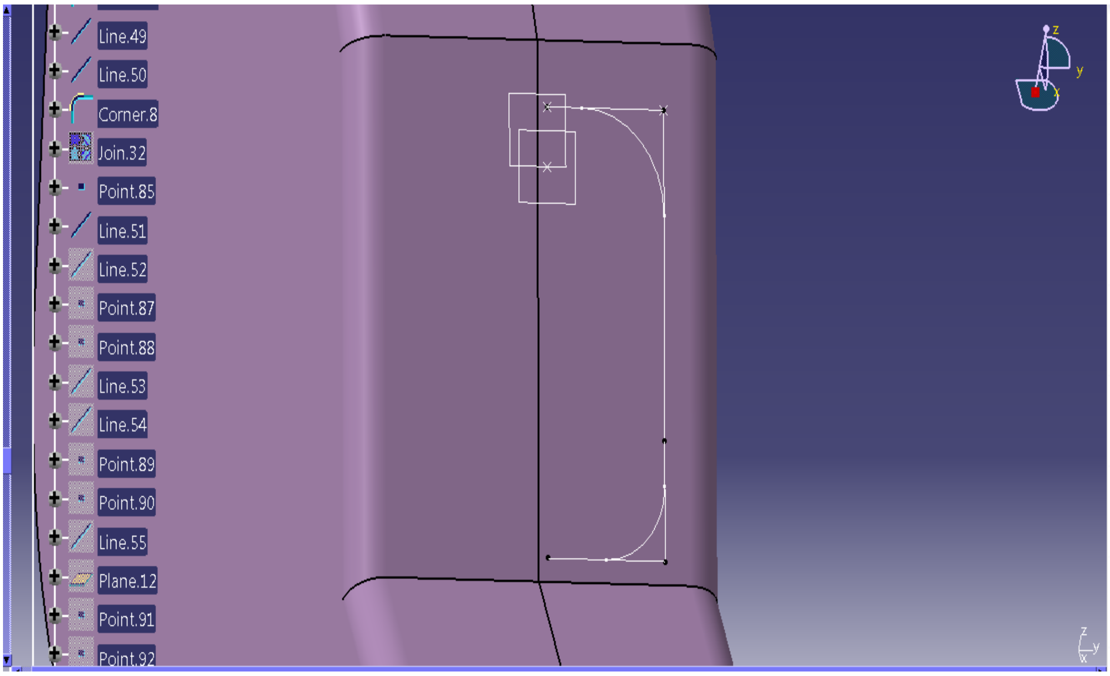
Picture 2.5.16: Console's complete volume with edge filletings.



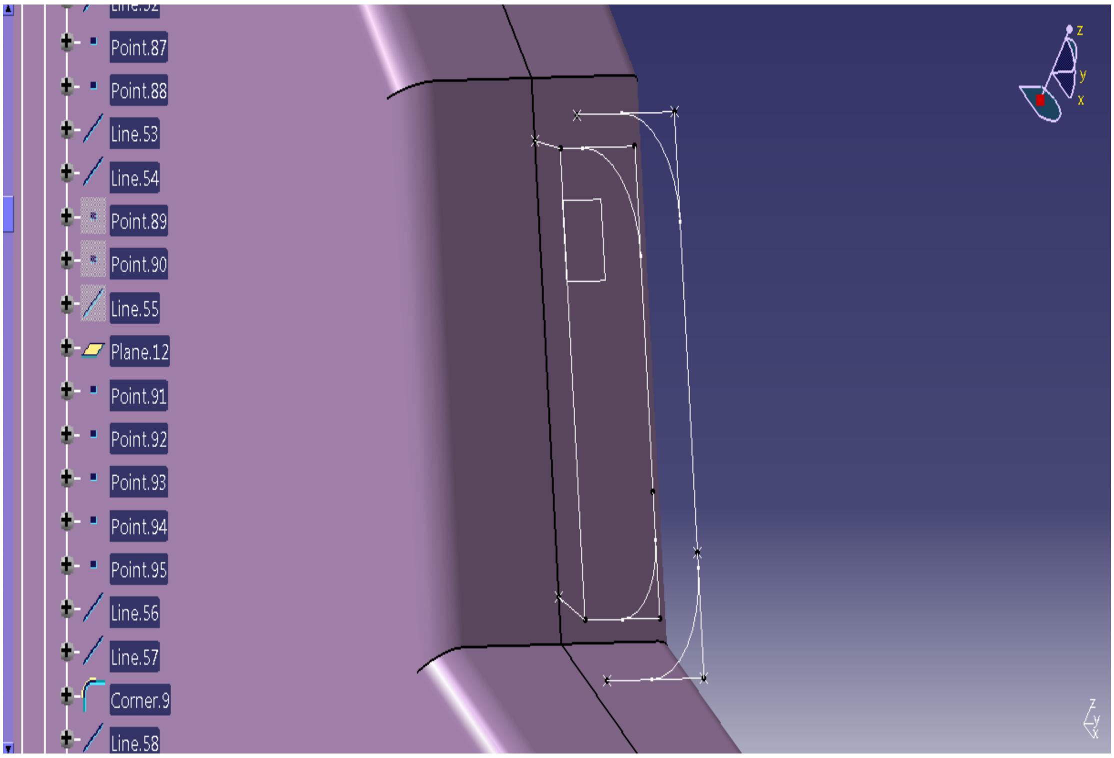
Picture 2.5.17: Touch screen's internal edge wireframe model.



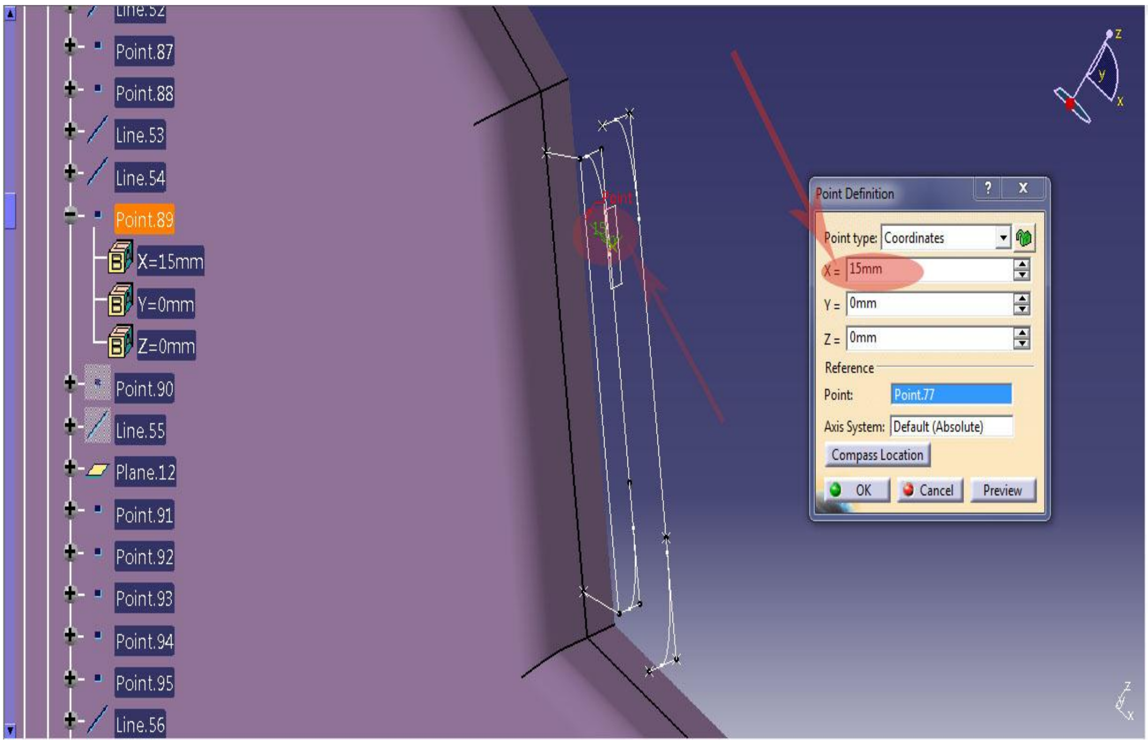
Picture 2.5.18: Touch screen's wireframe – plane's inclination.



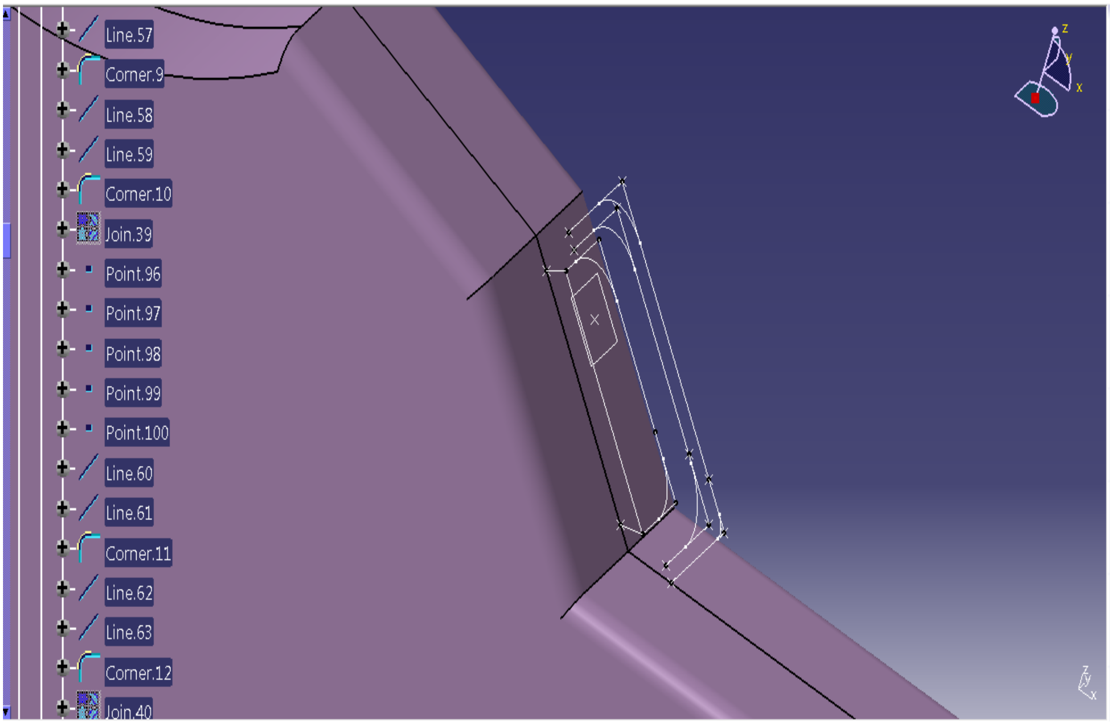
Picture 2.5.19: Touch screen - wireframe' cornered ends.



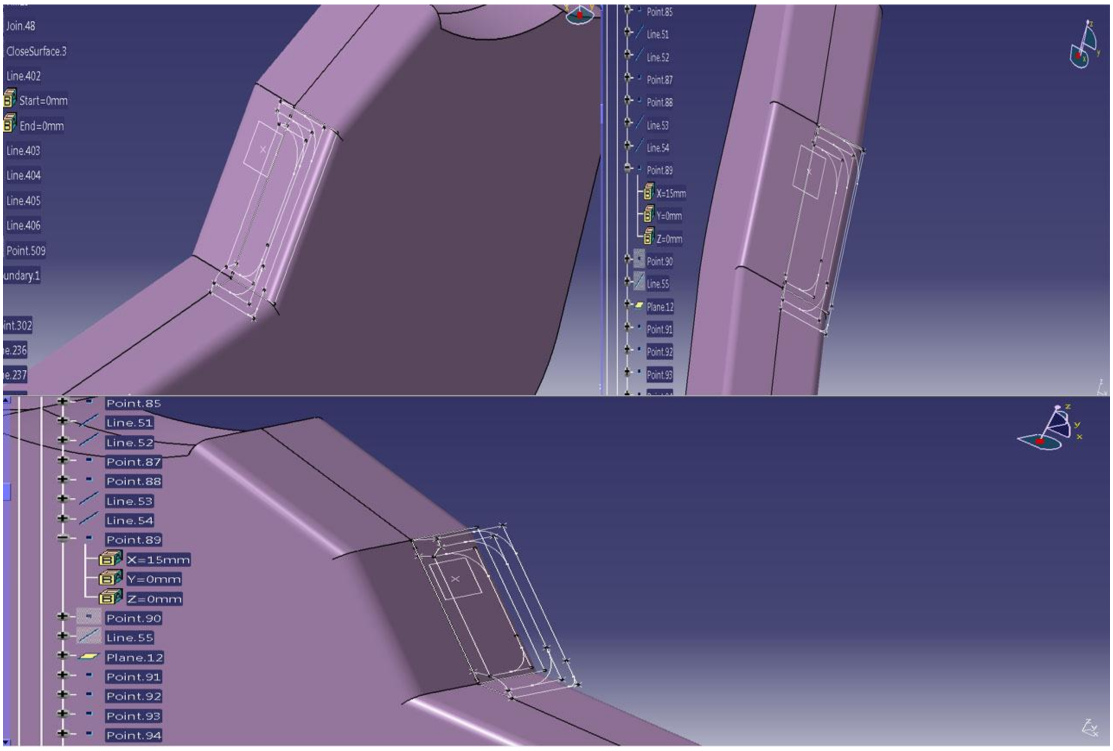
Picture 2.5.20: Touch screen's wireframe 2nd Section.



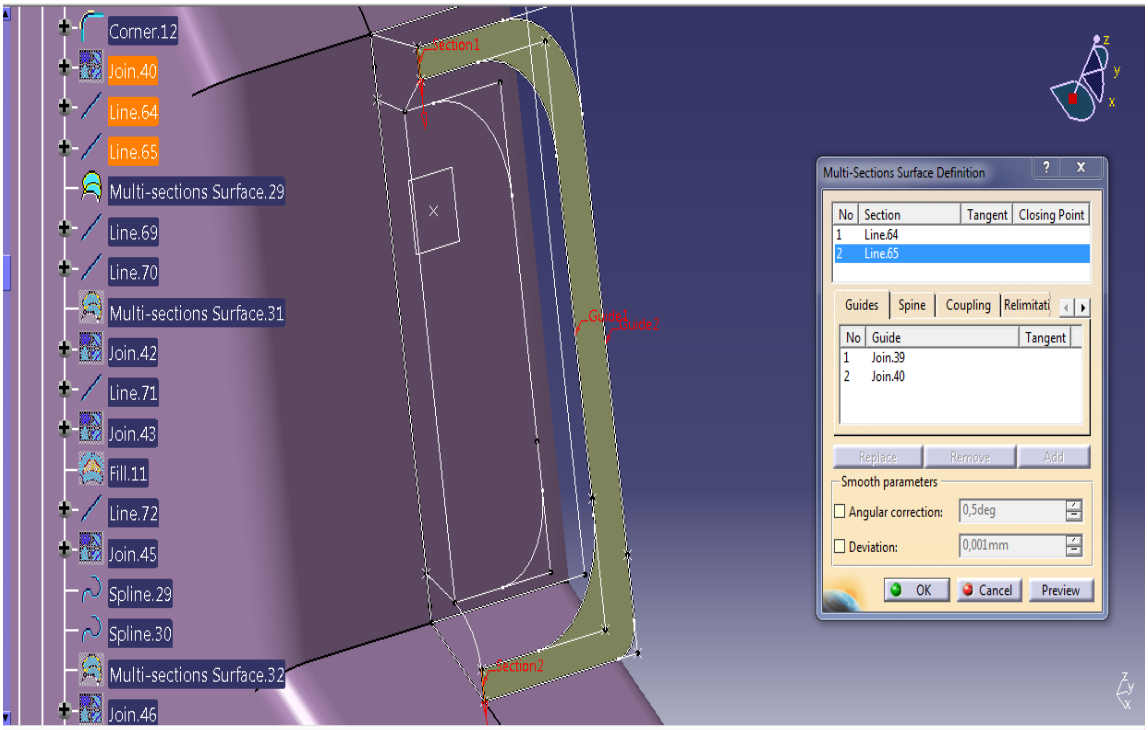
Picture 2.5.21: Distance from section number one.



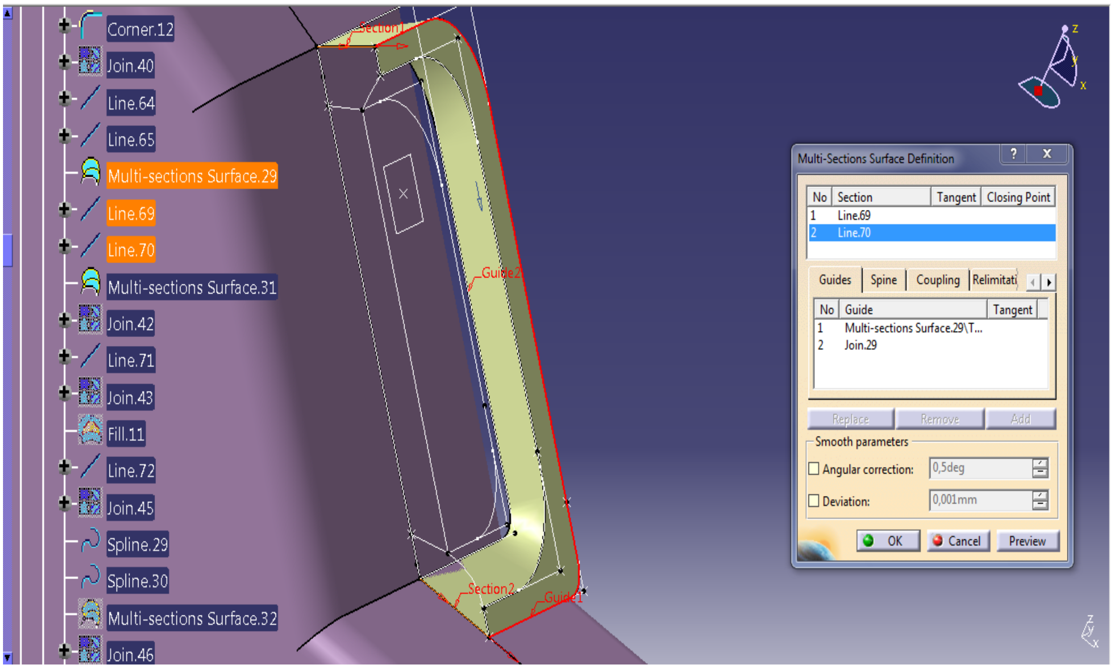
Picture 2.5.22: External’s section wireframe



Picture 2.5.23: Touch screen’s complete wireframe model.



Picture 2.5.24: External's frame surface creation.



Picture 2.5.25 Touch screen's peripheral external surface.

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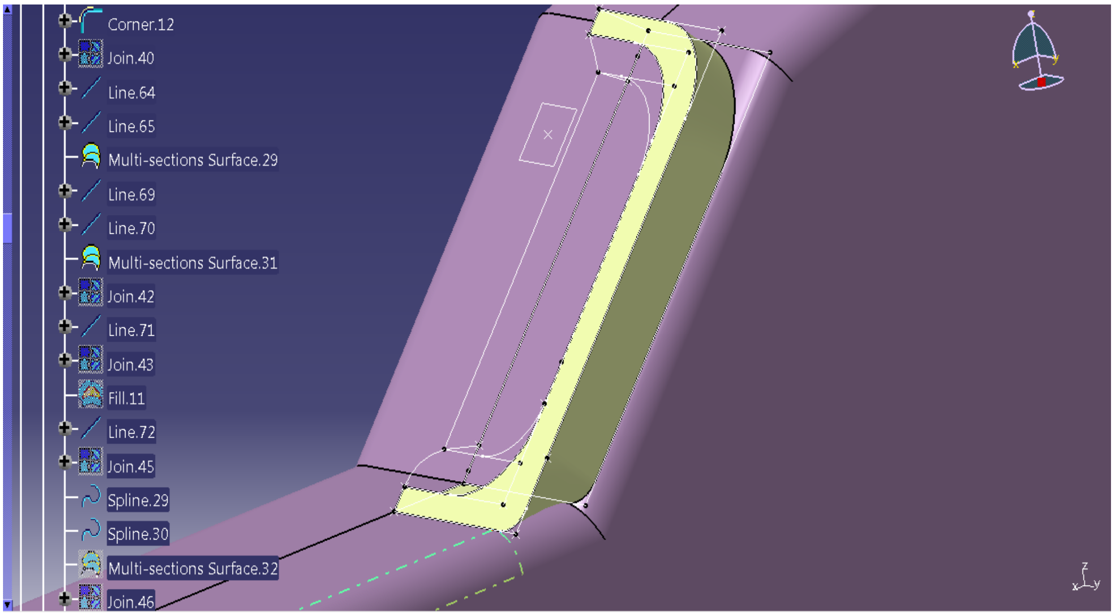
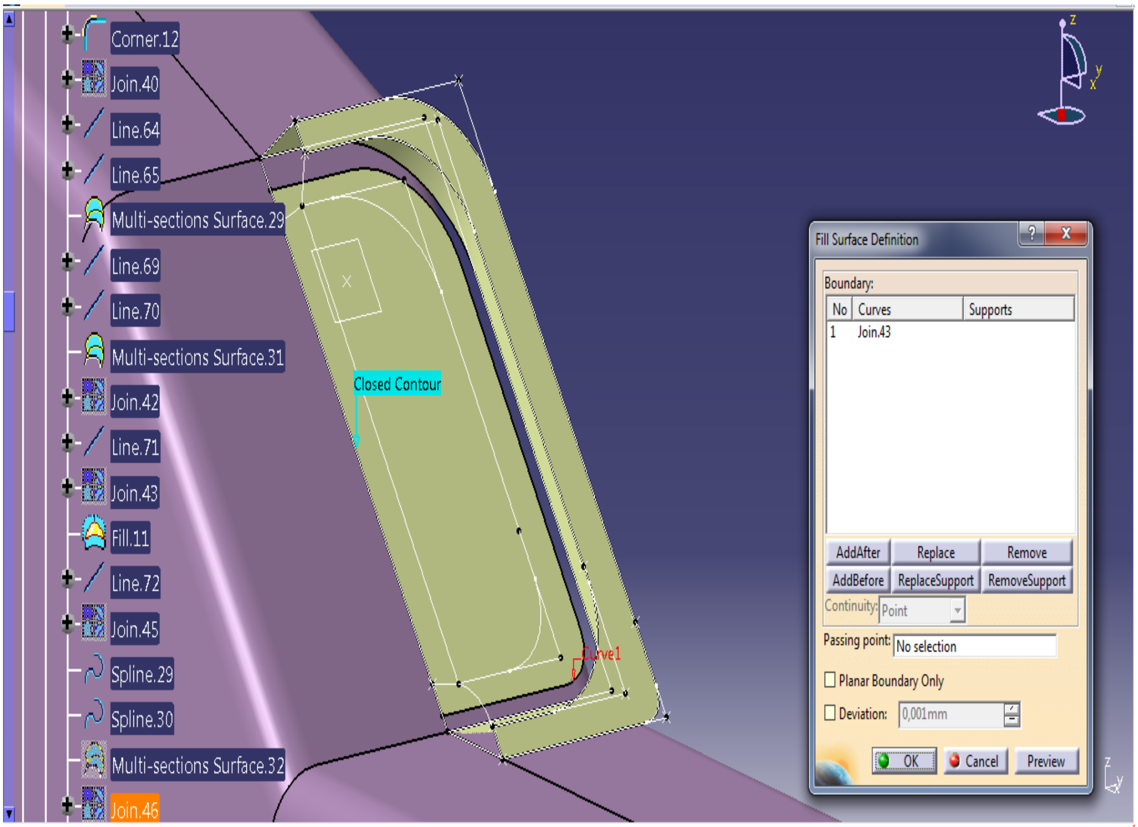
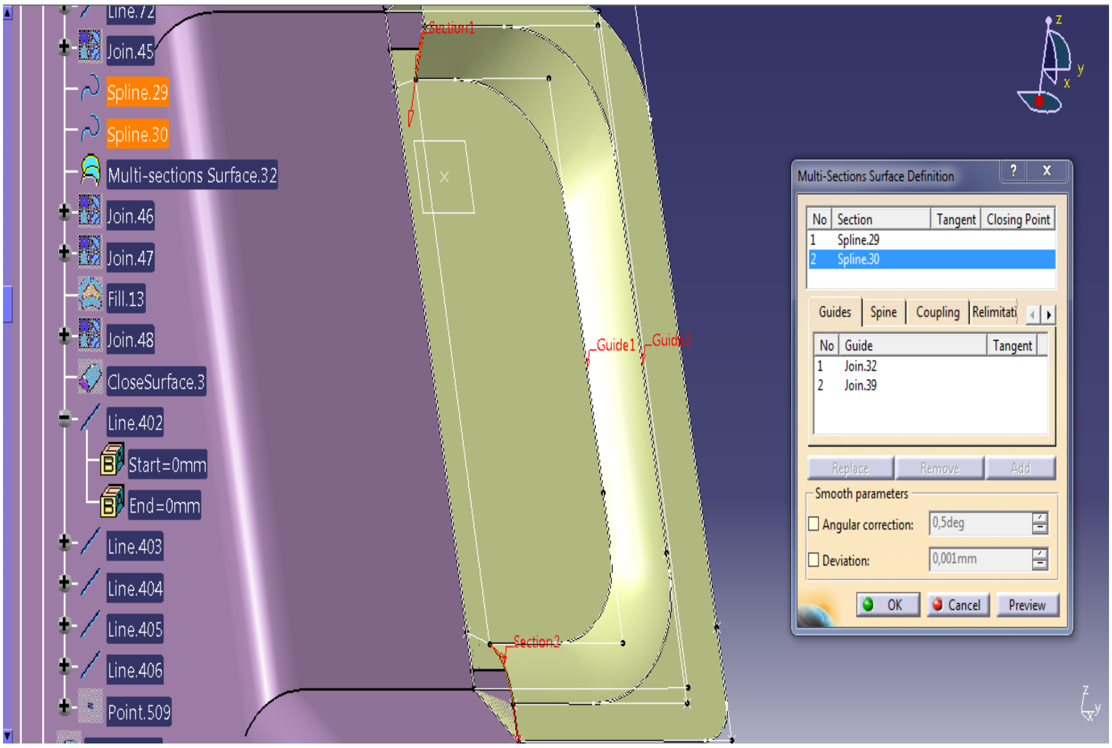


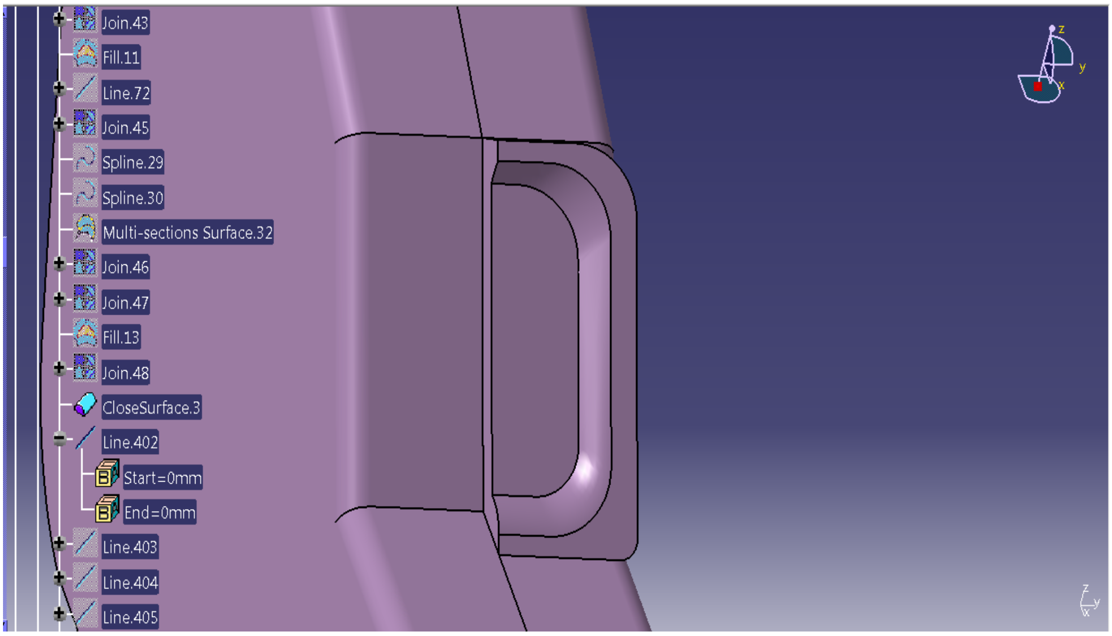
Picture 2.5.26: Touch screen's peripheral external surface.



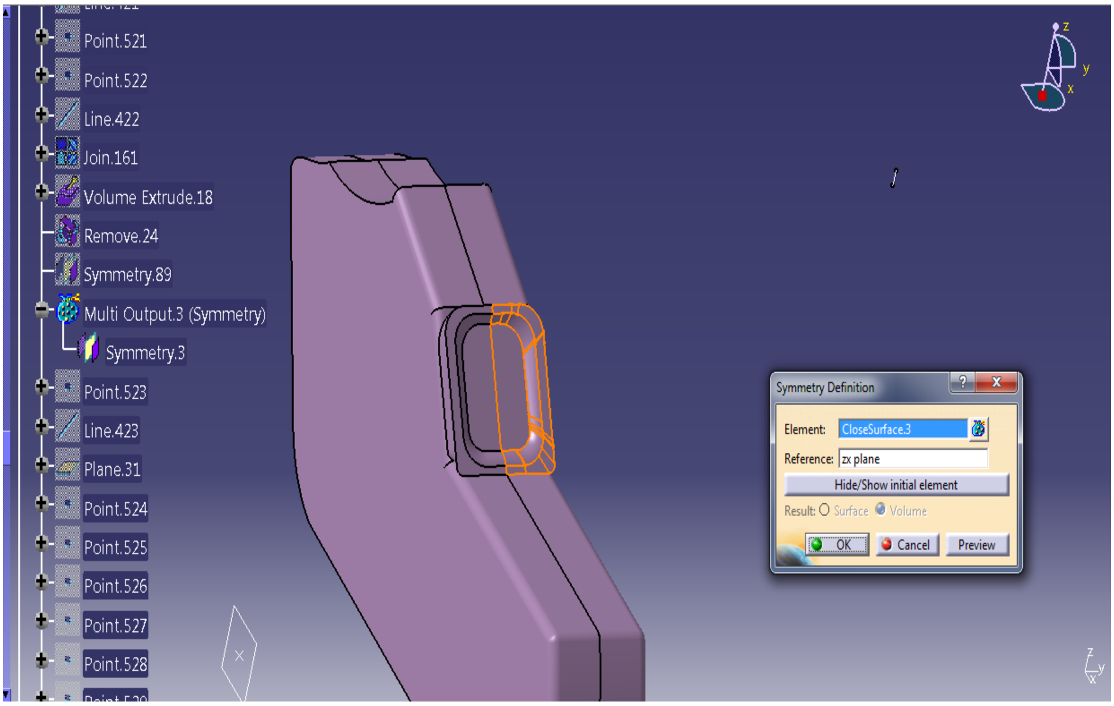
Picture 2.5.27: Screen's surface.



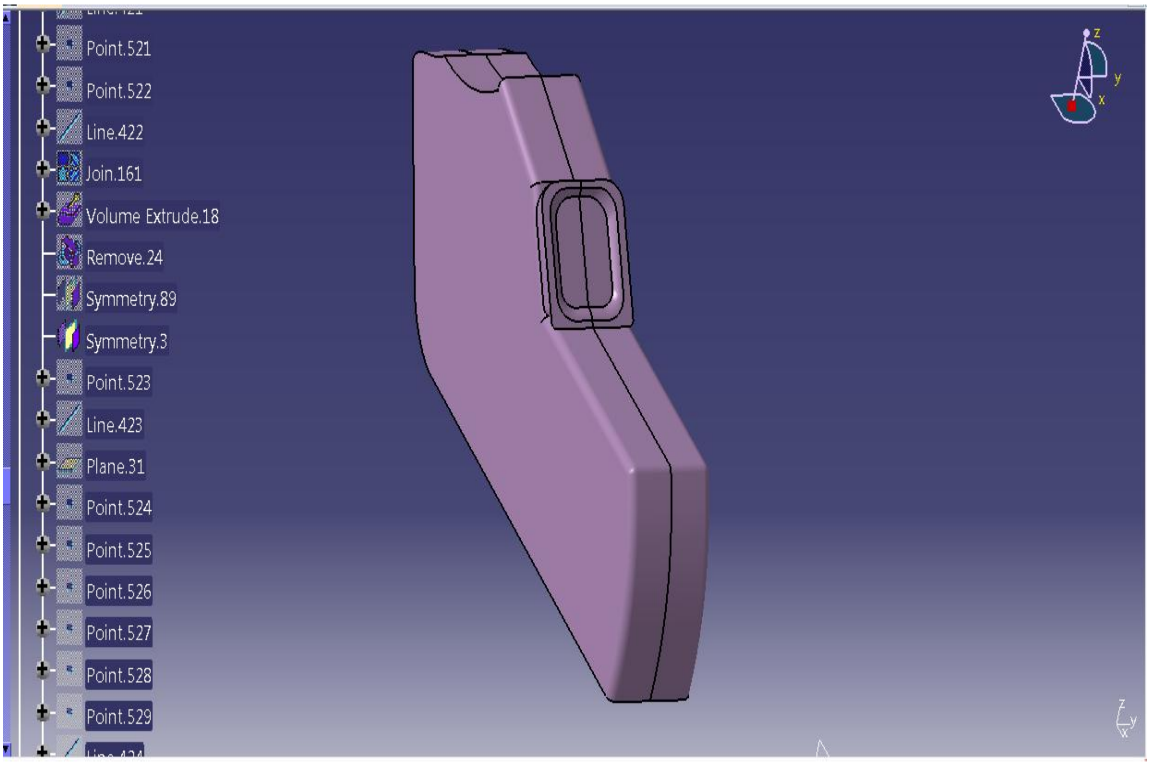
Picture 2.5.28: Creation of screen's transitive surface.



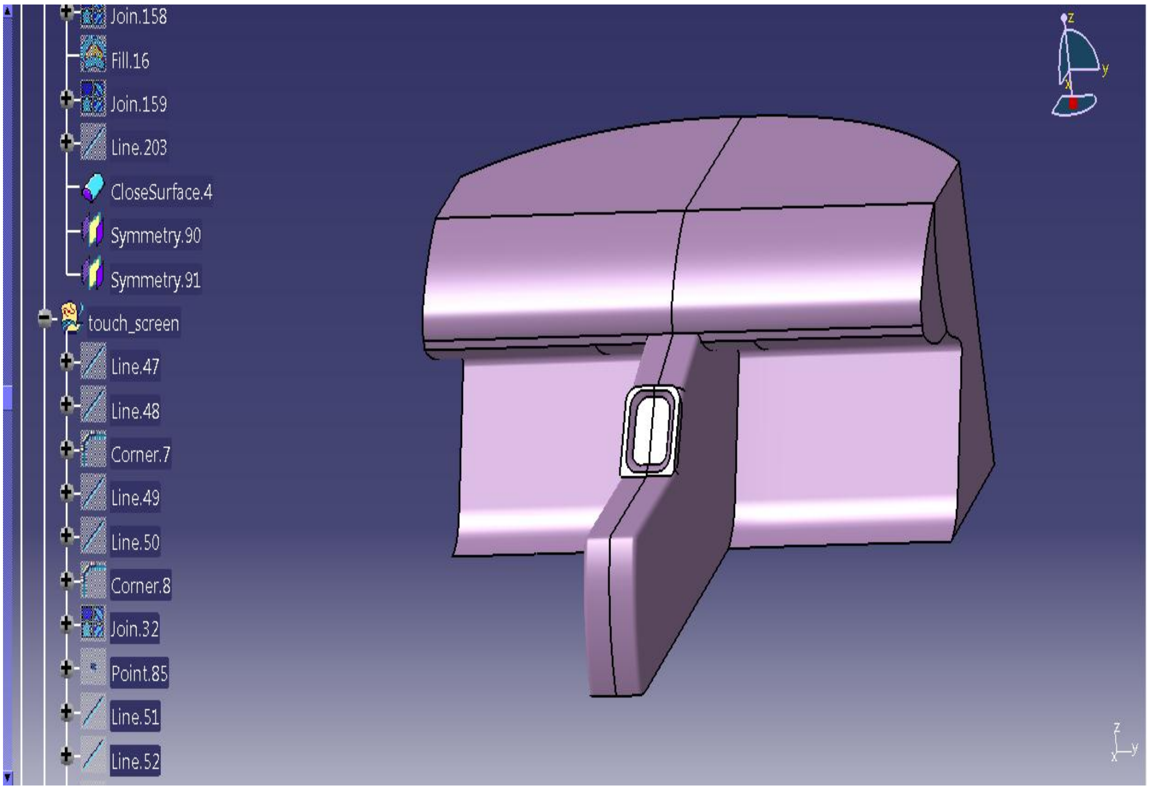
Picture 2.5.29: Touch screen's first symmetrical half solid volume.



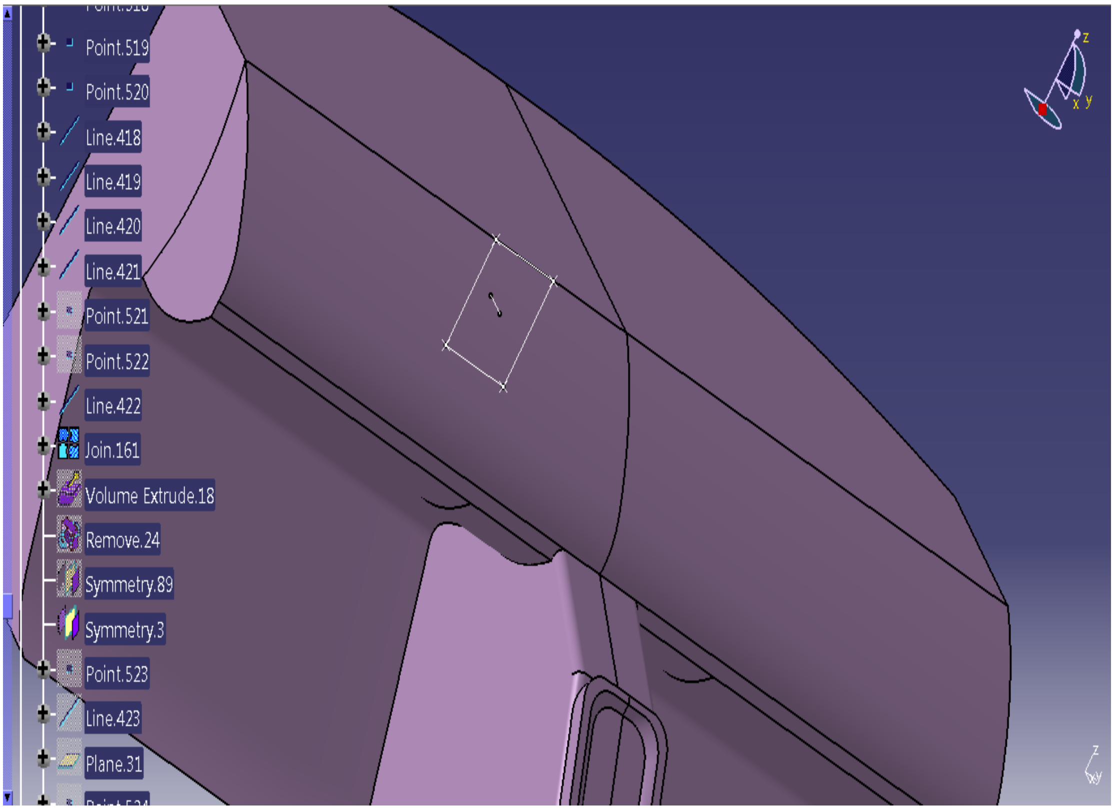
Picture 2.5.30: Touch's screen solid volume – second symmetrical volume.



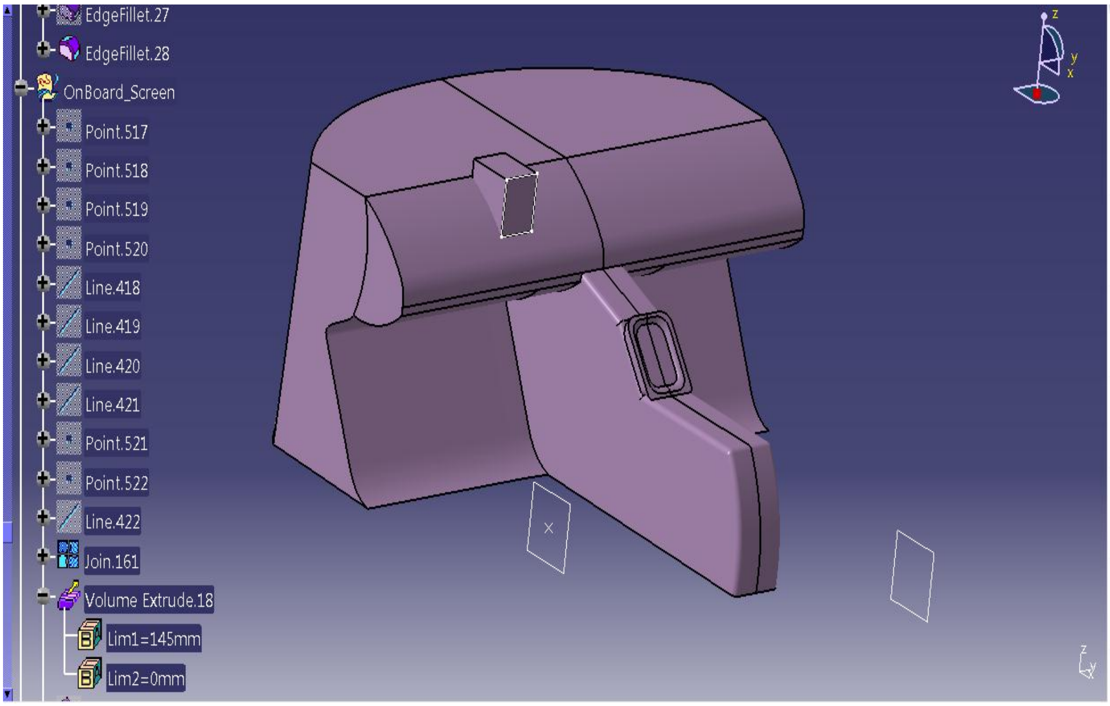
Picture 2.5.31: Touch's screen solid volume – second symmetrical volume.



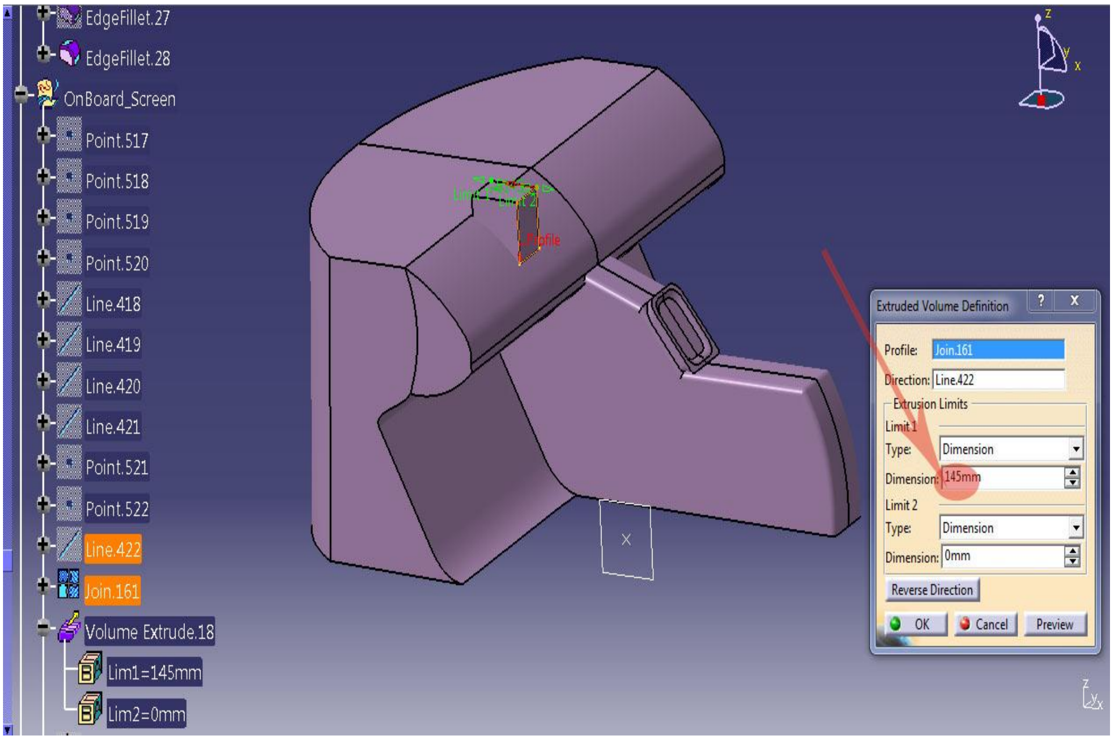
Picture 2.5.32: Dashboard with touch screen attached.



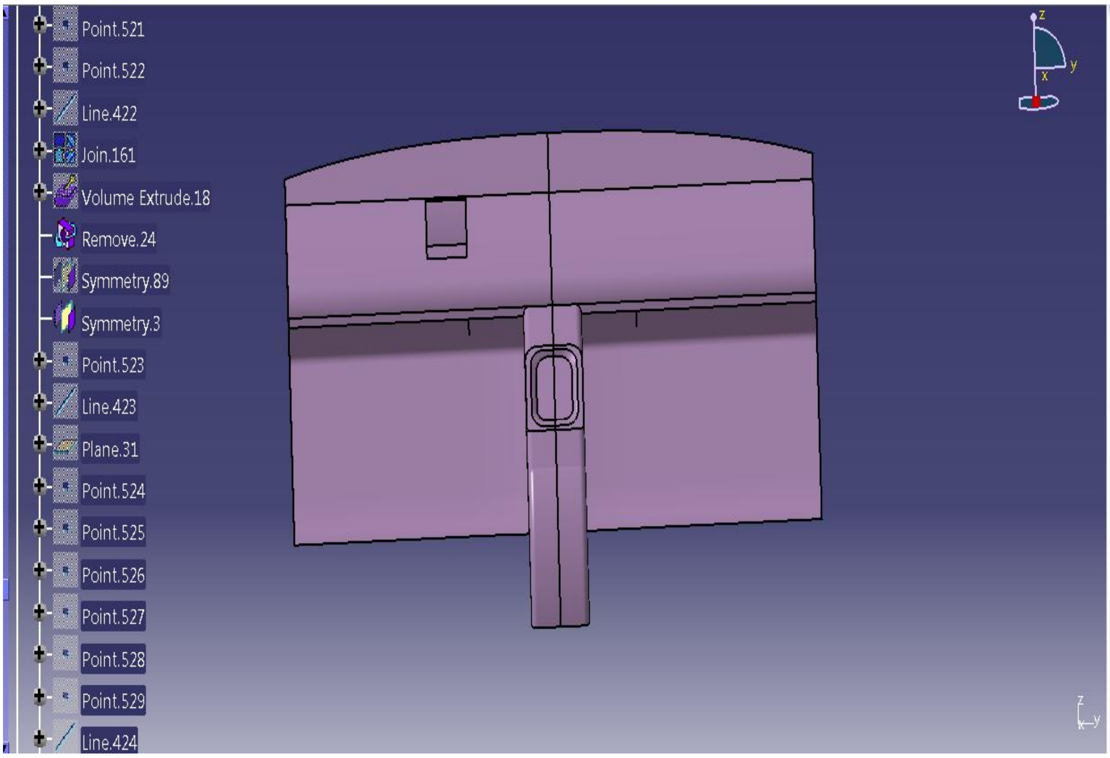
Picture 2.5.33: Dashboard's auxiliary volume wireframe.



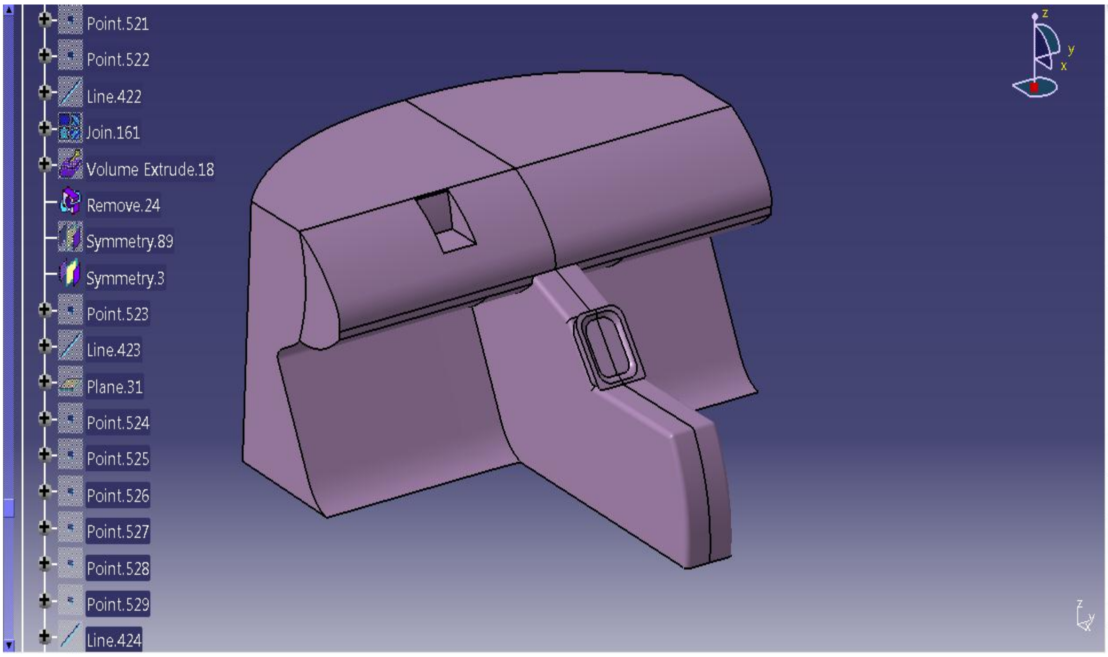
Picture 2.5.34: Auxiliary volume definition.



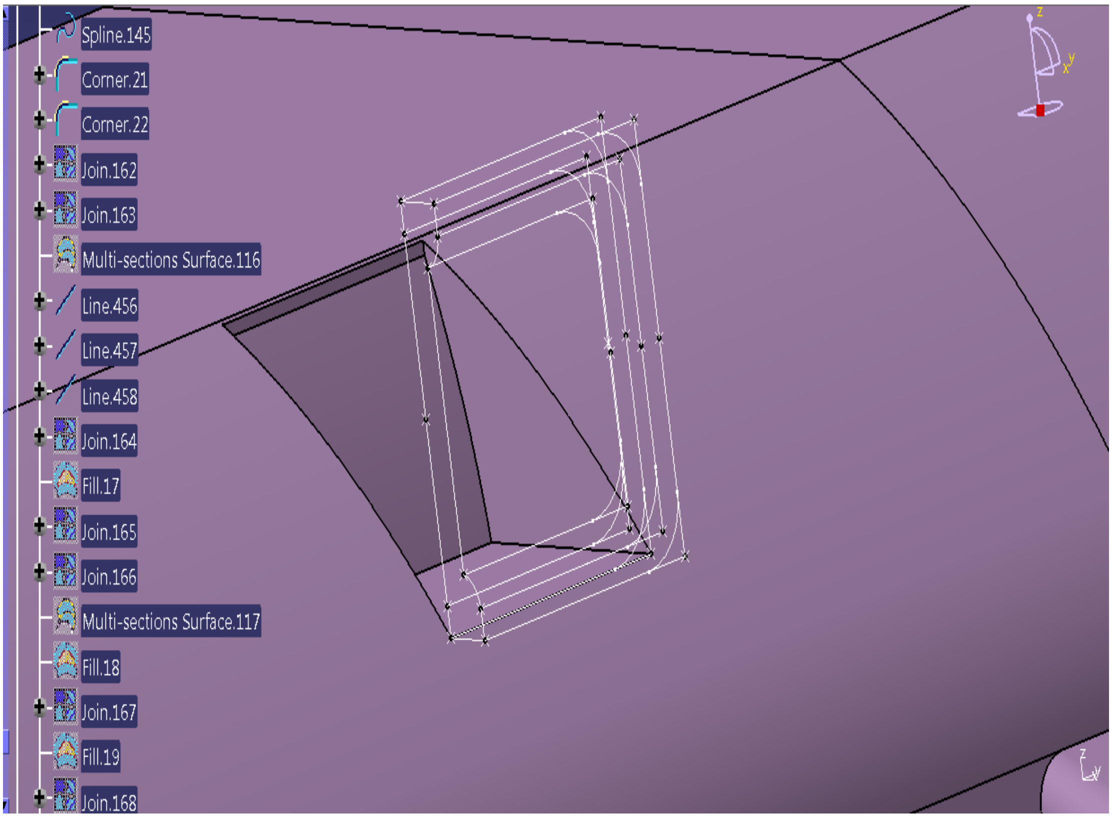
Picture 2.5.35: Auxiliary volume definition.



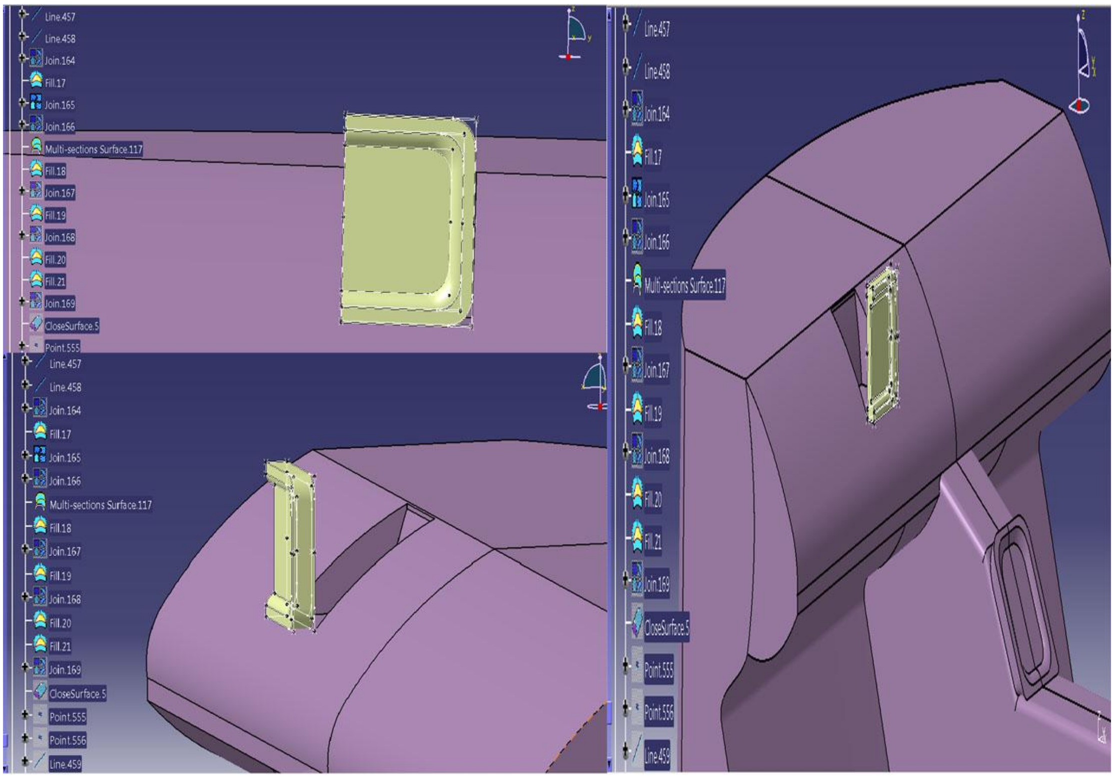
Picture 2.5.36: On board screen area of attachment.



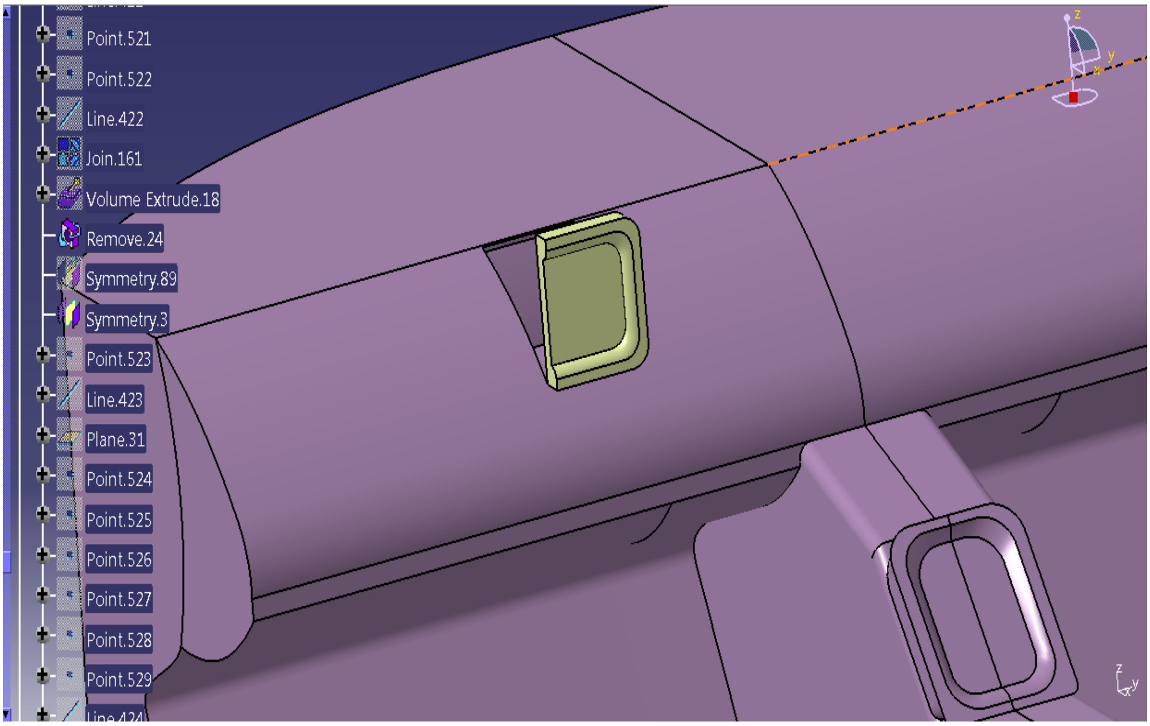
Picture 2.5.37: On board screen area of attachment.



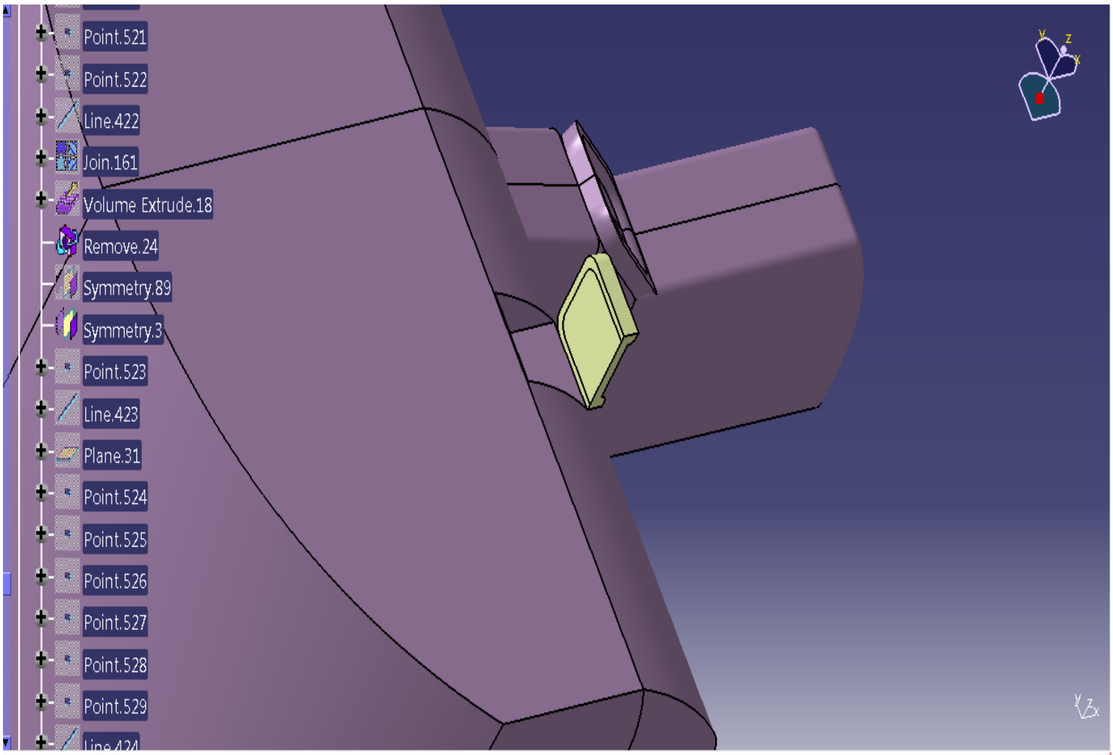
Picture 2.5.39: On board screen wireframe model.



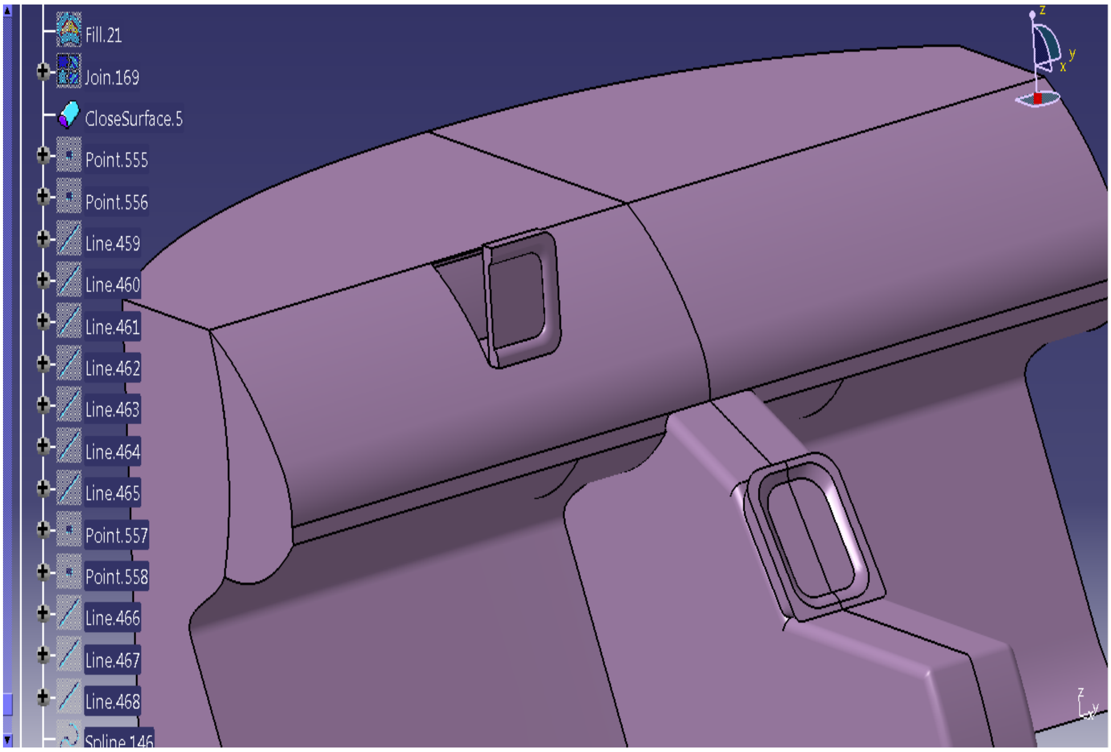
Picture 2.5.40: Creation of screen's surface I.



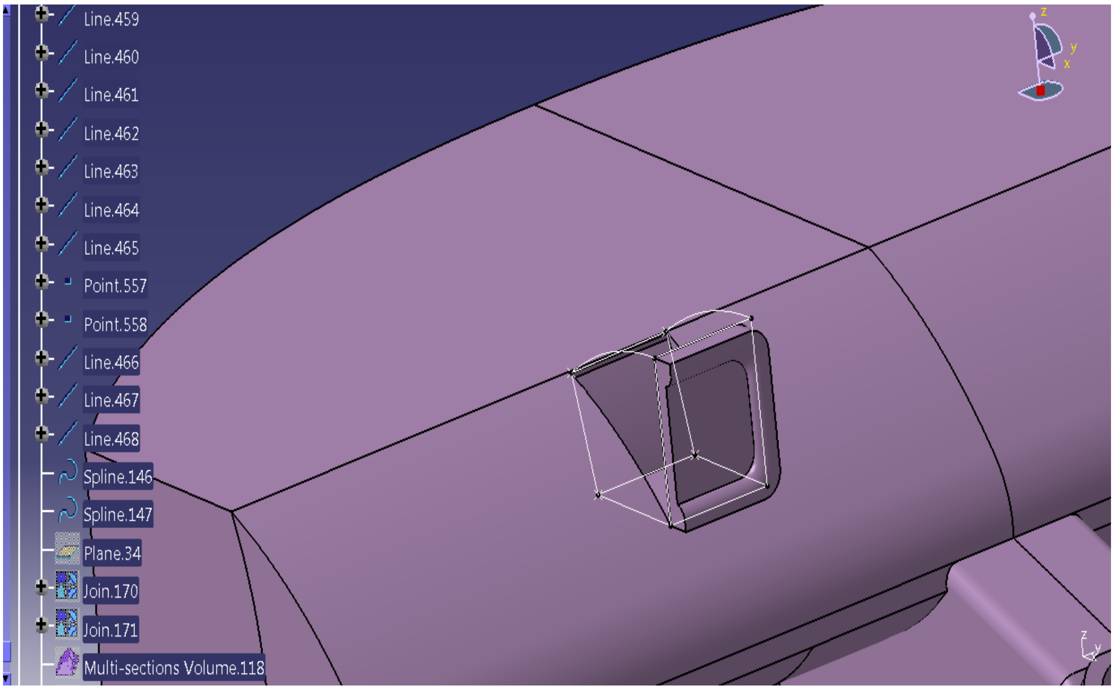
Picture 2.5.41: Creation of screen's surface II.



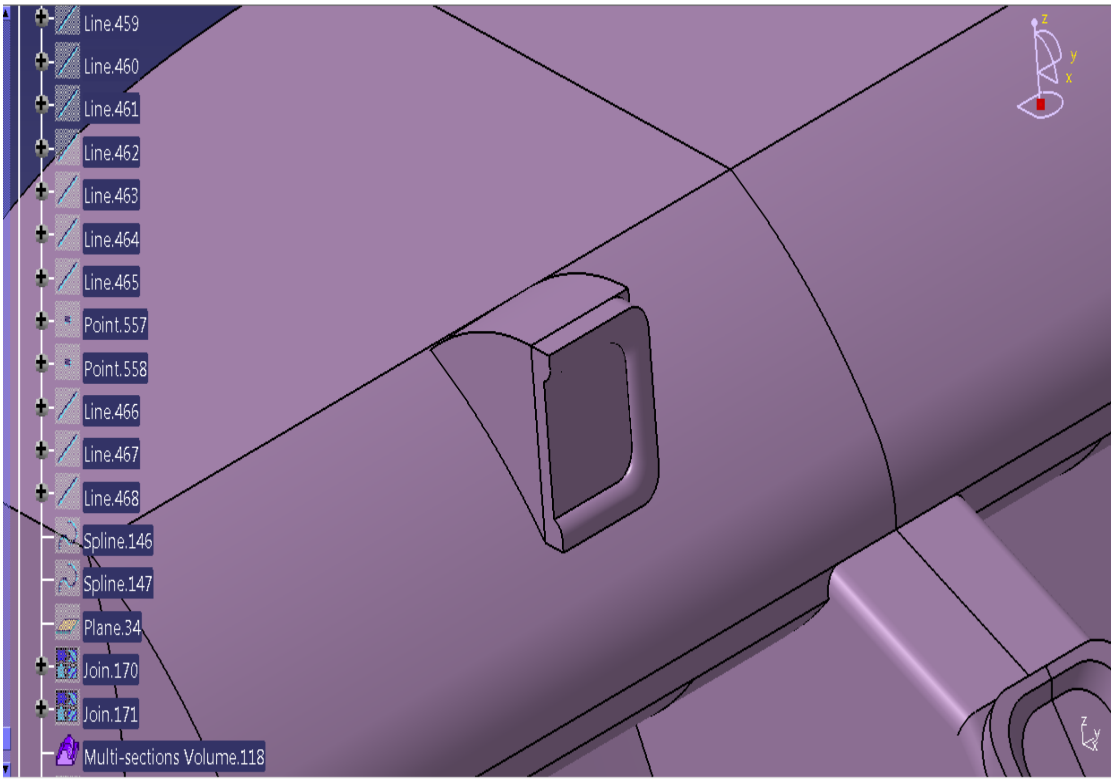
Picture 2.5.42: Formation of screen's surface III.



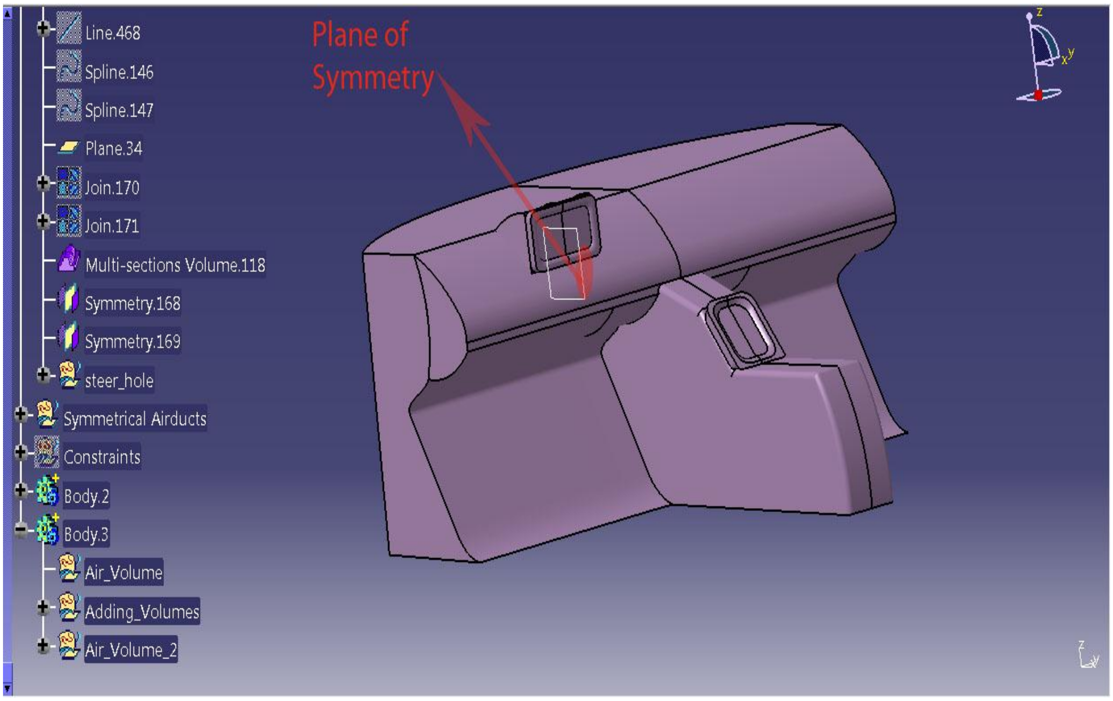
Picture 2.5.43: Dashboard's screen volume.



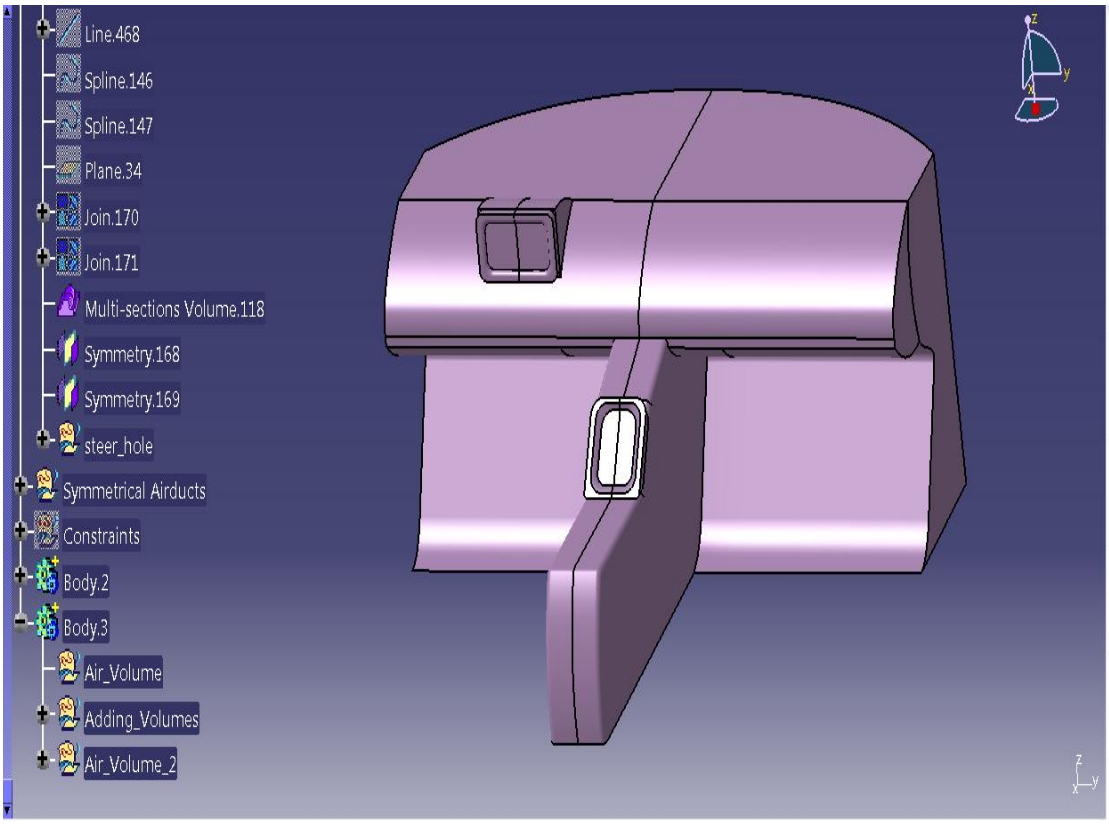
Picture 2.5.44: Connective volume wireframe.



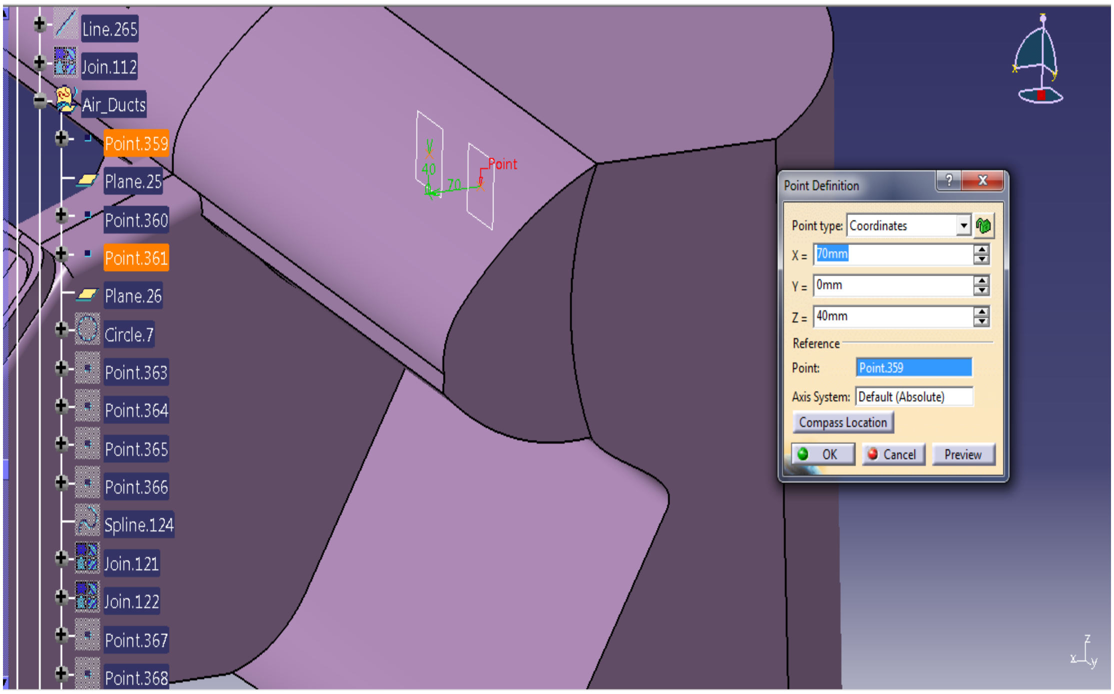
Picture 2.5.55: Formation of connective volume.



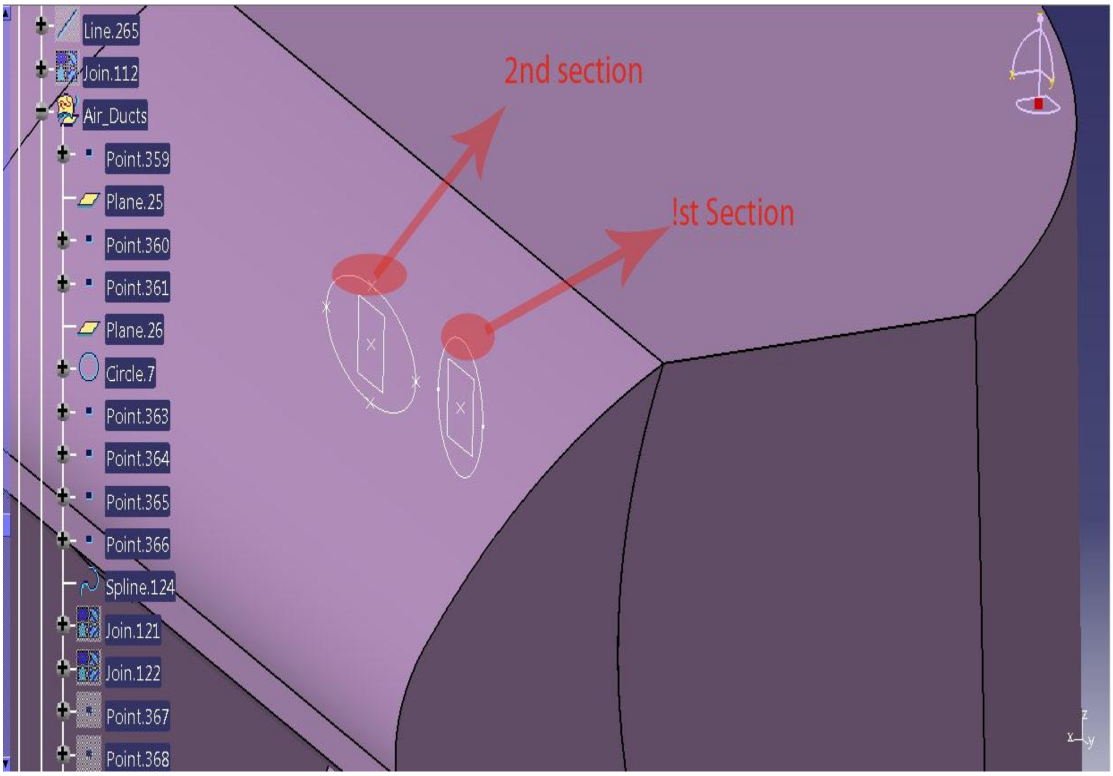
Picture 2.5.56: Plane of symmetry definition.



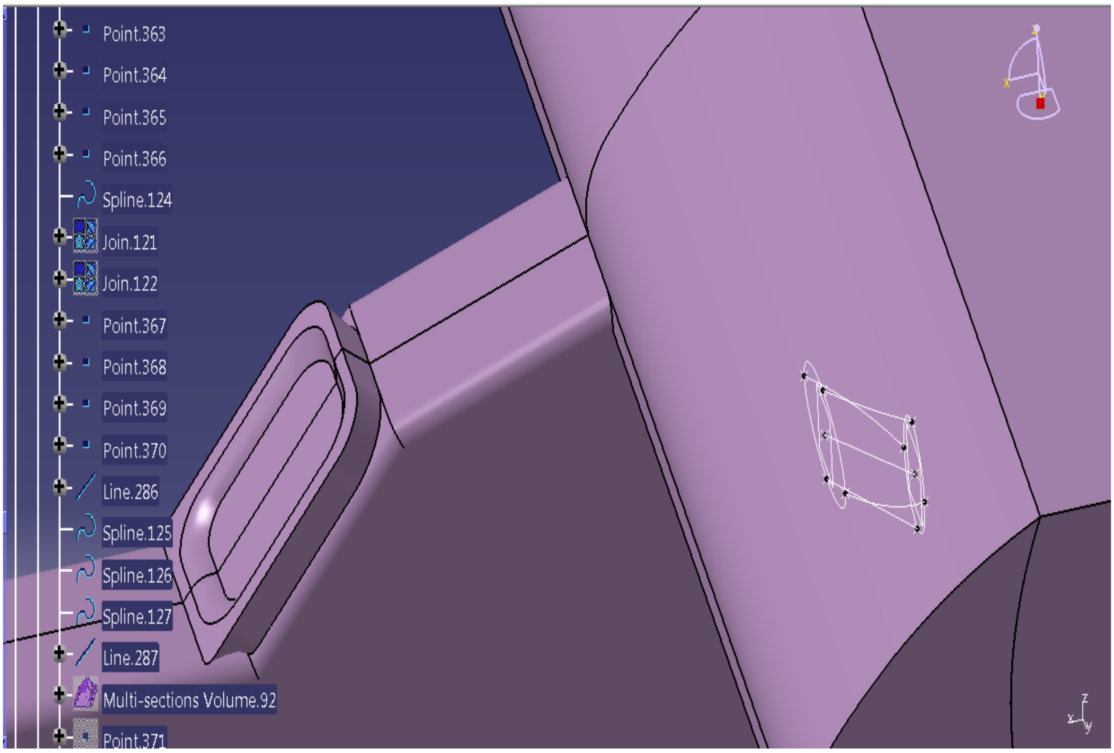
Picture 2.5.57: Dashboard's solid volume.



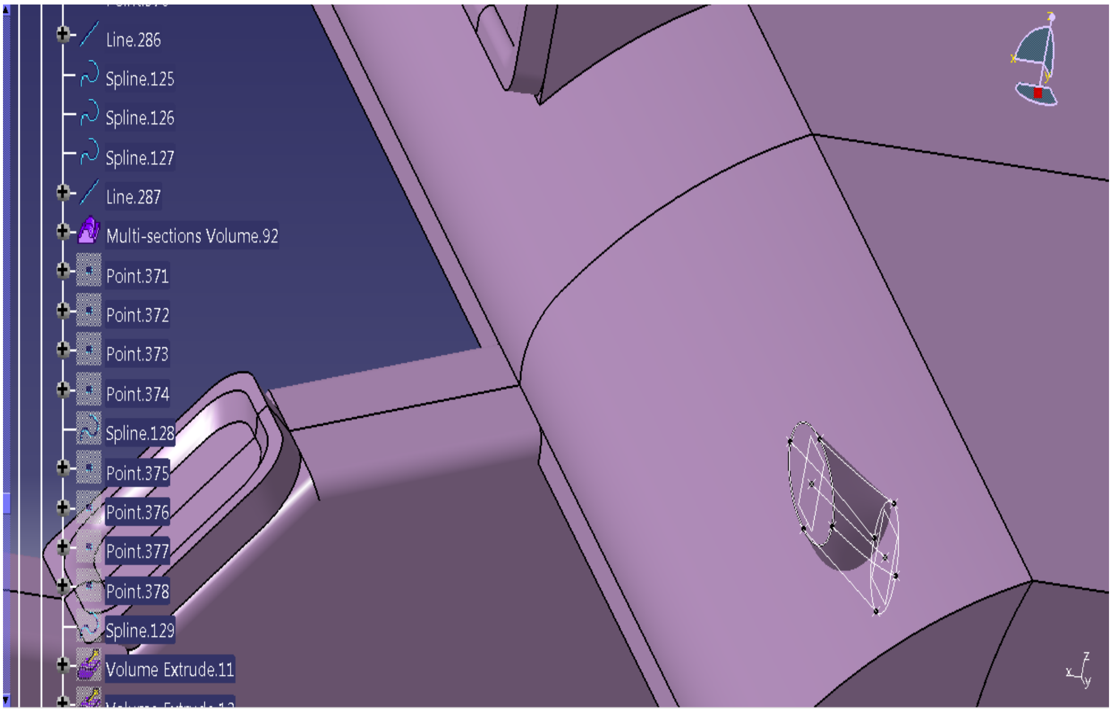
Picture 2.5.58: Dashboard's air-ducts.



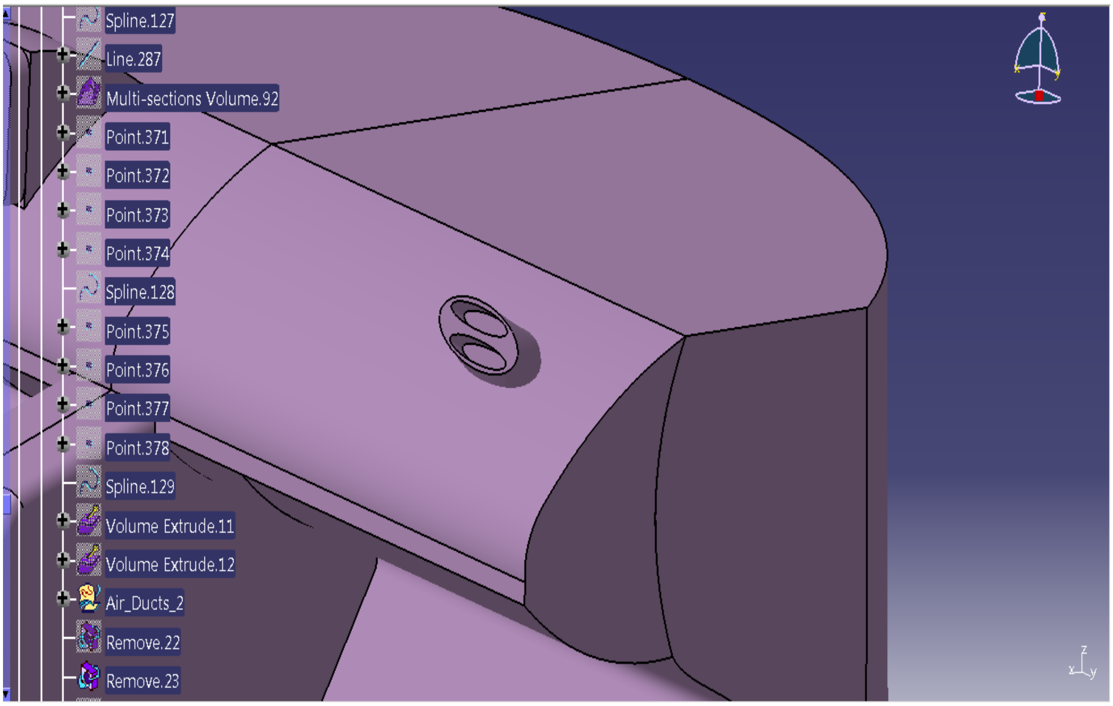
Picture 2.5.59: Air-duct's sections wireframe model.



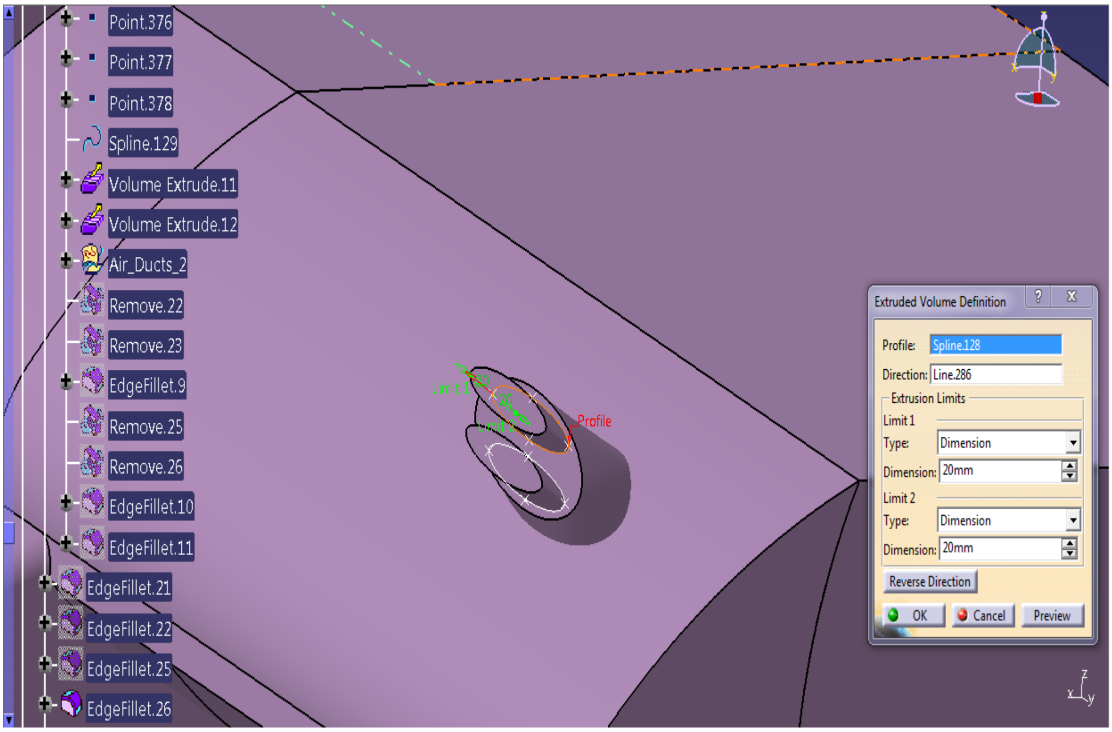
Picture 2.5.60: Air-duct' s guide curves.



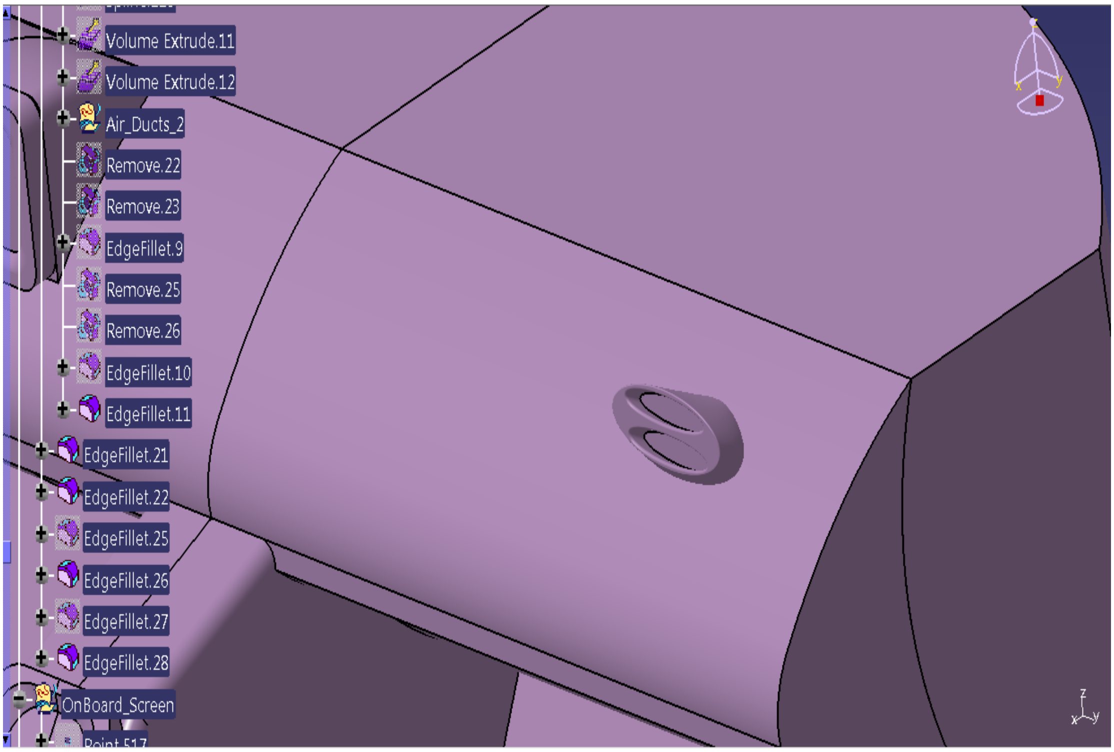
Picture 2.5.61: Air-duct's solid volume.



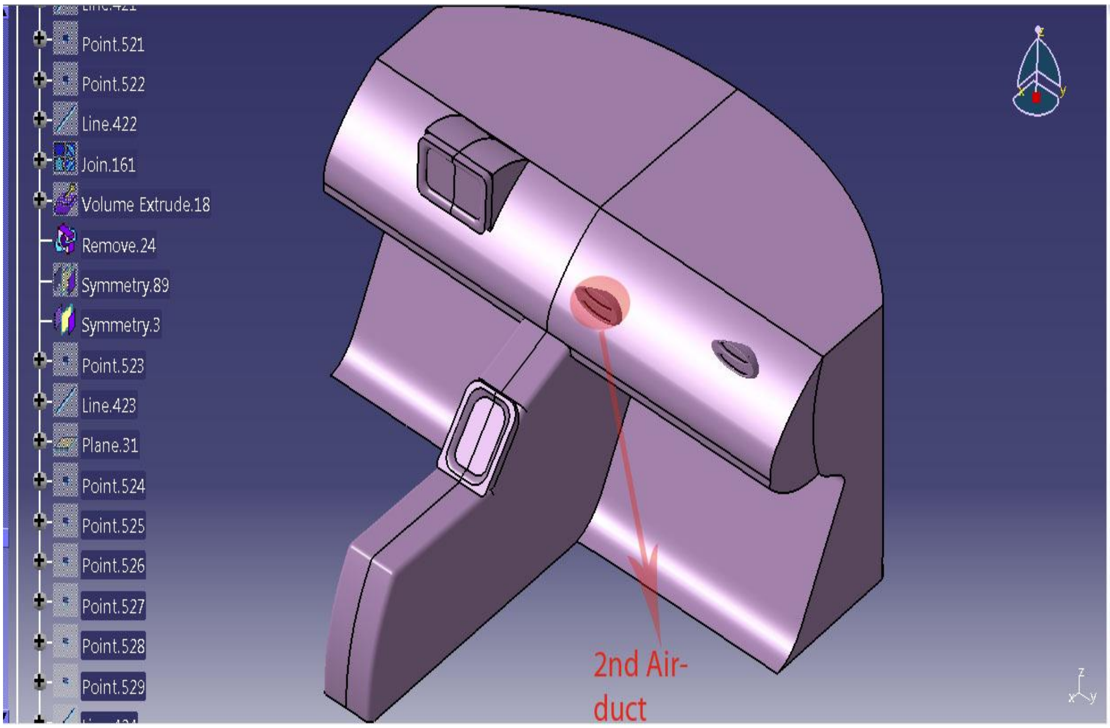
Picture 2.5.62: Air-duct's solid volume with air outlets.



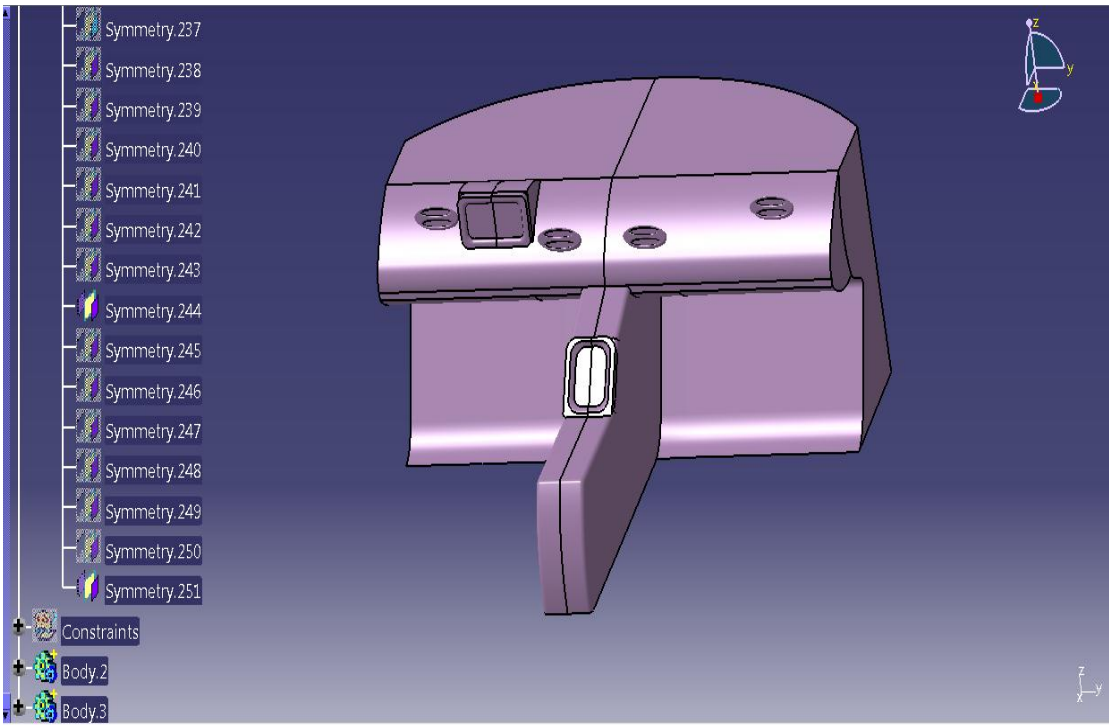
Picture 2.5.63: Air-duct auxiliary volume of extrusion definition.



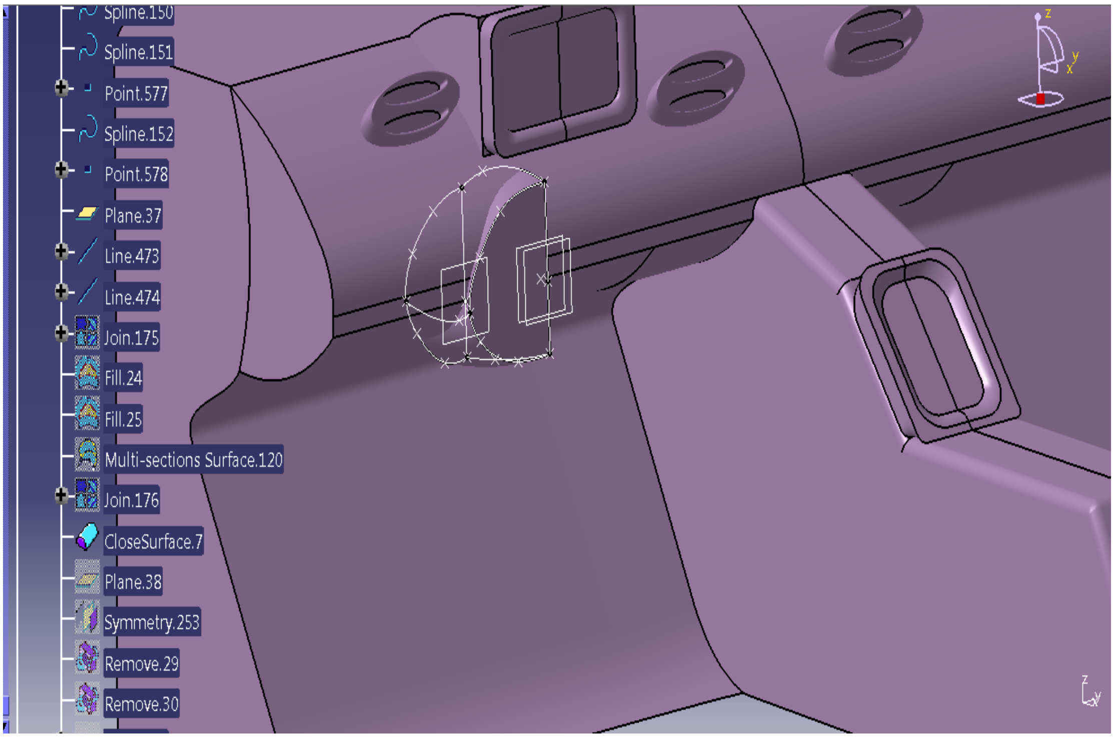
Picture 2.5.64: Air-duct volume with edge filletings.



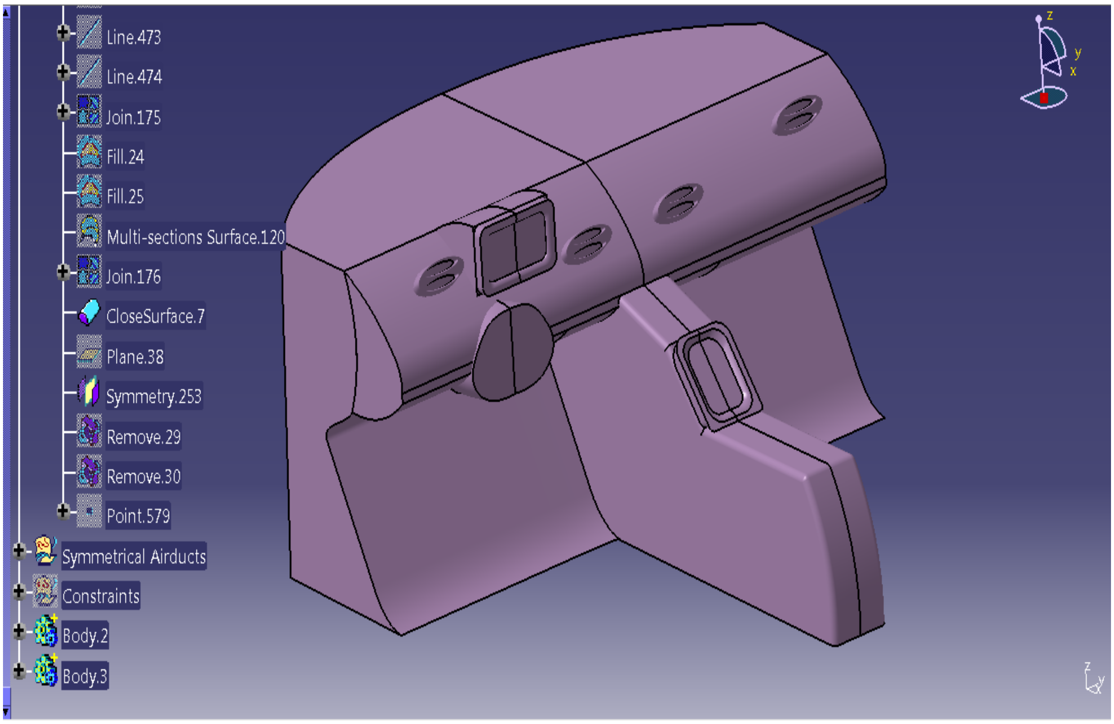
Picture 2.5.65: 2nd Air-duct volume.



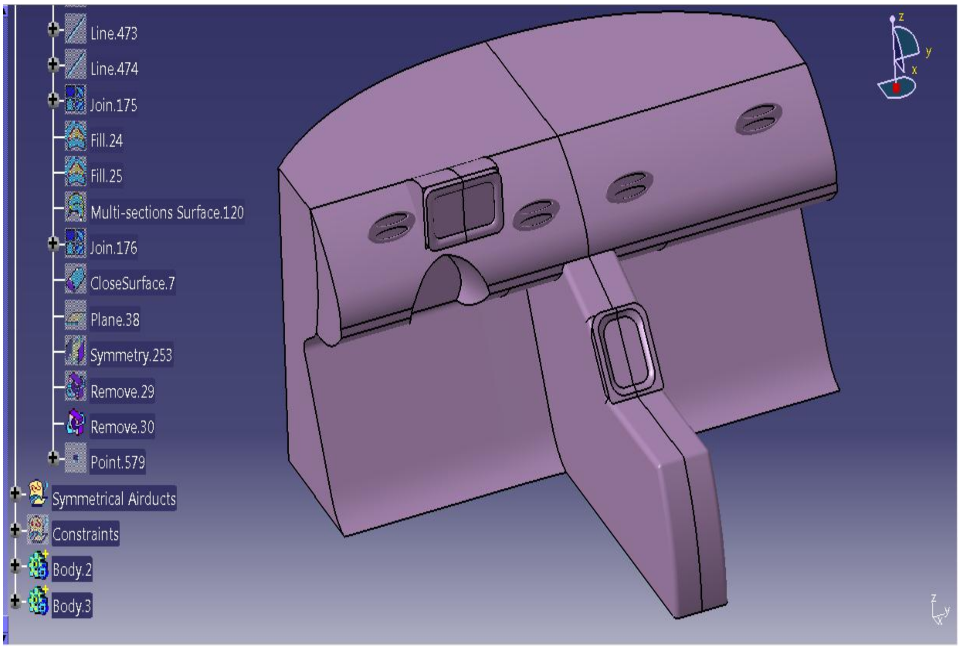
Picture 2.5.66: Air-ducts symmetrical volumes.



Picture 2.5.67: Volume definition for implementation of “*remove volume*” command.



Picture 2.5.68: Creation of auxiliary volume.



Picture 2.5.69: Hole for steering wheel's part attachment.

The dashboard's 3-d model dimensions were calculated upon an actual dashboard of an existing urban car. The primary dimensions that were determined were the total height of the dashboard, the front face length and the length of the board's console. The width of the dashboard, was emerged from the final assembly of the driver's cabin with the vehicle's volume, therefore there was no need to be calculated.

The dimensions were depicted at the table below.

Table 2.5.1

Total Height	660 mm
Front Face Length	174 mm
Console Length	770 mm

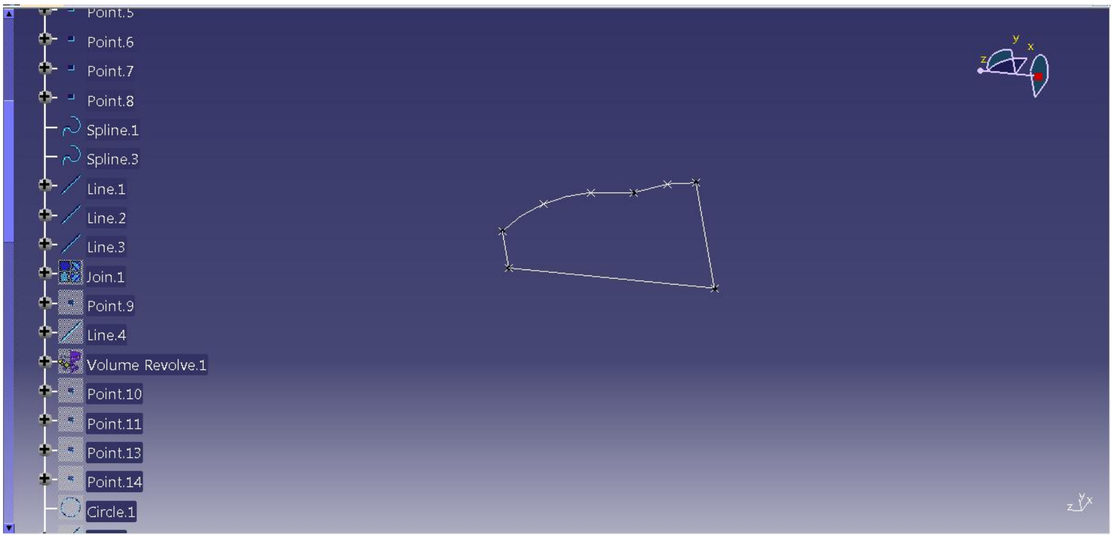
2.6 INDICATOR LIGHT LEVER DESIGN

The indicator light lever part was derived as an assembly of five sub-parts, that were connected together, in Catia's assembly workbench. The shape of the final geometry was derived both from pictures of an actual's indicator light lever, which was already integrated inside a private car and from pictures from the internet. A notable remark, at this certain point, is that there wasn't need for such a detailed solid geometry for the indicator lights lever volume, at least from the scope of functionality. The detailed design of the indicator light lever was derived from shear passion and desire to create a part that fulfills very high standards, both on functional and aesthetic scope.

2.6.1 PART ONE- INDICATOR LIGHT LEVER HEAD

The first part of the lever's assembly, was a ''revolved'' solid geometry. The wireframe profile that was rotated by a symmetrical axis, was defined by creating two Splines and three lines, such as deriving a closed section (Picture 2.6.1.1). The revolution axis was initiated for the creation of axisymmetric geometry. As revolution axis was defined a line (line 4 specifically) that intersects the wireframe's profile (Picture 2.6.1.2). The final solid volume was constructed by using the "*revolve volume*" command. As profile inside the command's definition window, the section's wireframe was selected. The profile was revolved around the axis of revolution for 360 degrees (Picture 2.6.1.3).

For reasons of both completion and aesthetics, a number of grips were created upon the solid geometry. The wireframe model was first designated as shown upon the Picture 2.6.1.4 below. This wireframe was consisted of two sections. These sections were designed upon two planes perpendicular to each other. The two sections were united together with a B-Spline curve. The solid volume was created by using the "*multi-section volume*" creation command. As generative curves were selected the two sections from the wireframe model and as guides the Spline curve that connected the two sections (Picture 2.6.1.5). Finally, by using the angular pattern command, eleven more solid grips were created upon the revolved volume of the part. The angular space between each grip volume, was 27 degrees (Picture 2.5.1.6). Finally, some edge fillets were inserted upon the sharp edges of the solid volume (Picture 2.6.1.7).



Picture 2.6.1.1: First part of indicator's light lever – wireframe profile.

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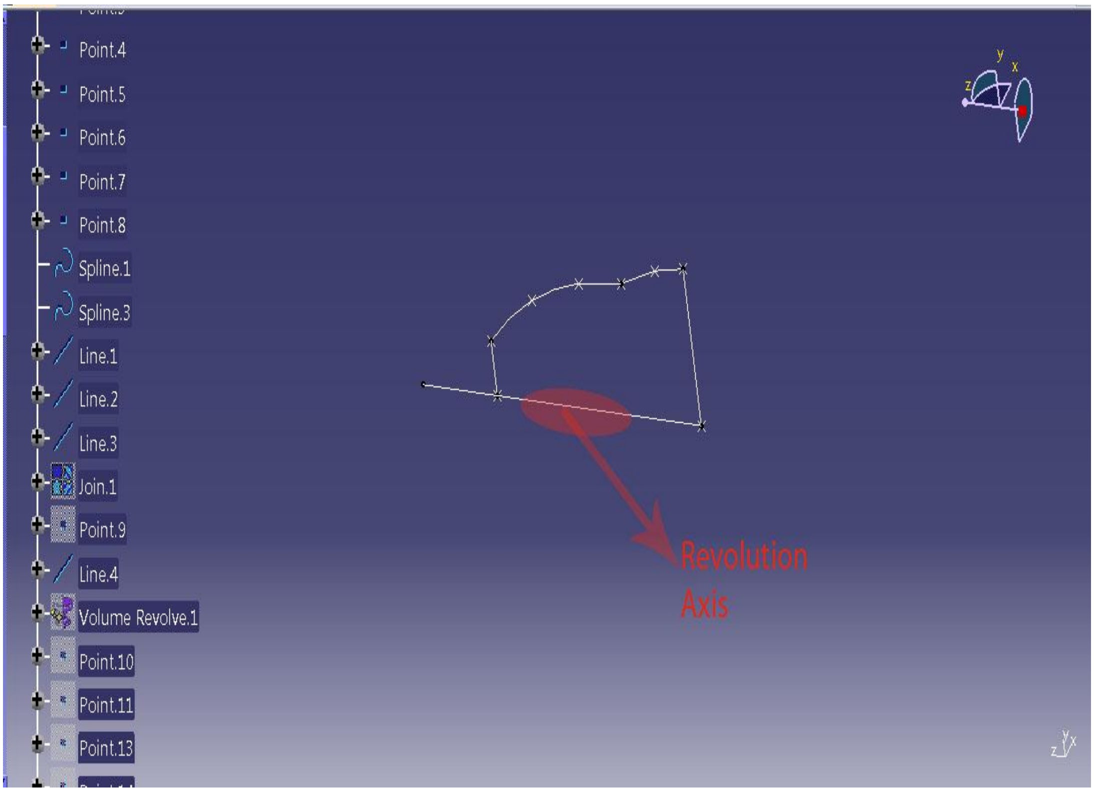
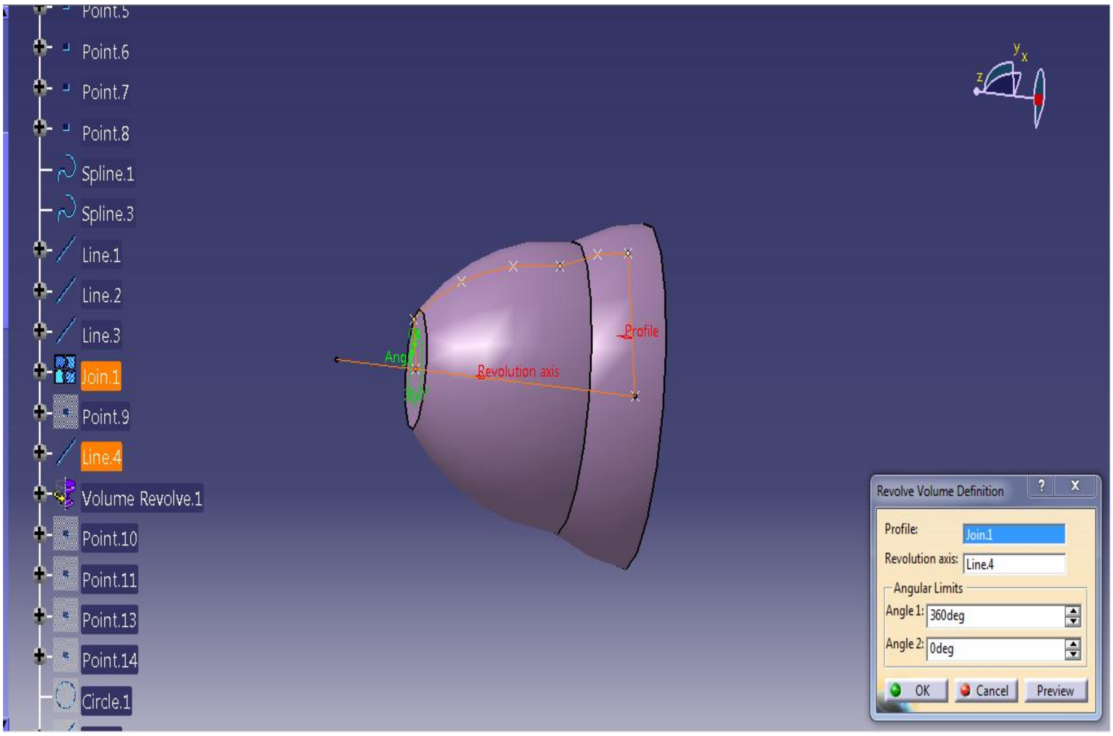
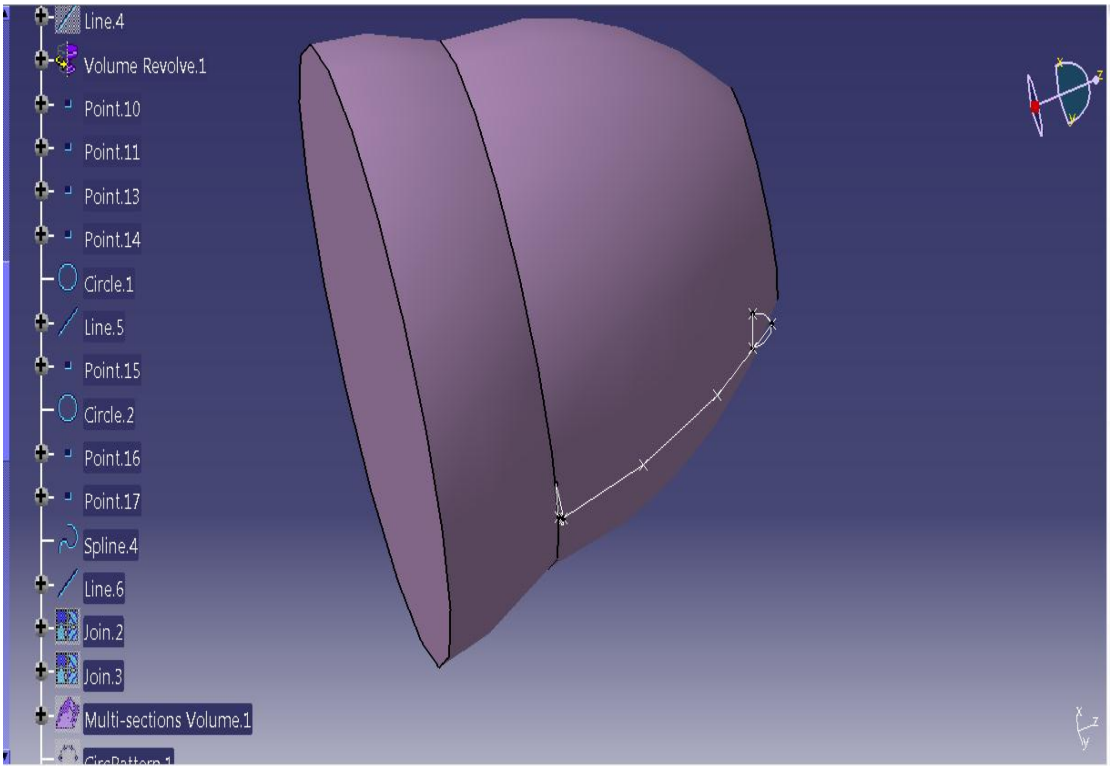


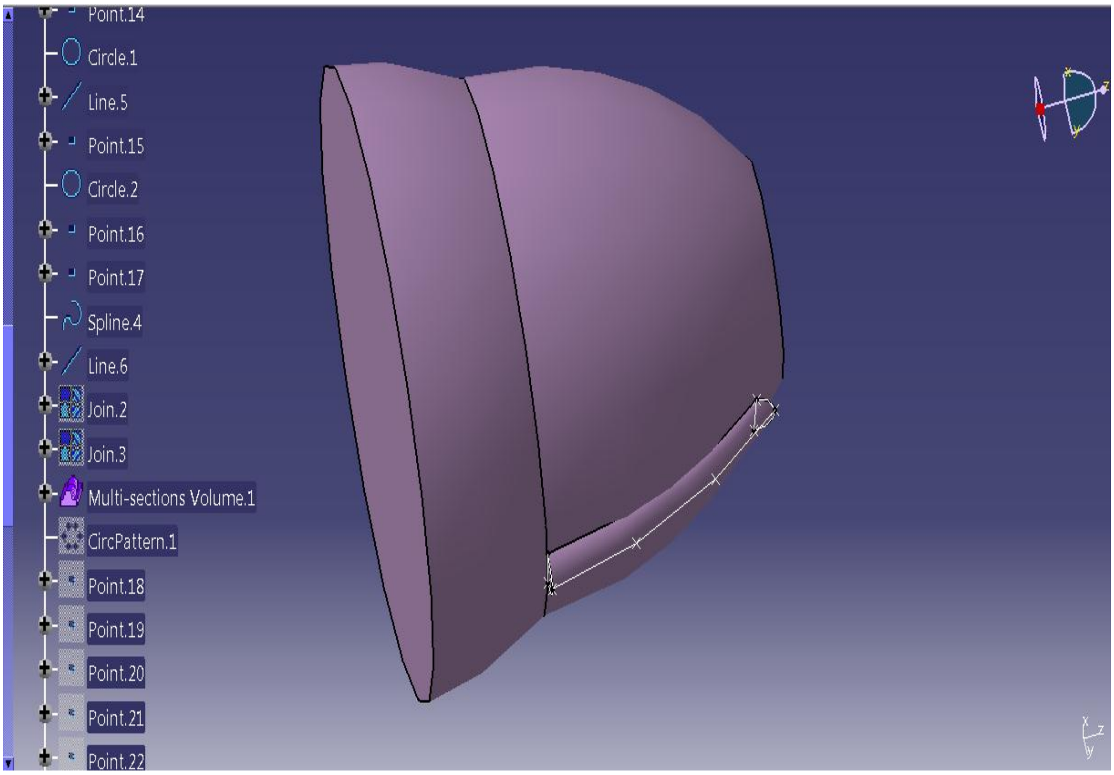
Picture 2.6.1.2: Definition of axis of revolution.



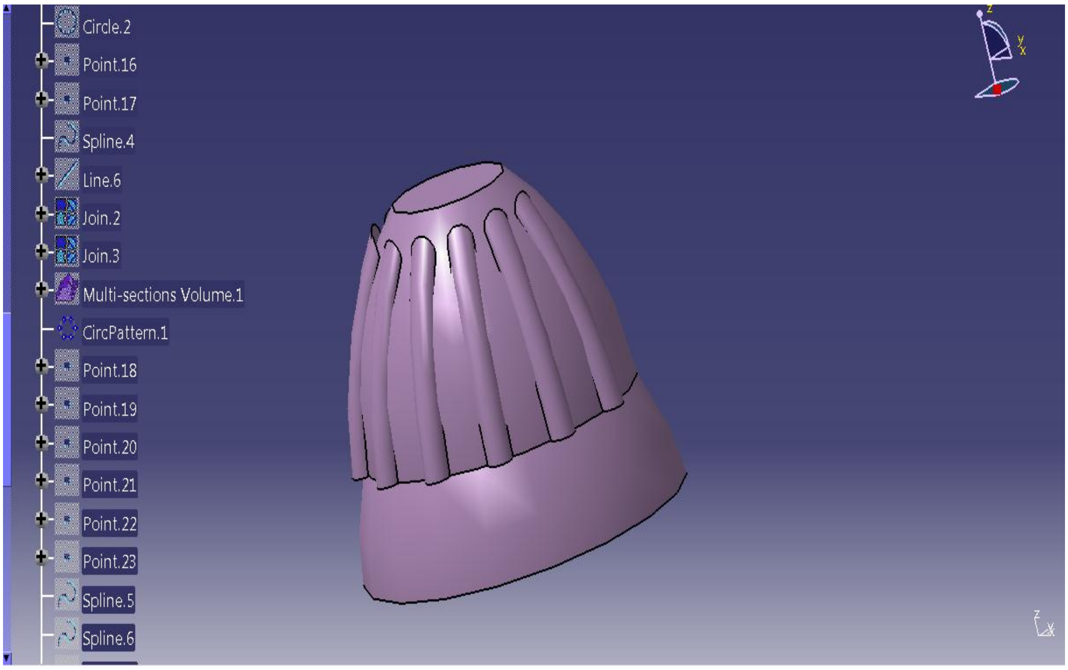
Picture 2.6.1.3: Revolved Solid Geometry



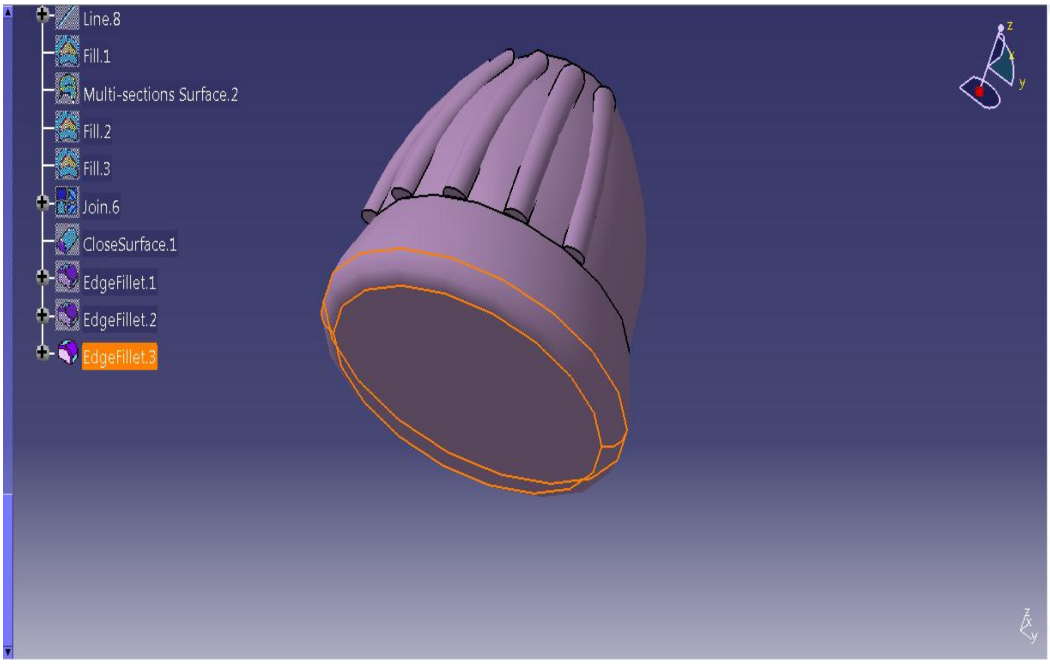
Picture 2.6.1.4: Grip's wireframe model.



Picture 2.6.1.5: Grip's solid volume.



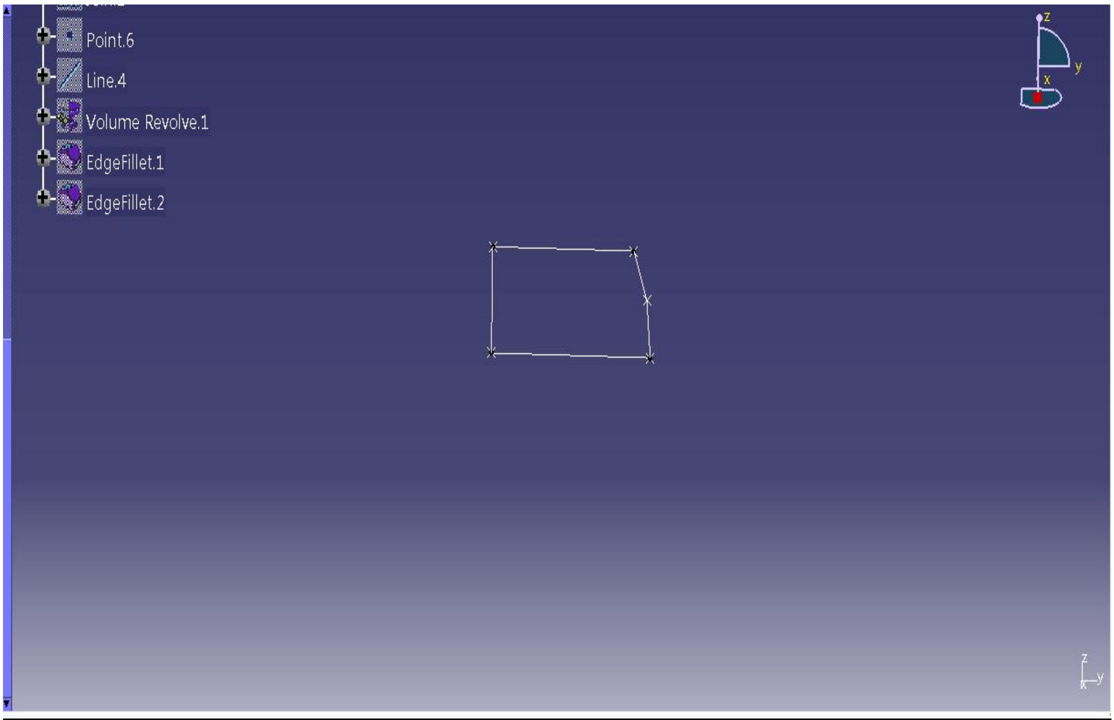
Picture 2.6.1.6: Circular pattern of the initial grip volume.



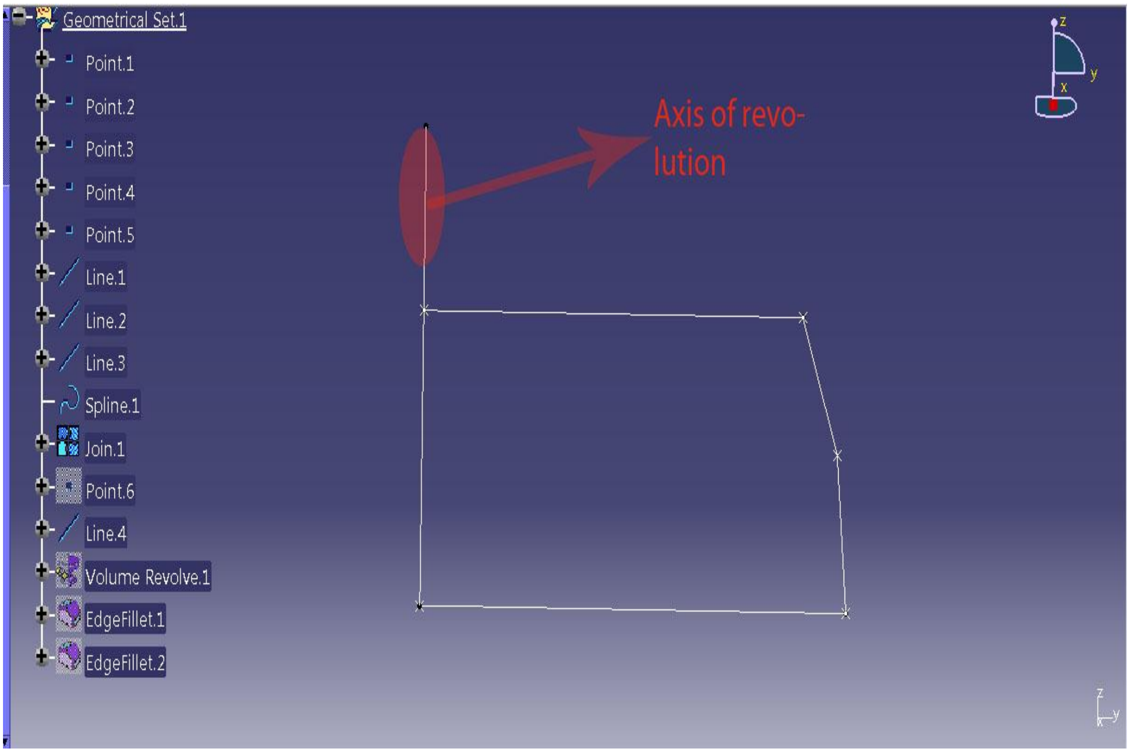
Picture 2.6.1.7: Final solid volume of part one with edge filletings.

2.6.2 DESIGN OF INDICATOR'S LIGHT LEVER SECOND PART

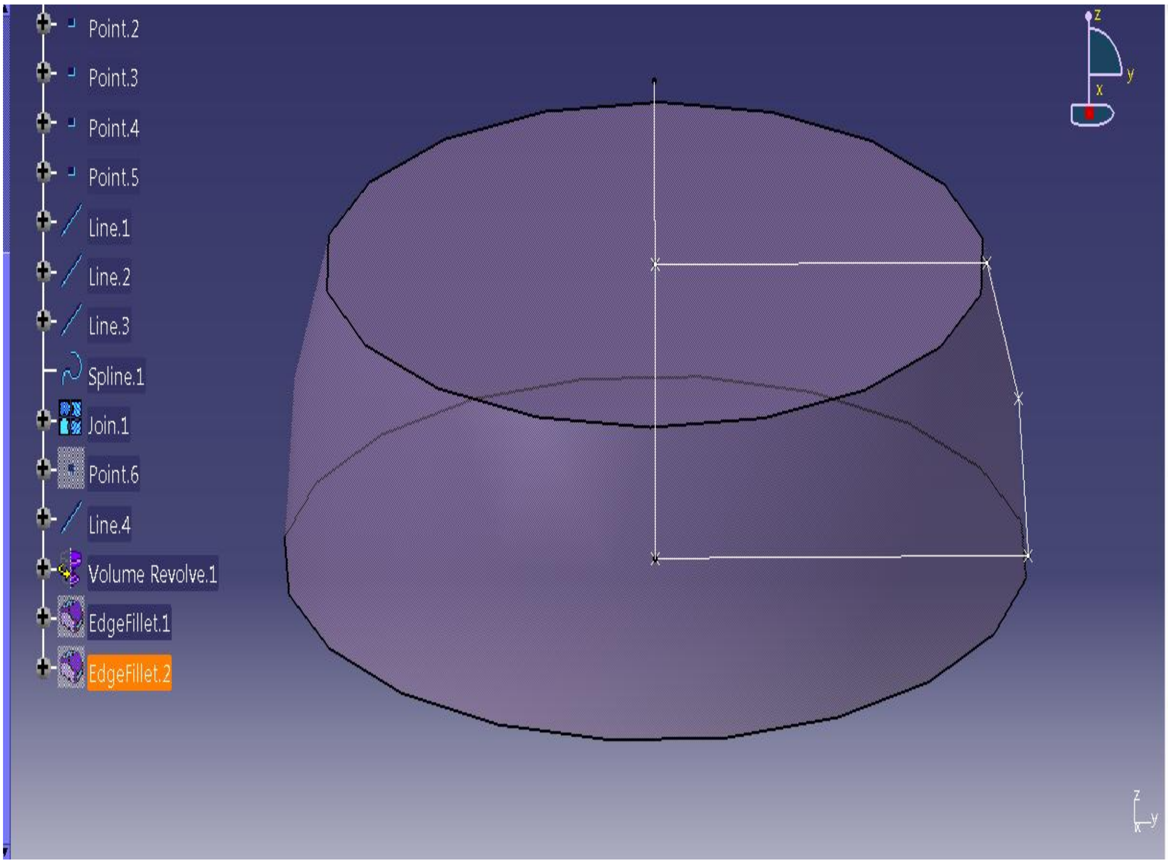
The second volume of indicator's light lever was also a revolved solid part. So, at first the wireframe solid geometry was designed. The closed section that resulted from the wireframe model, consisted of three lines and a B-Spline (Picture 2.6.2.1). After section's wireframe designation, followed a unification of the four curves. The curves were unified by the "*Join*" operation command. Moreover, the line that was acted as the axis of revolution, was initiated (Picture 2.6.2.2). Then the final solid geometry was created with the use of "*revolution*" command. As generative curve the section's wireframe was inserted. This profile was revolved for 360 degrees around the revolution axis (Picture 2.6.2.3). Finally, two filletings were added to the sharp edges of upper and lower surfaces of the part's solid volume (Picture 2.6.2.3- 2.6.2.4).



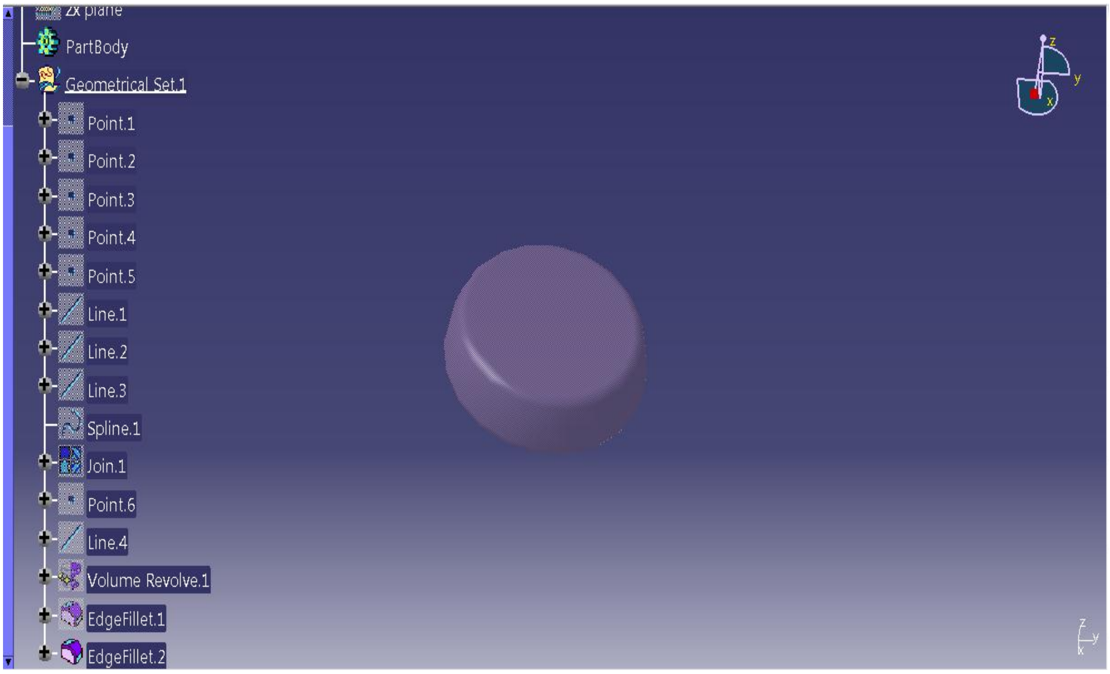
Picture 2.6.2.1: Indicator light lever second part wireframe.



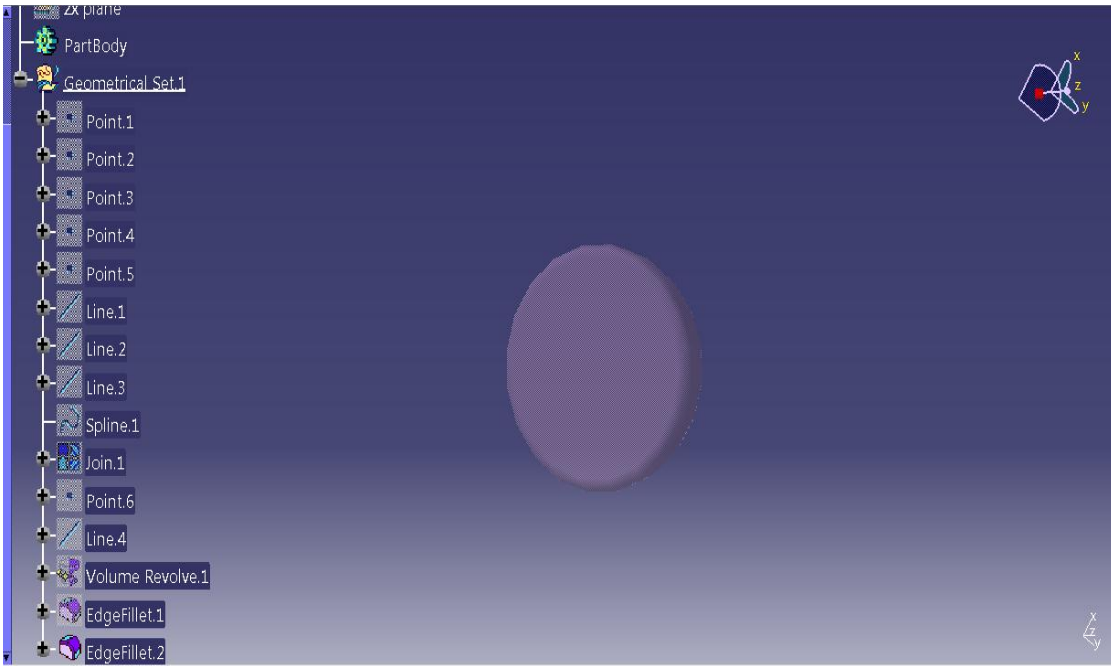
Picture 2.6.2.2: Wireframe's model axis of revolution.



Picture 2.6.2.3: Creation of indicator light lever, second part's solid volume.



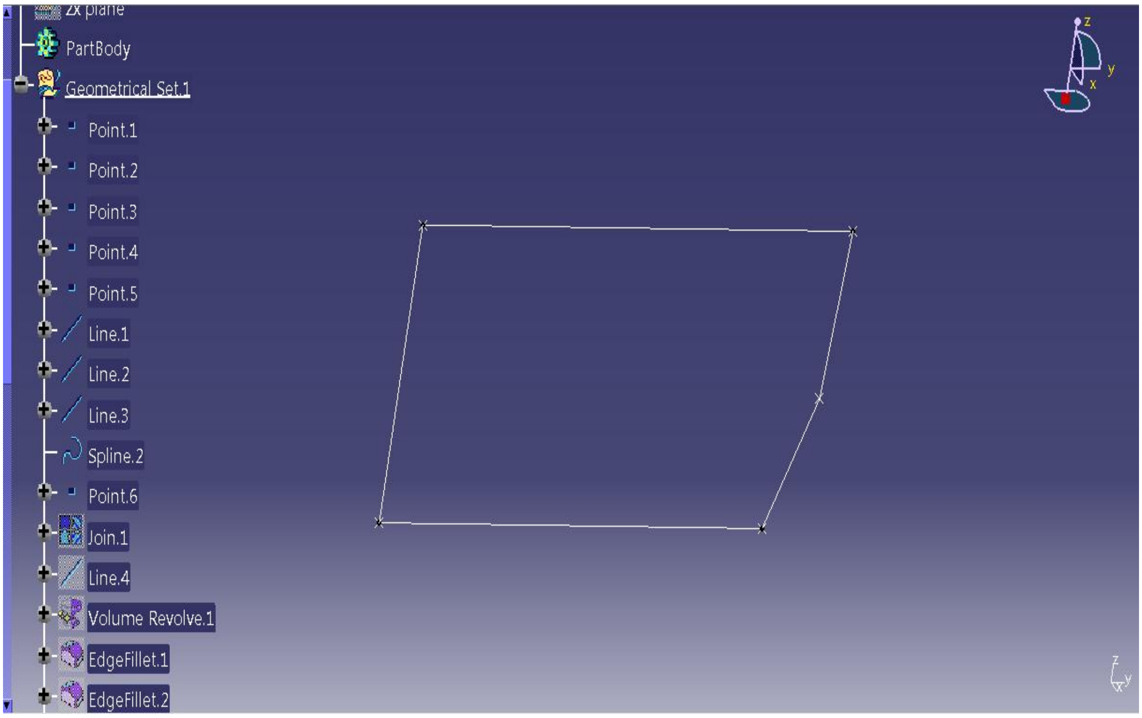
Picture 2.6.2.3: Second part’s volume with edge filletings.



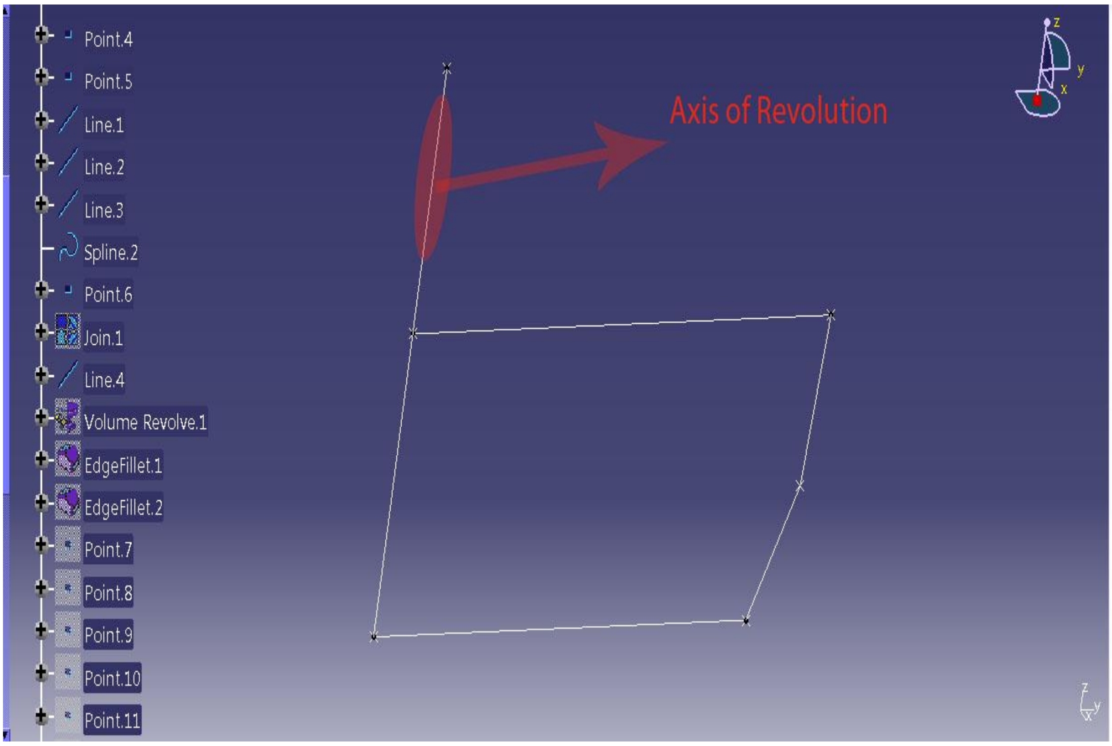
Picture 2.6.2.4: Second part’s volume with edge filletings bottom view.

2.6.3 DESIGN OF INDICATOR LIGHT LEVER THIRD PART

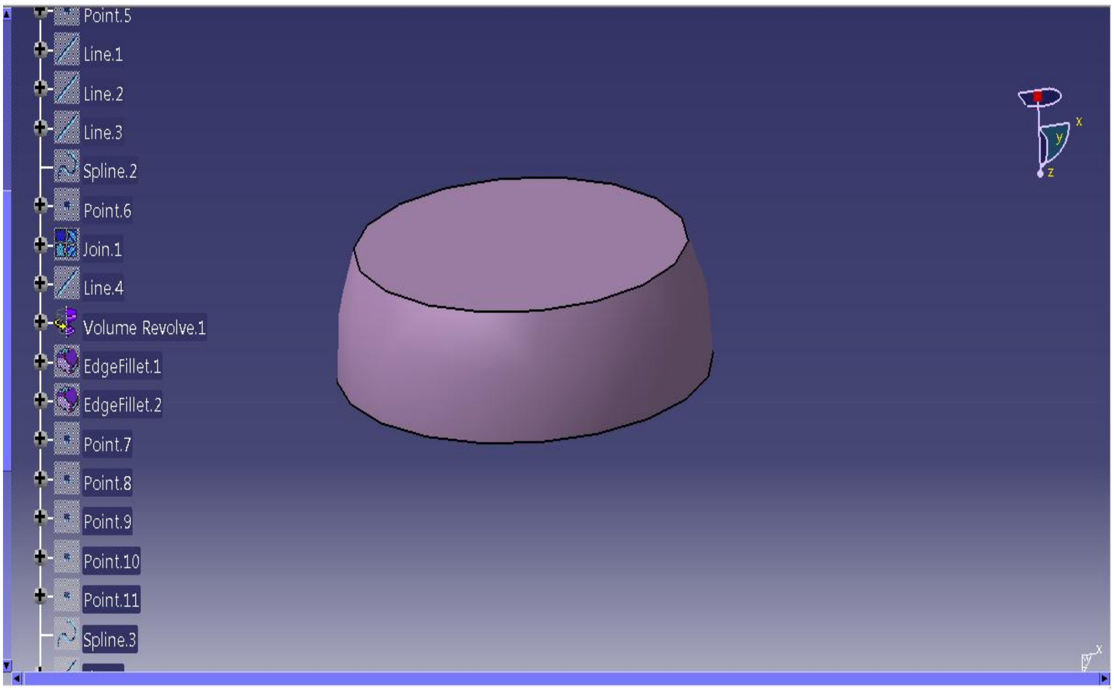
The solid volume of the third part was similar to the second's part solid volume. Similarly as before, the wireframe model was determined first. It was also consisted of three lines and a B-Spline curve such as a closed wireframe section was derived (Picture 2.6.3.1). Next in sequence was the definition of model's axis of revolution. As revolution axis was defined a line, that was intersecting the section's wireframe profile (shown at the Picture 2.6.3.2 below). The part's solid volume was created as a revolution volume. The section was defined as generative curve for the execution of the command and as axis of revolution the line above was used (Picture 2.6.3.3). After the solid volume creation, two additional edge fillets were introduced upon the sharp edges of the volume (Picture 2.6.3.4.). Furthermore, upon the solid part, a number of grips was integrated. At first, the wireframe of the first grip was designed (as show in Picture 2.6.3.5 below). The wireframe model consisted of two sections, perpendicular to the initial volume. The two sections were connected to each other by three B-Spline curves. By using the two sections as generative curves, and the three Splines curves as guide curves, the grip's multi section volume was constructed. Then, with the use of "*circular pattern*" operation, twenty-four copies of the grip's volume were incorporated at the solid geometry. The angular space between each solid grip was fifteen degree (Pictures 2.6.3.6 – 2.6.3.7).



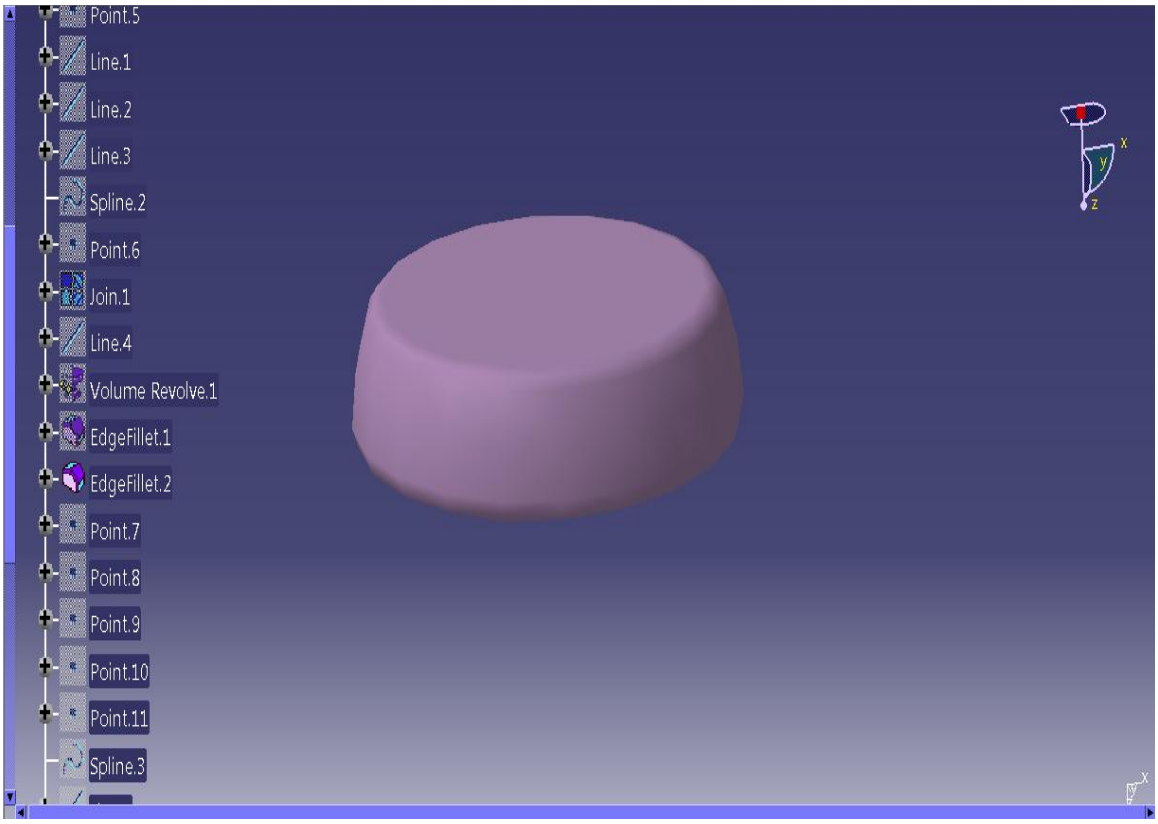
Picture 2.6.3.1: Indicator light lever third part wireframe model.



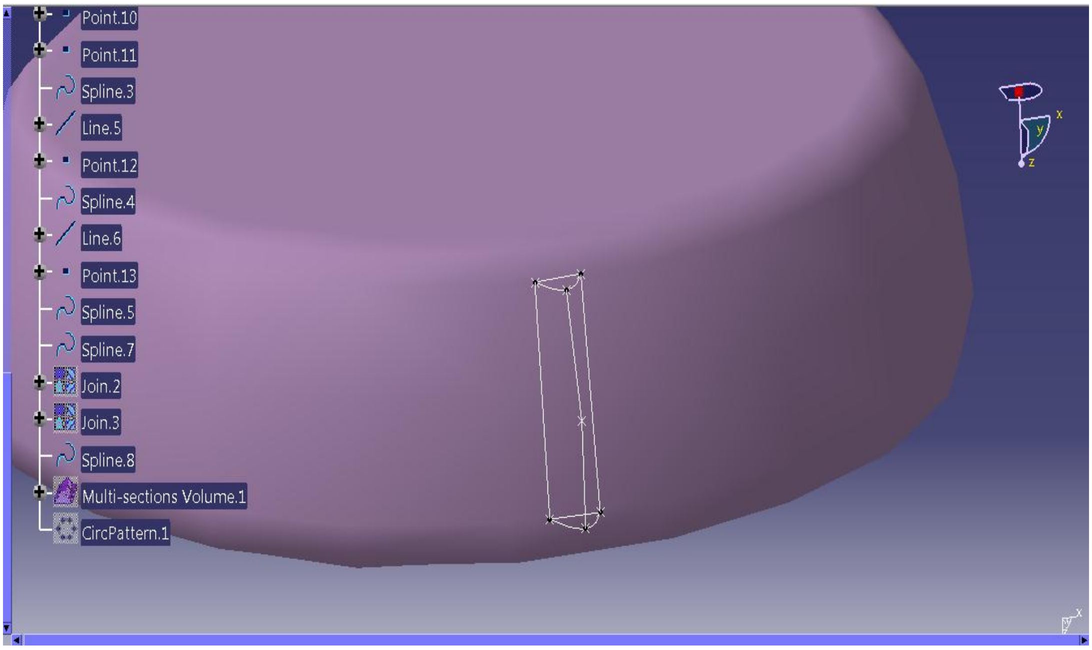
Picture 2.6.3.2: Definition of axis of revolution.



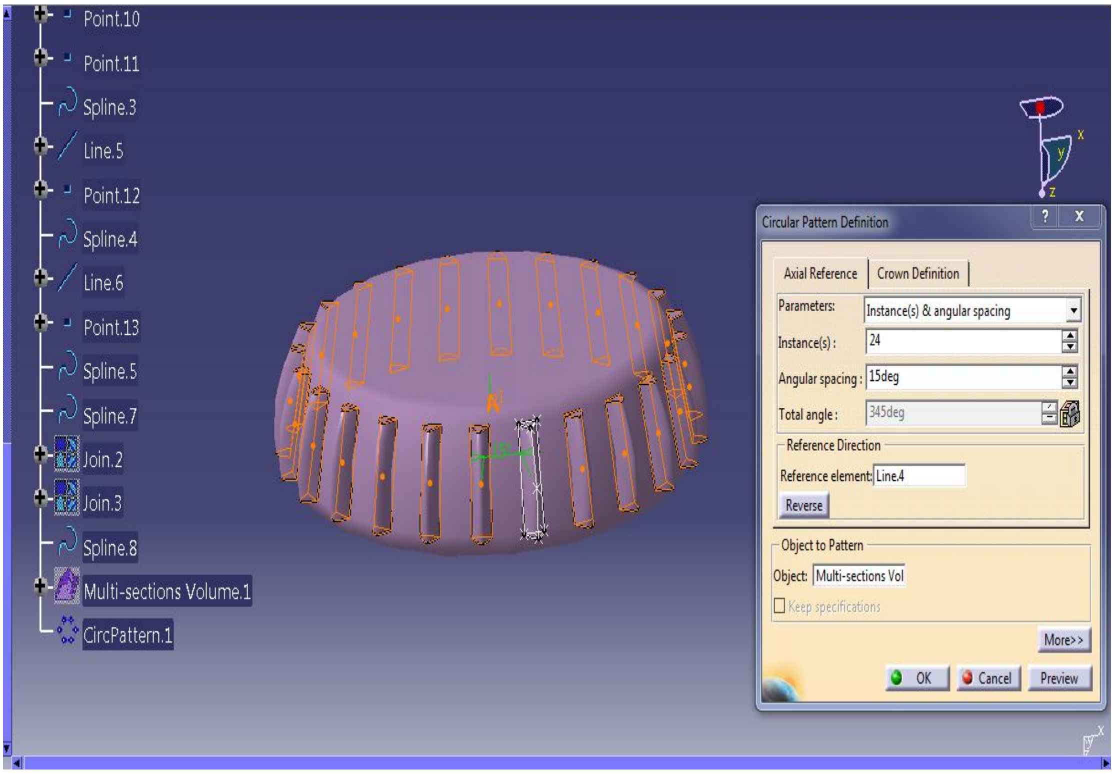
Picture 2.6.3.3: Creation of indicator light third part solid volume.



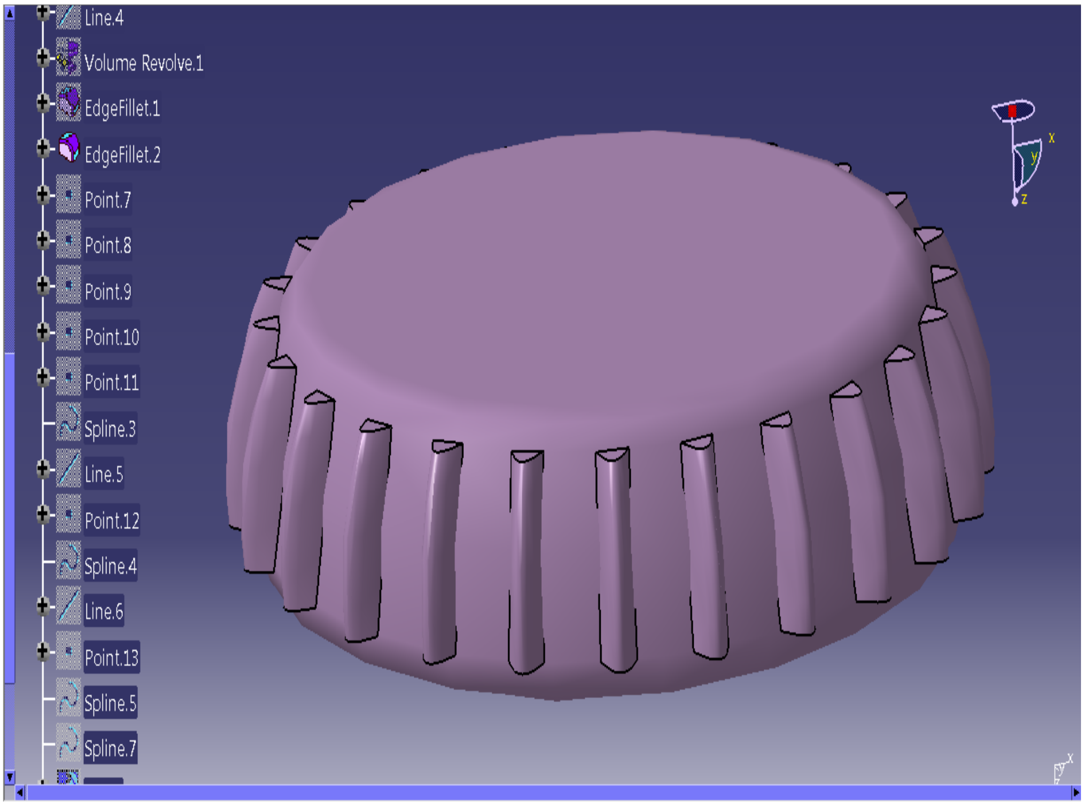
Picture 2.6.3.4: Initiation of indicator light lever third part volume edge filletings.



Picture 2.6.3.5: Definition of grip's wireframe.



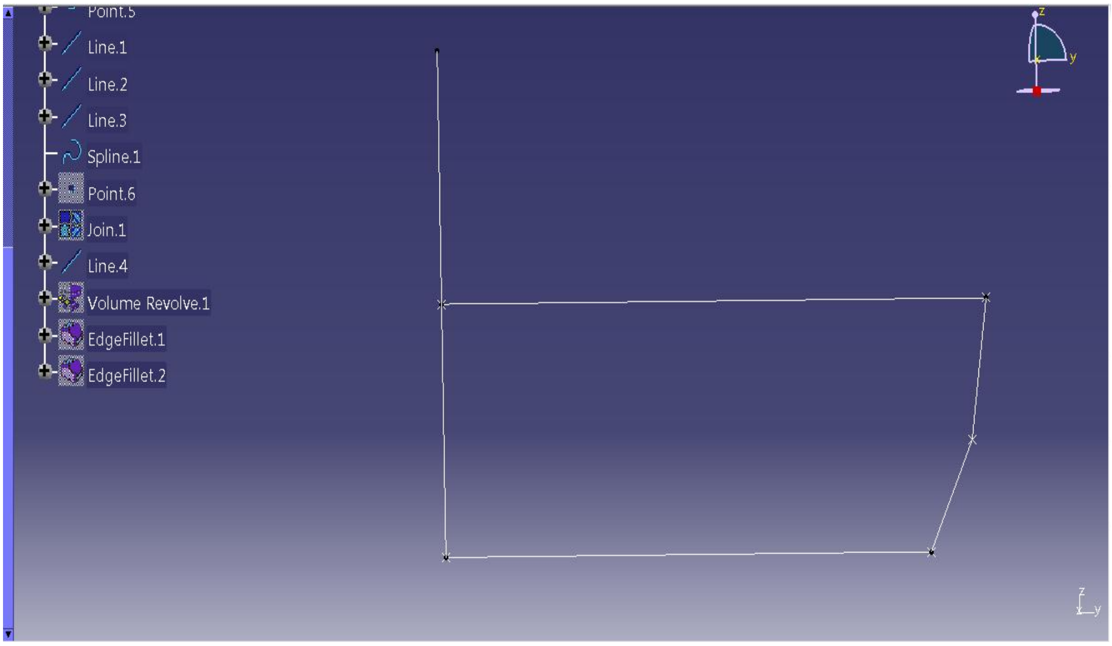
Picture 2.6.3.6: Third part's solid volume with the addition of grips.



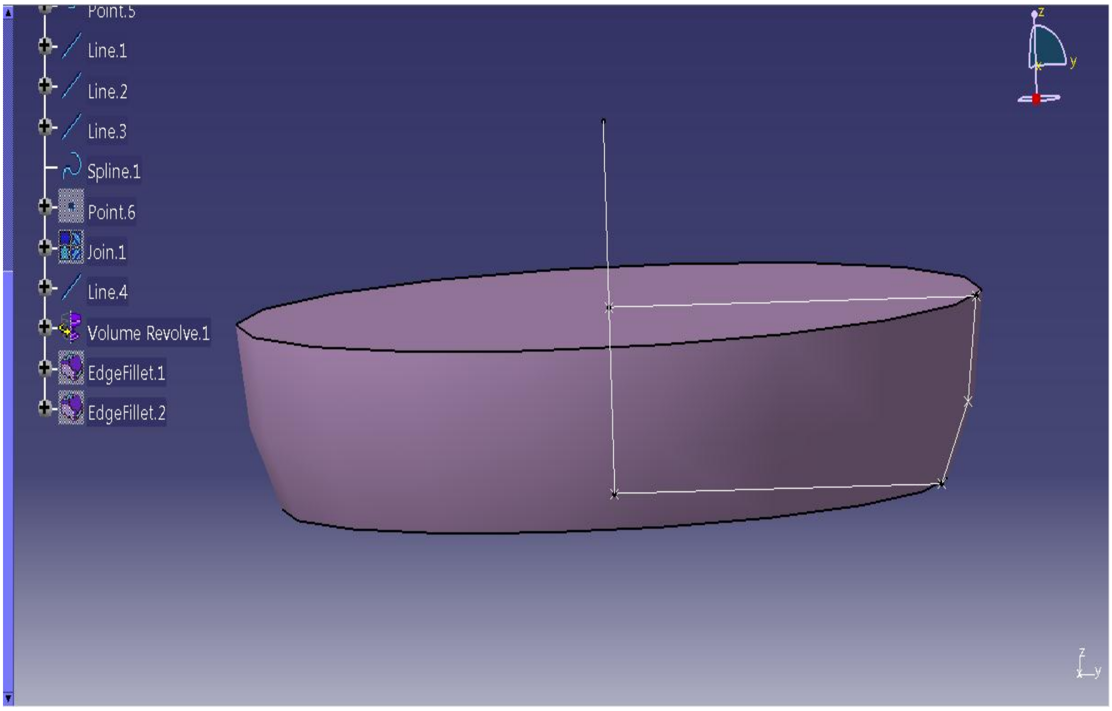
Picture 2.6.3.7: Indicator light lever third part final solid volume.

2.6.4 DEFINITION OF INDICATOR LIGHT LEVER FOURTH PART

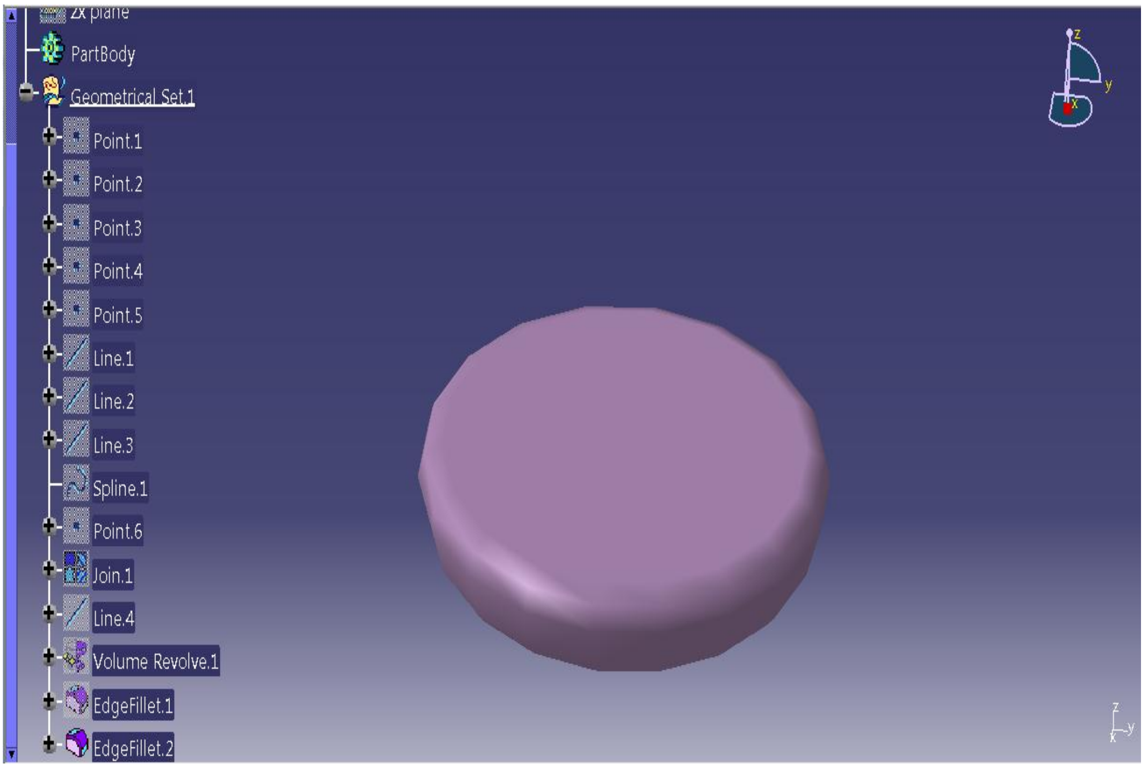
Same as the other lever's parts, this fourth part was also a revolved part. The same procedure was followed for the implementation of this certain part. First, the wireframe model was defined along with the definition of the solid's axis of symmetry (Picture 2.6.4.1). The solid volume was created using the "*multi section volume*" command (Picture 2.6.4.2). Finally, two roundings upon the sharp edges of the solid were inserted (Picture 2.6.4.3).



Picture 2.6.4.1: Wireframe model of indicator light lever fourth part.



Picture 2.6.4.2: Fourth part's revolved volume.

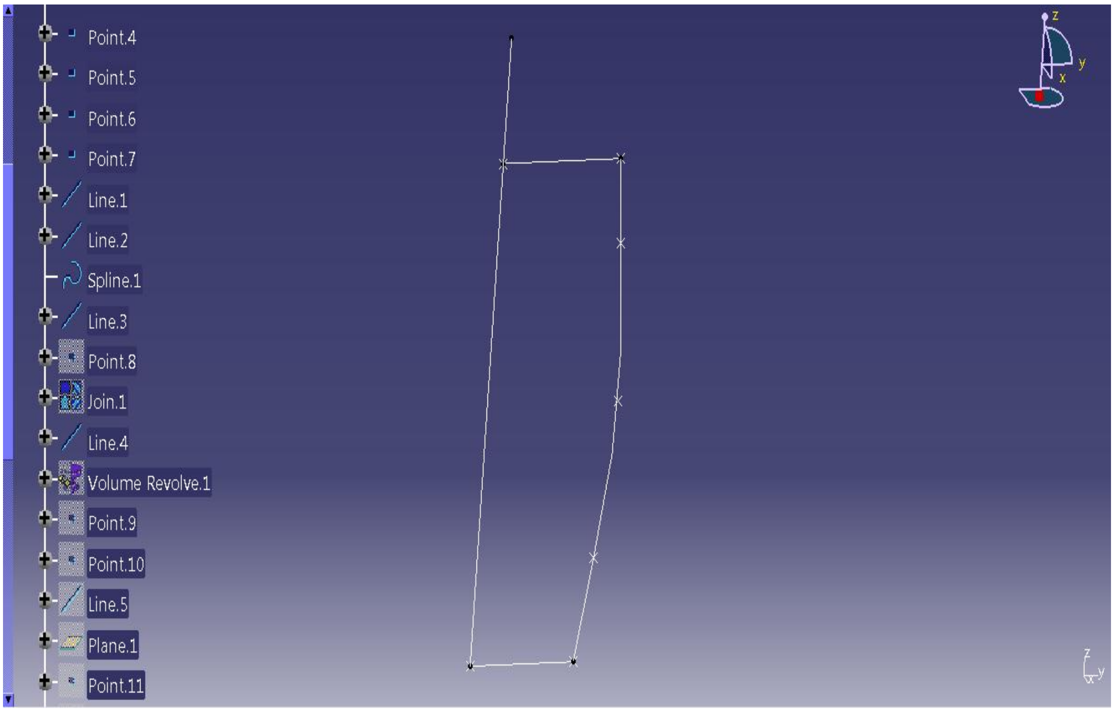


Picture 2.6.4.3:Indicator light lever fourth part final volume.

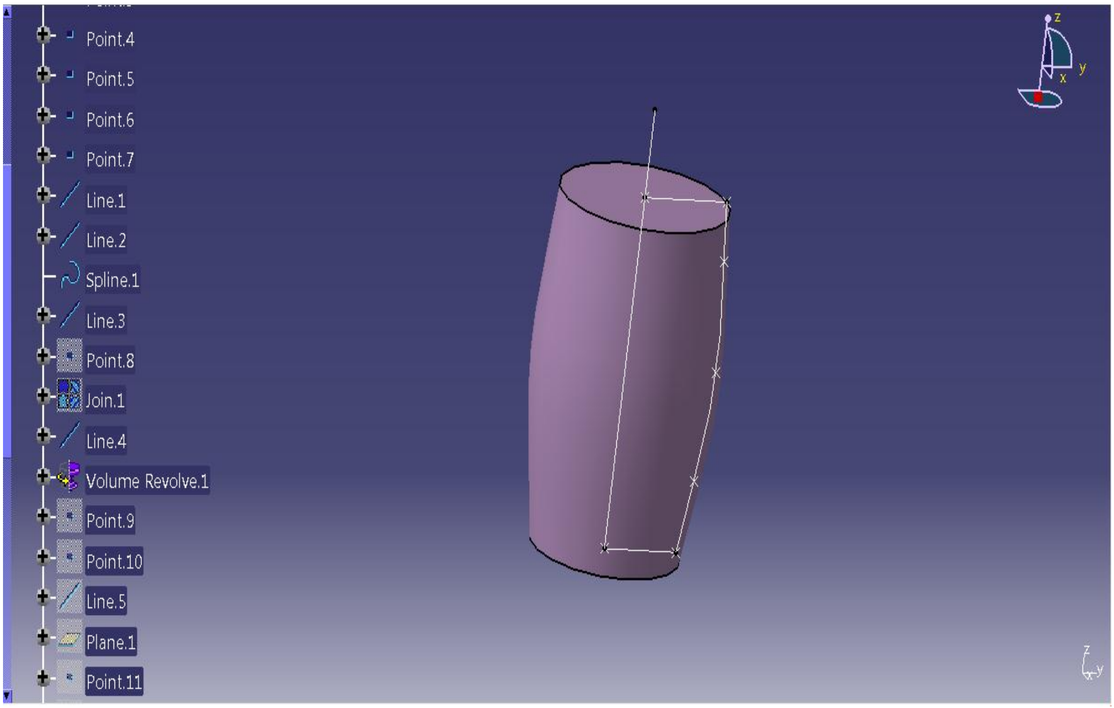
2.6.5 DESIGN OF INDICATOR LIGHT LEVER FIFTH PART

This part is the largest and most complicated component of indicator’s light lever final assembly. For the creation of the solid geometry the exact same methodology was implemented as in the previous parts. The wireframe model, along with the axis of revolution were defined first (Picture 2.6.5.1). Then, the solid volume of the part was constructed by using the “*revolve volume*” operation, which was defined similarly as in the previous parts (Picture 2.6.5.2).

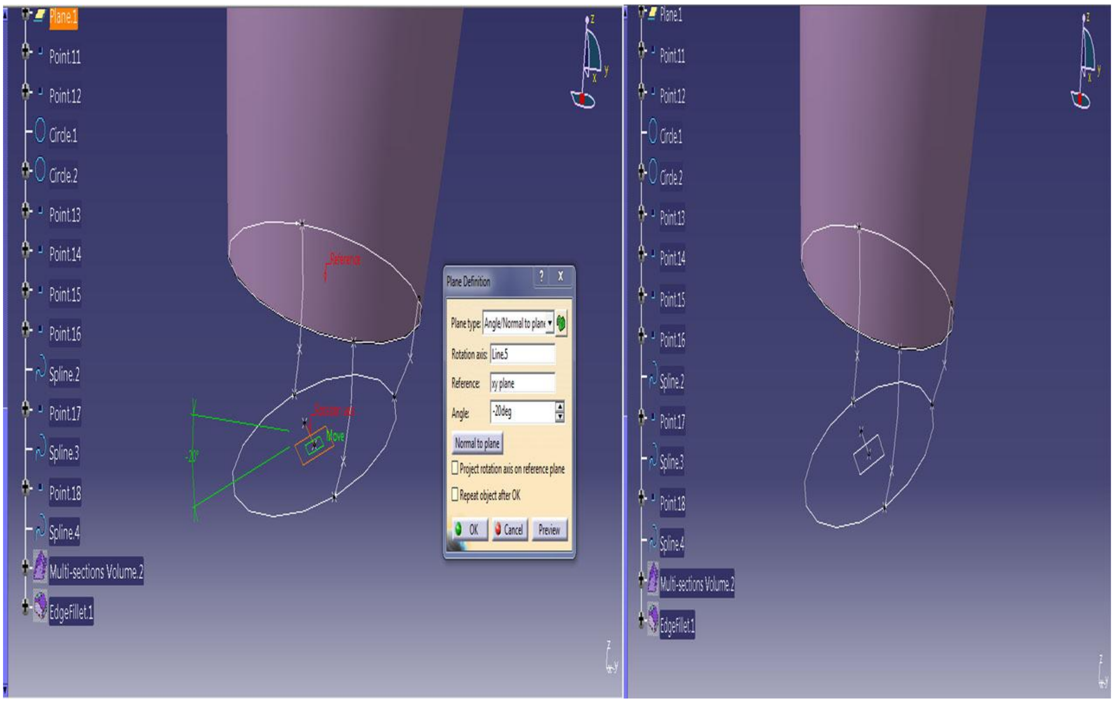
The major difference of this certain part from the other parts of indicator’s light lever volume, is the existence of an additional feature. This feature is the volume that functions as the bridge between the part’s first volume and the steer’s volume that designed specifically for the indicator light lever attachment. First the wireframe model was needed to be produced (Picture 2.6.5.3). For the wireframe designation a plane with twenty degrees inclination from xy - plane was defined. Upon the plane a circular section was defined. A second section, was defined at the existed volume’s bottom surface. This second section followed the circular layout of the surface. In addition, three more B-Spline curves were defined to connect the two sections. Then, the solid volume was created by executing the “*multi section volume*” creation command. As generative curves were inserted the two sections’ wireframes and as guides curves were defined the three additional B-Spline curves (Picture 2.6.5.4). For reasons of embellishment a rounding was imported upon the sharp edge of the upper surface that belongs to the first volume of the part (Picture 2.6.5.5).



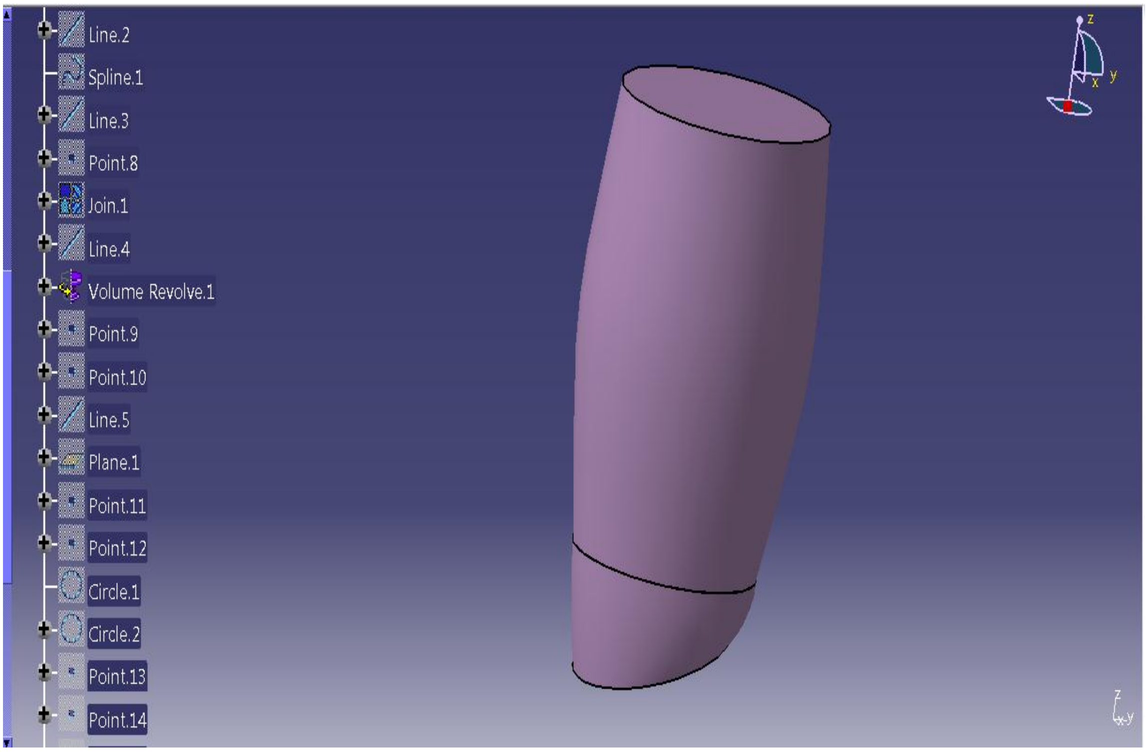
Picture 2.6.5.1: Wireframe model of indicator light lever fifth part.



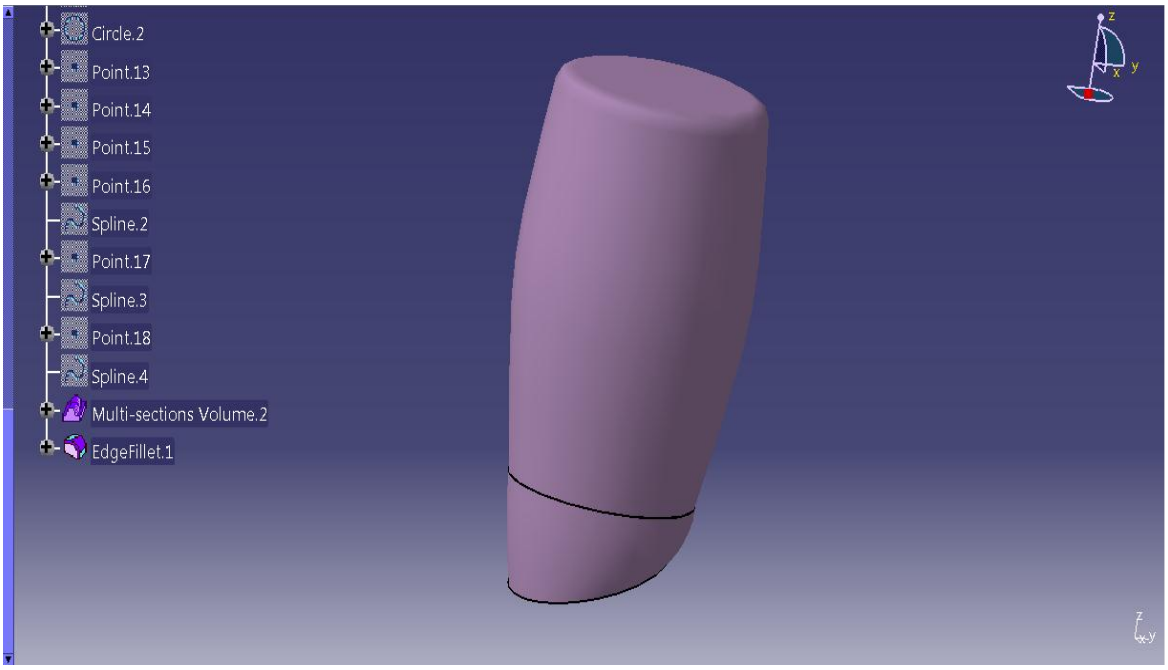
Picture 2.6.5.2 Revolve volume of the fifth part.



Picture 2.6.5.3: Additional volume wireframe definition.



Picture 2.6.5.4: Creation of fifth part's additional volume.



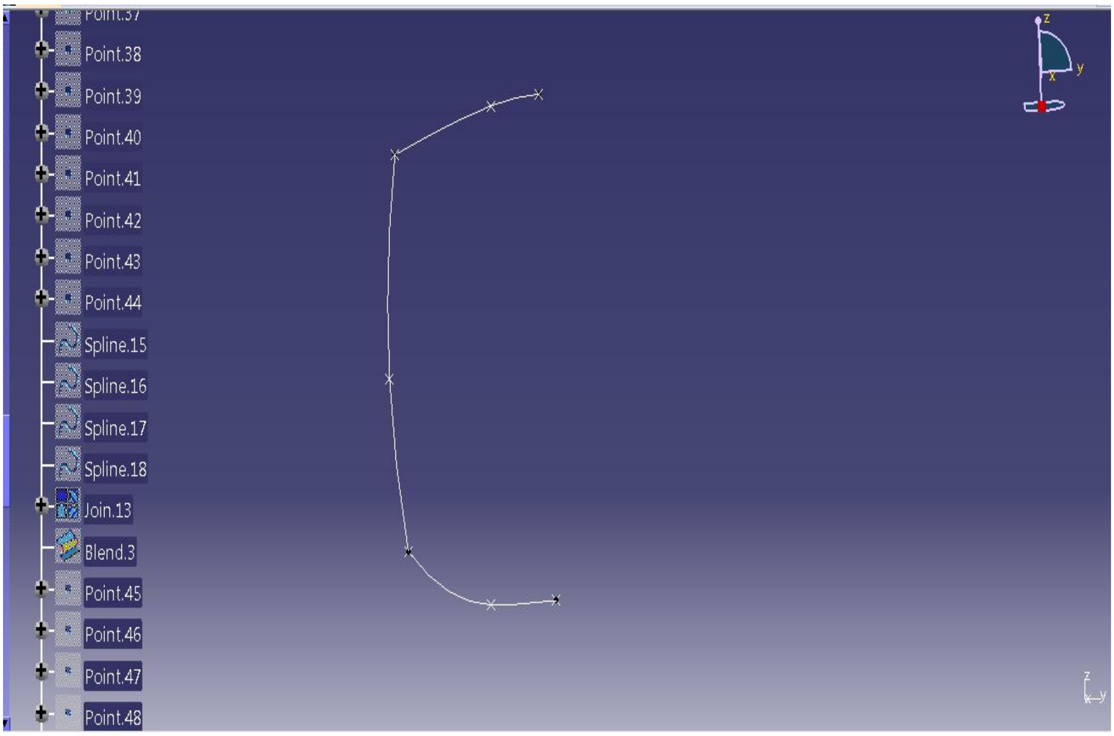
Picture 2.6.5.5: Indicator light's lever fifth part complete volume.

2.7 GEARSHIFT PART

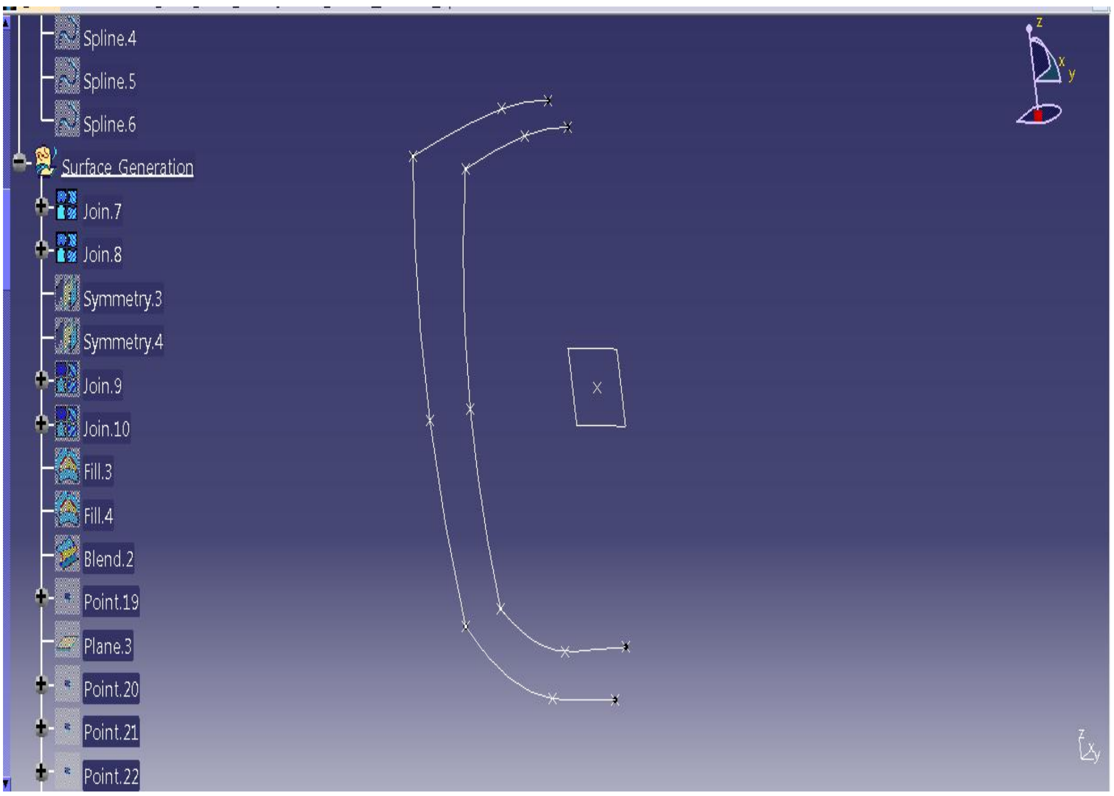
Just like the indicator lights lever, this part is the result of an assembly of five separate entities. Another common aspect between the two assembled volumes (gearshift and indicator light) is that the detailed and complicated design doesn't serve any specific functional purpose. It was the result of the desire to produce a stylish 3-D representation of an actual gearshift volume.

2.7.1 DESIGN OF GEARSHIFT'S BOOT LIFTER PART

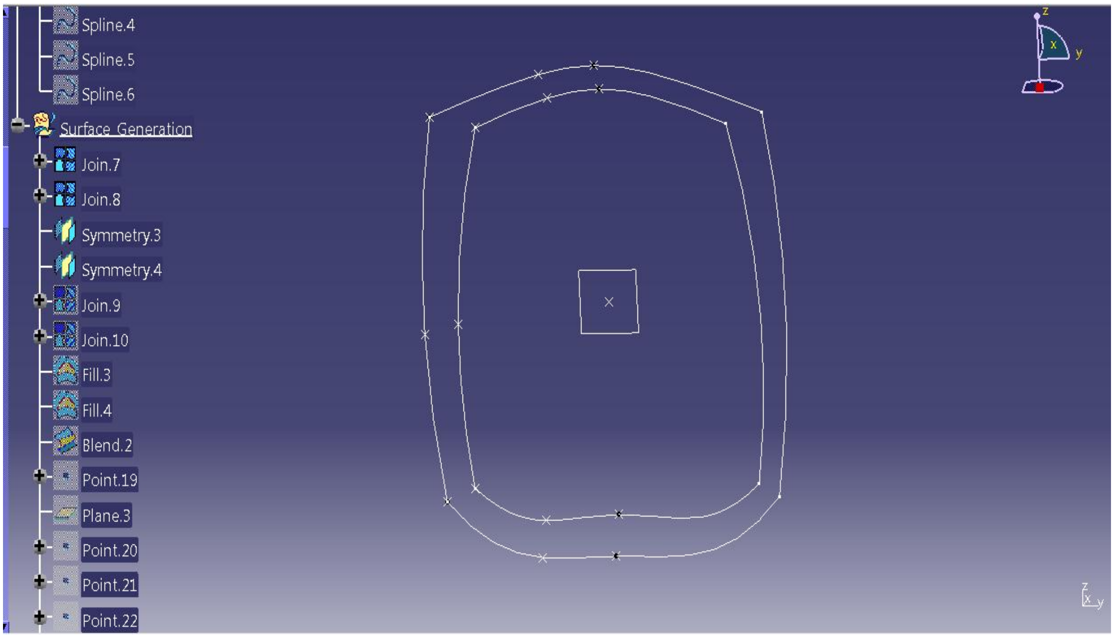
This certain part acted as the transitive entity, connecting the gearshift with the dashboard's console volume. For simplification purposes, many of the gearshift's sub parts were designed as empty surfaces. This certain part also belongs to this category. The wireframe model of the part was defined first. As it will later become clear, the total wireframe consisted of several levels. Each one of these levels contained a separate section. The section of the first level which lied upon yz - plane is shown in Picture 2.7.1.1 below. The wireframes at the following levels were designed in a similar way as the first one. The second level was constructed upon a plane parallel to yz - plane, 8 mm above it (Picture 2.7.1.2). Next followed the unification of the two wireframes. The symmetrical halves of the sections were created by using the "*symmetry*" operation technique. As symmetry plane the xz – plane was inserted (Picture 2.7.1.3). Then, two surfaces were constructed with the "*fill*" command, by defining the two sections' layouts as boundaries (Picture 2.7.1.4). The design of these two surfaces was for the single purpose of creating the peripheral surface between the two surfaces. This surface was created using the "*blend surfaces*" command. As inputs for the implementation of "*blend*" command the two sections were defined. Inside this command, an additional feature was needed to be defined. This certain feature is the surface, to which each section is adjacent. The result of this process is shown in Picture 2.7.1.5 below. The third level wireframe was created on a plane with a distance of 8 mm from the previous section's level. The geometry of the wireframe was similar to that of the previous sections (Picture 2.7.1.6). Then, an extruded surface was produced, by setting the section from the third level as a generative curve. The direction of the extrude operation was set to coincide with a vector's direction, perpendicular to zy - plane. The extruded surface was constrained to be extended to the second level's fill surface (Picture 2.7.1.7). Then followed a cutting operation using "*split surface*". In order to execute this command a cutting surface and a surface to be cut should be initiated. For this certain subtraction operation, as cutting surface the extruded surface was imported and as surface to be cut the fill surface was selected. The result of the operation was a single surface, which contained a portion of each initial surface (Picture 2.7.1.8). The fourth level wireframe was designed on a plane 2 mm under the second level's plane. The section's geometry was similar to that of the sections before (Picture 2.7.1.9). By choosing the "*fill*" surface command, an additional surface was created on the fourth level wireframe. Then the fifth section was generated as a layout on the external edge of the surface of this part. The fourth and fifth sections were united together by using the "*blend surface*" operation. The resulting surface was a transitive surface between the fifth and the fourth section (Picture 2.7.1.10). Finally, a recess upon the surface was made. The functional purpose of the recess was the definition of the area of mobility of the gearshift on the boot lifter. This mobility of the gearshift represented the selection of different gear ratios. This recess was the result of two similar sections designed with a distance of 1,5 mm between them. A "*fill*" surface was created on the second section's wireframe. Then the two wireframes were connected together with "*blend surface*" (Picture 2.7.1.12). The complete geometry of the boot lifter's surface is depicted in Picture 2.7.1.13.



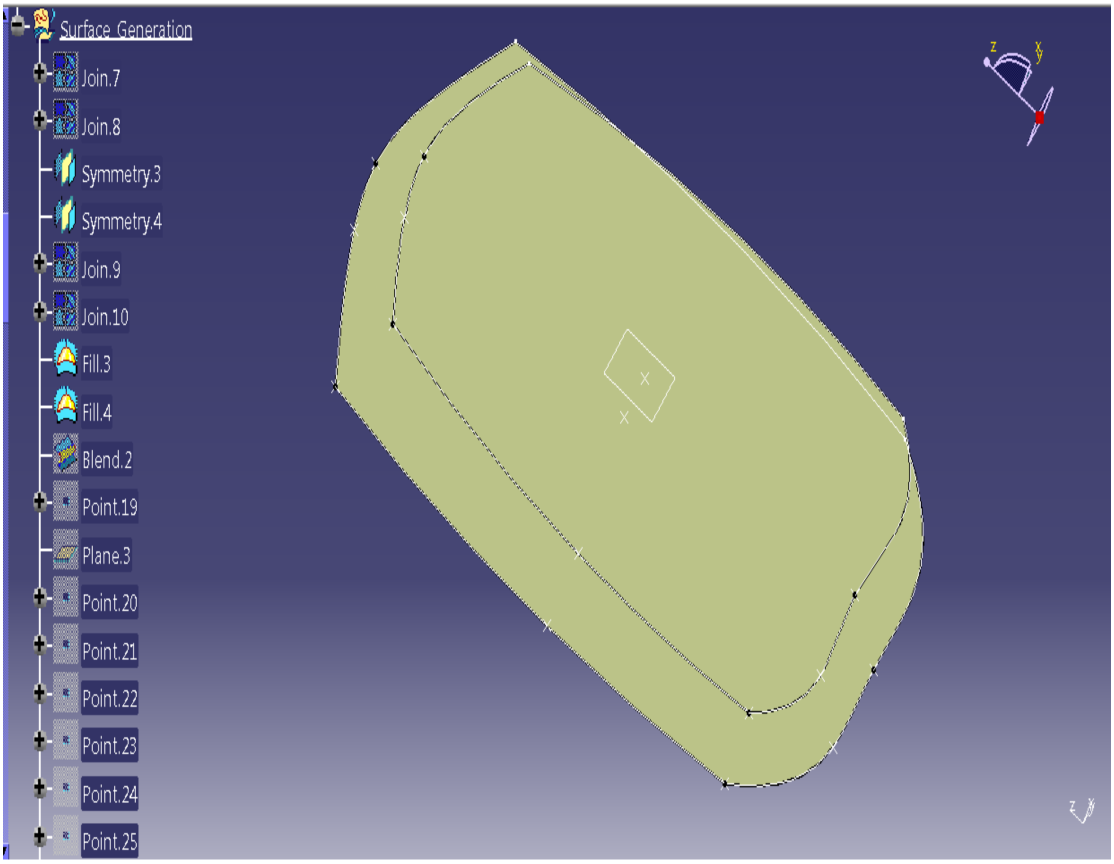
Picture 2.7.1.1: Boot lifter's first level wireframe model.



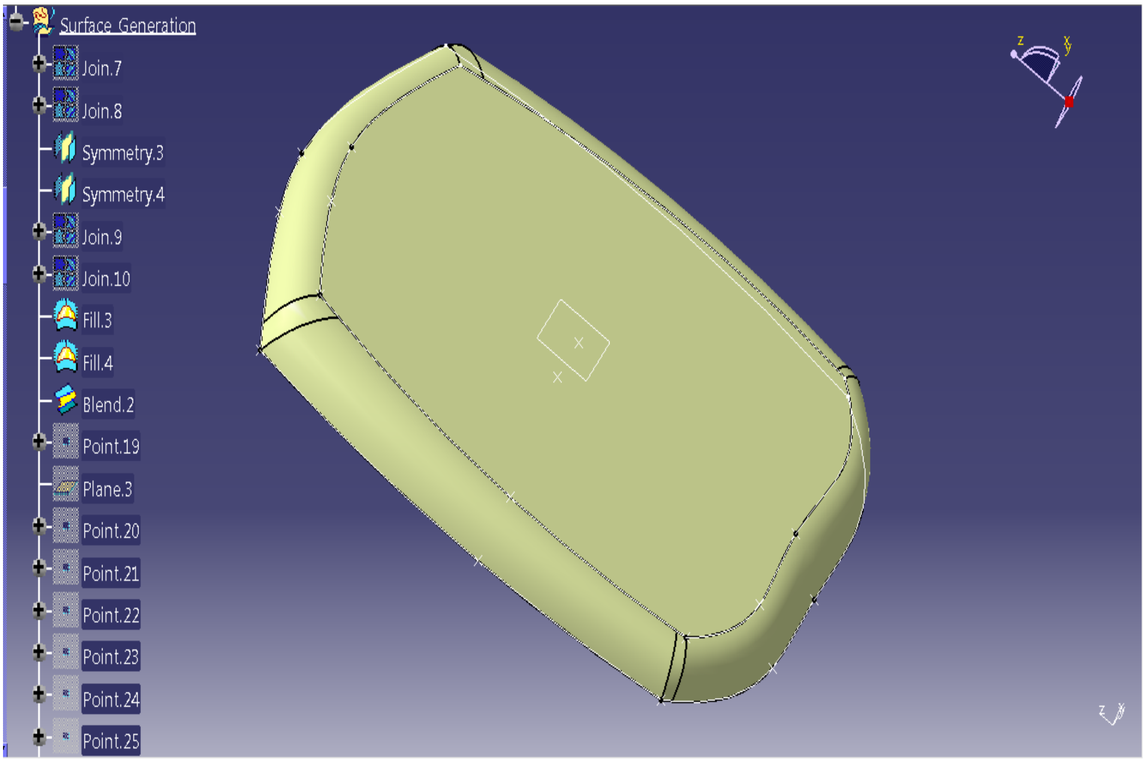
Picture 2.7.1.2: Boot lifter's second level wireframe model.



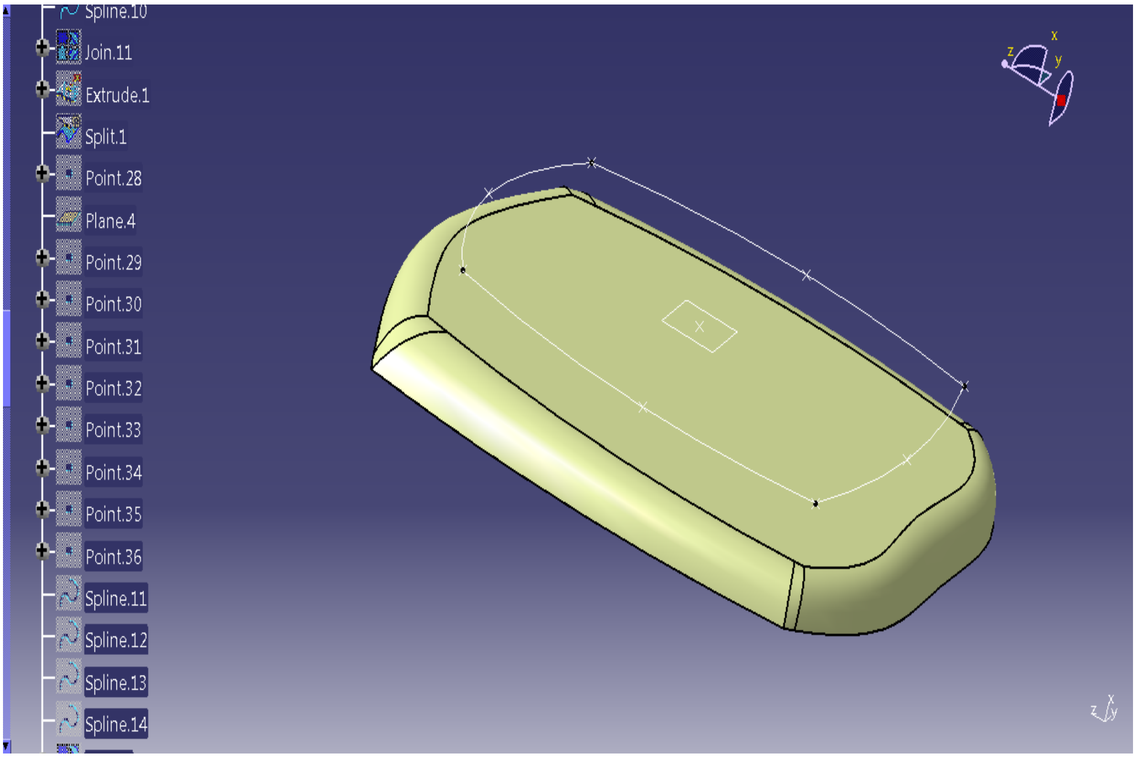
Picture 2.7.1.3: Creation of the symmetrical wireframes of the first two sections.



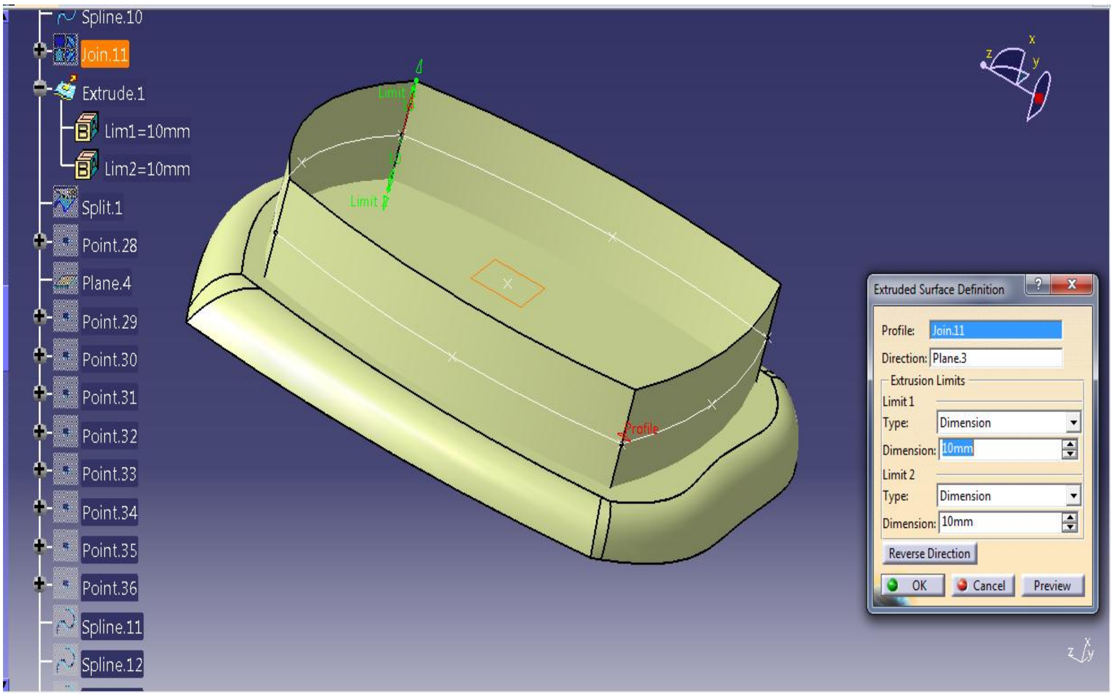
Picture 2.7.1.4: Creation of fill surfaces of level one and two.



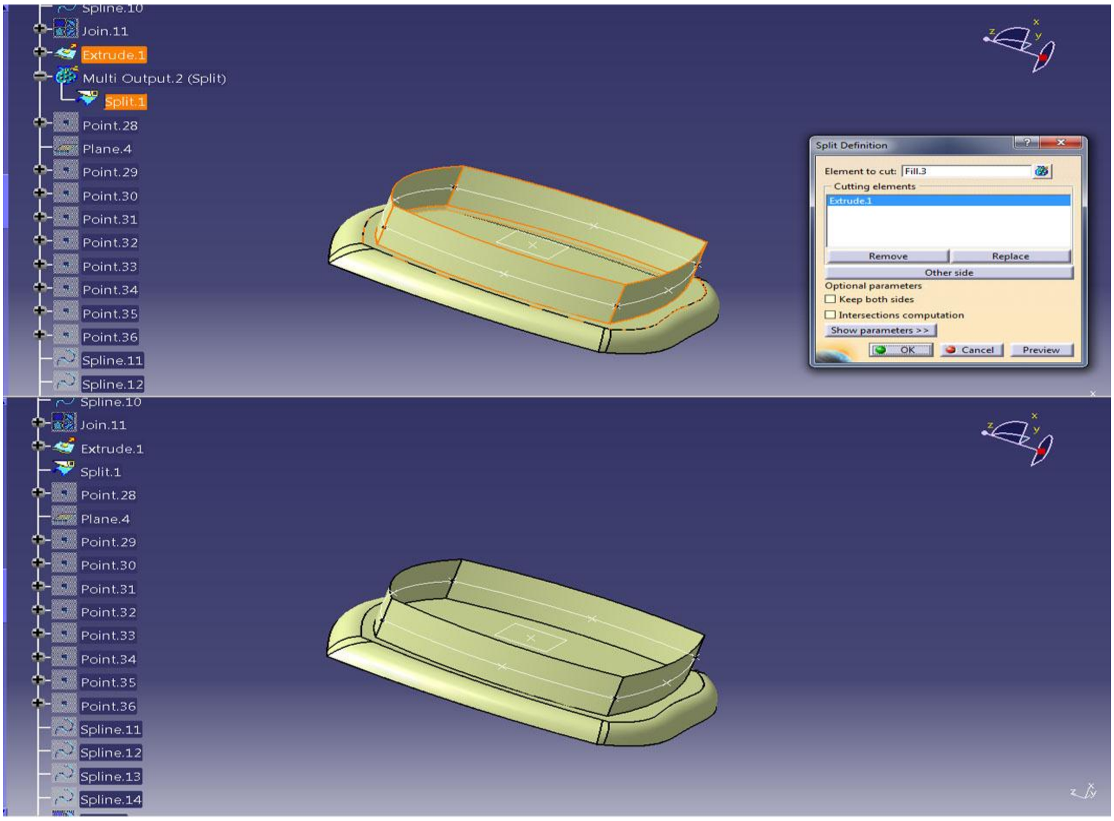
Picture 2.7.1.5: Creation of boot lifter's peripheral "blend" surface.



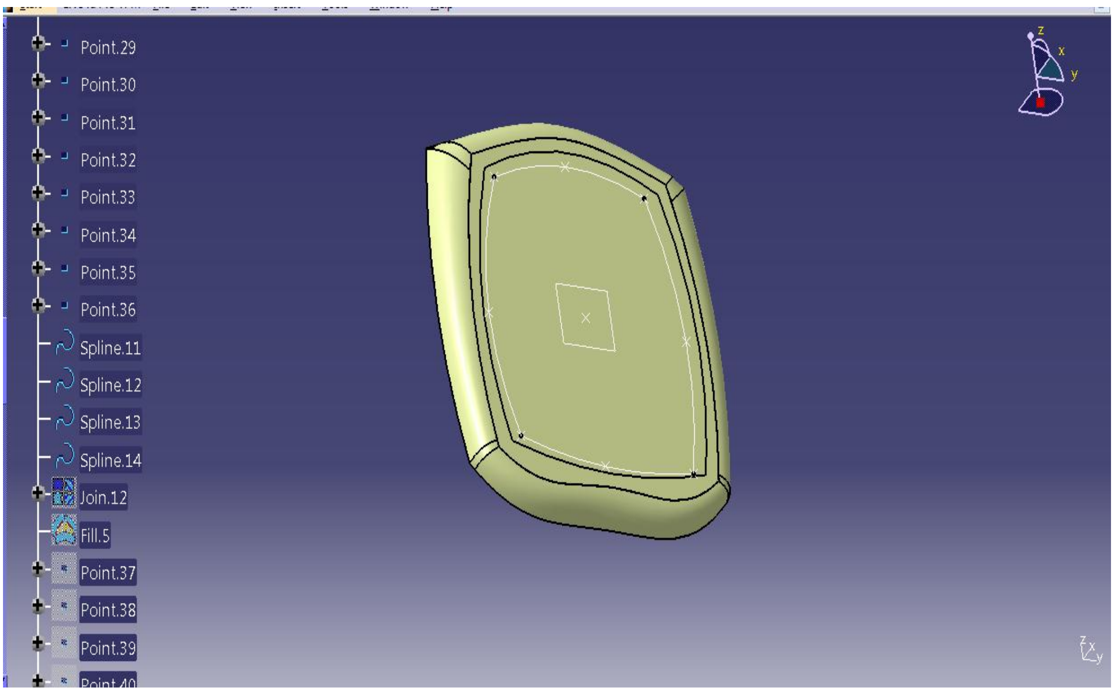
Picture 2.7.1.6: Boot lifter's wireframe of third level.



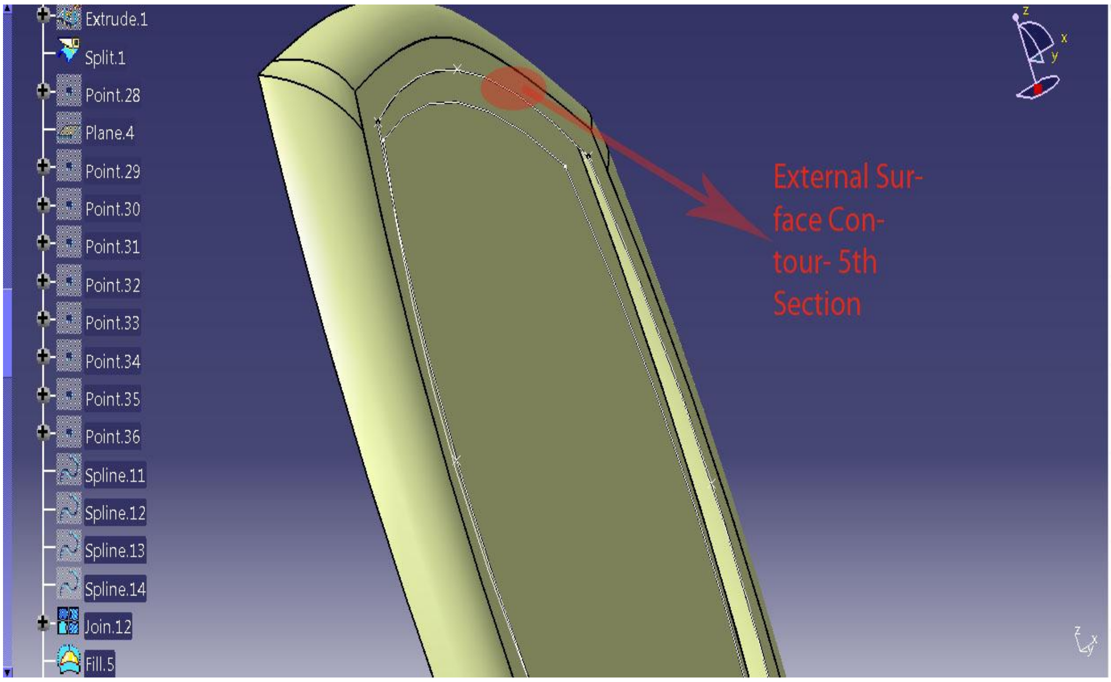
Picture 2.7.1.7: Third level's extruded surface.



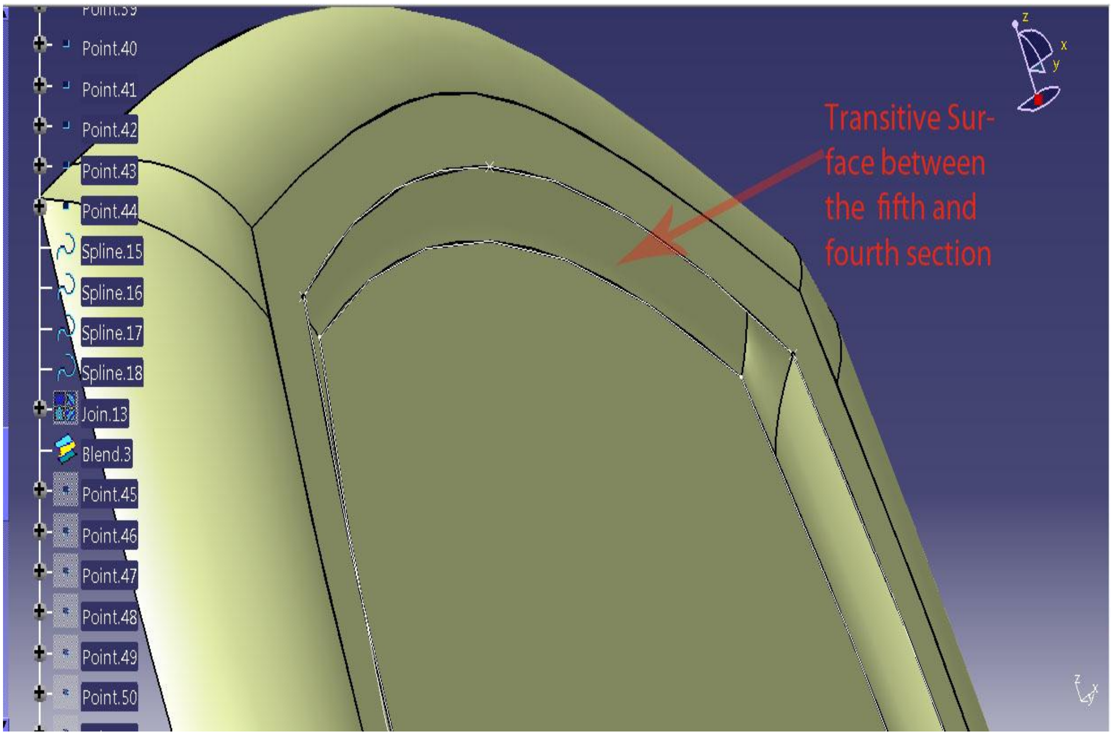
Picture 2.7.1.8: Definition and execution of “*split surface*” command.



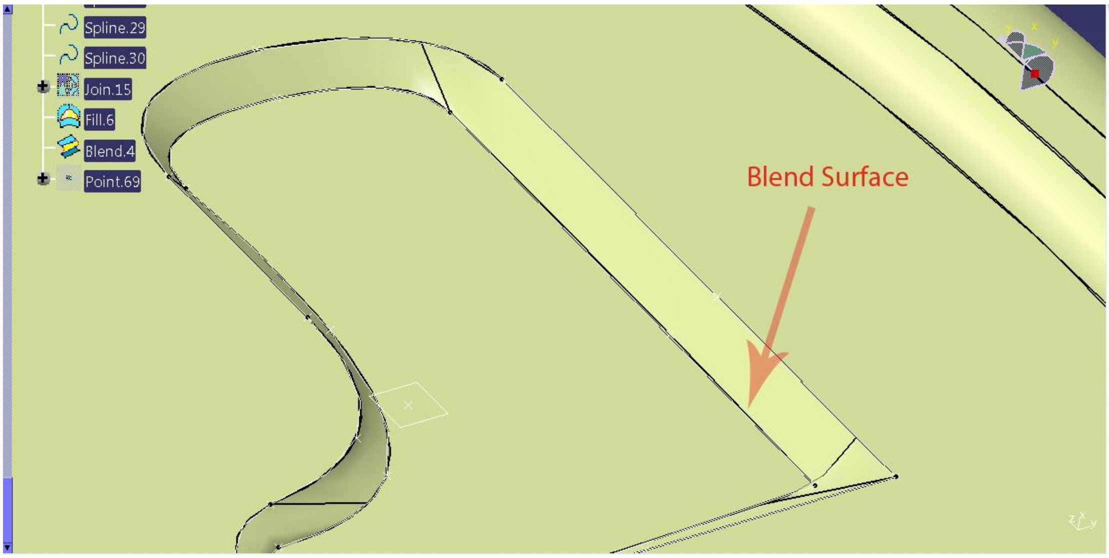
Picture 2.7.1.9: Creation of fourth level’s section wireframe.



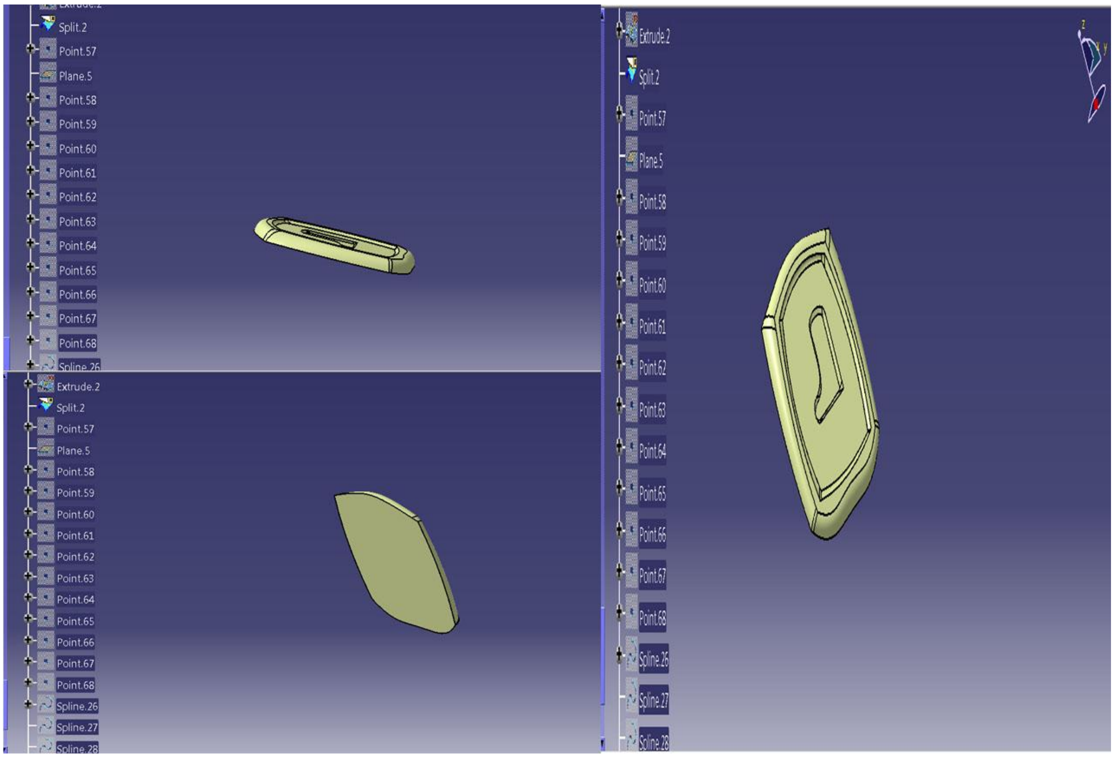
Picture 2.7.1.10: Boot lifter's wireframe of fifth section.



Picture 2.7.1.11: Definition of boot lifter's transitive surface between the fourth and fifth sections.



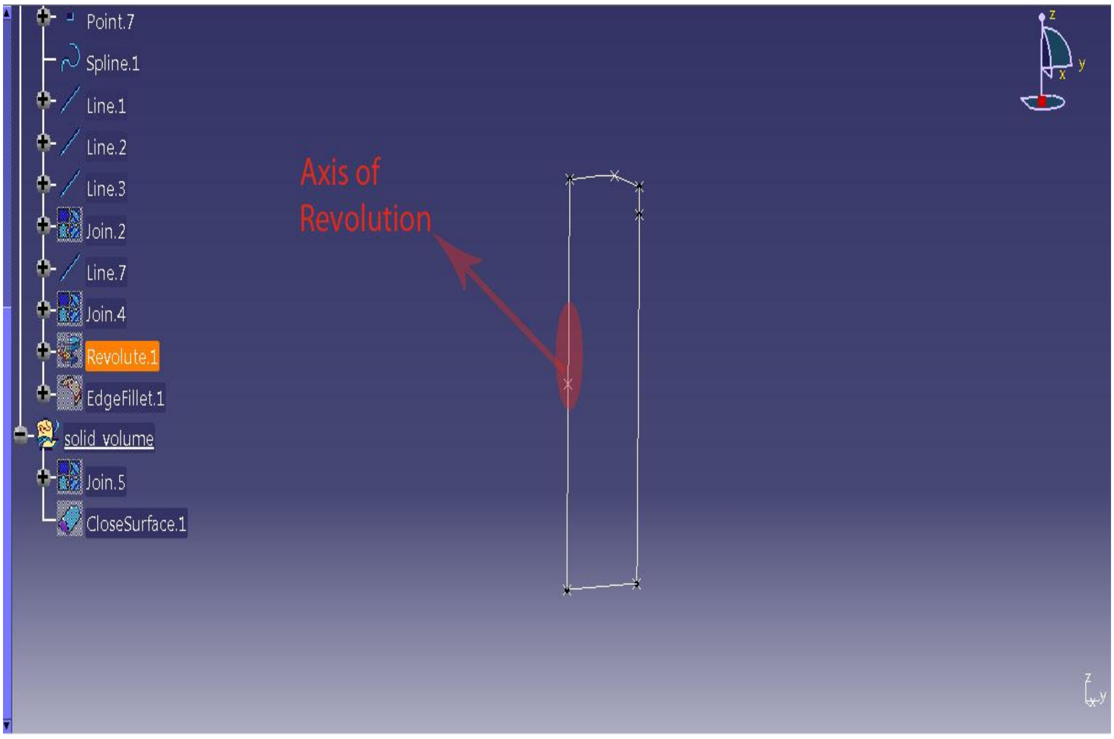
Picture 2.7.1.12: Blended transitive surface between the part's last two sections.



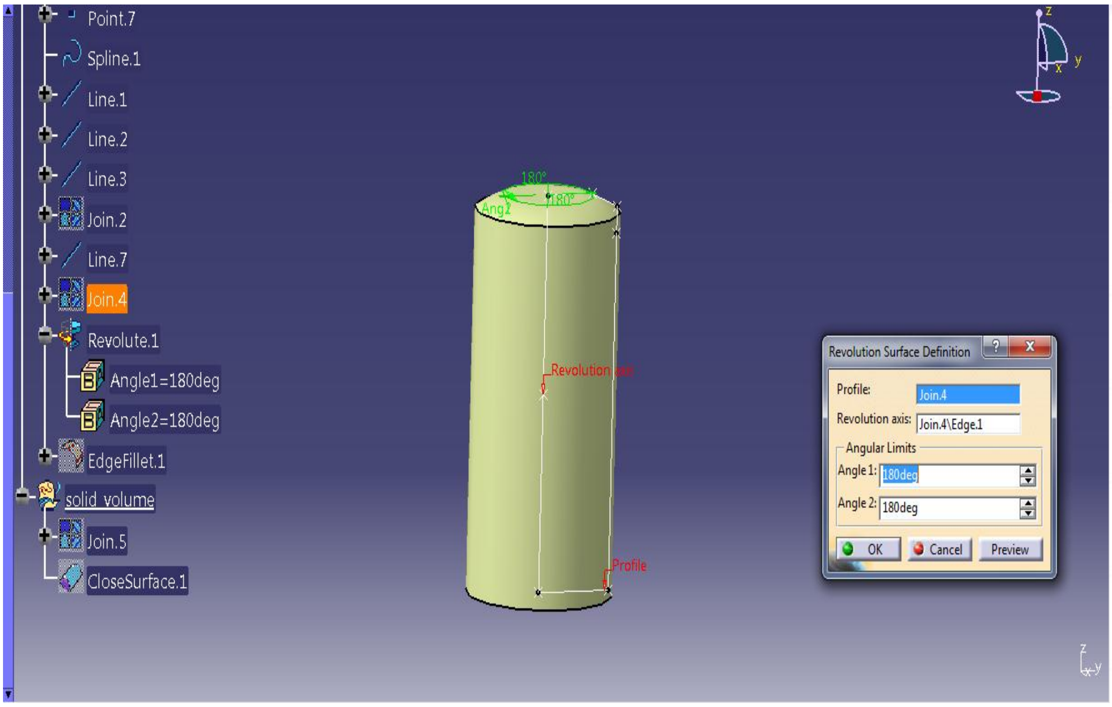
Picture 2.7.1.13: Complete boot lifter part.

2.7.2 GEARSHIFT DESIGN

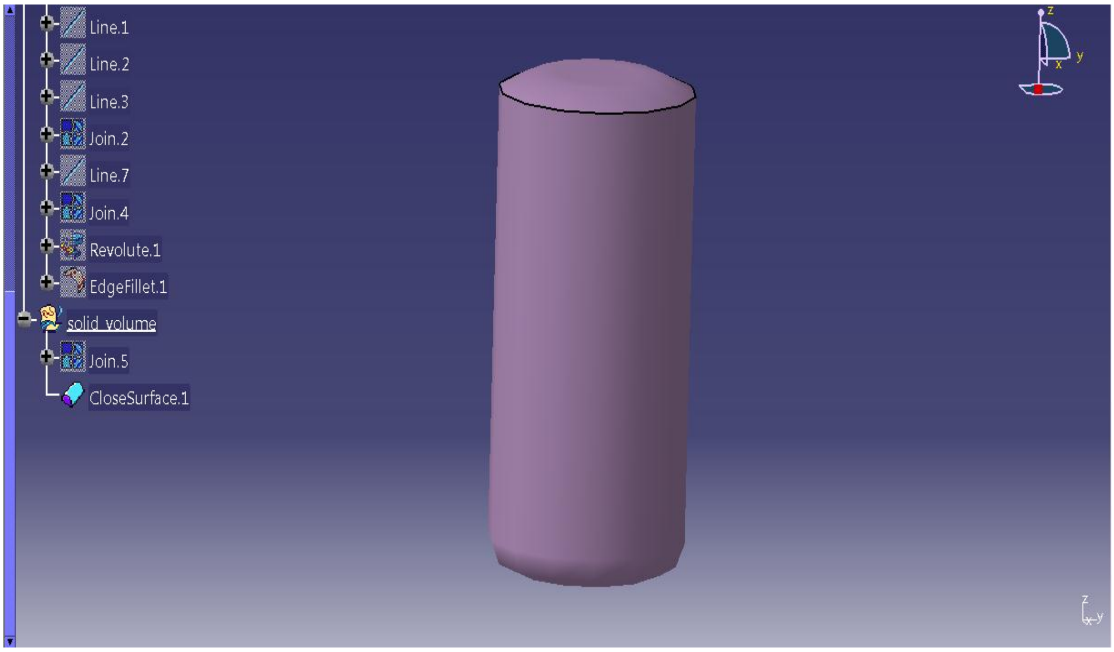
This part is functionally acting as the connection between the knob unit and the boot lifter part. Naturally, this geometry was constructed as a revolved volume. Firstly the wireframe model was constructed (Picture 2.7.2.1). This wireframe model consisted of four lines and an additional B-Spline. The wireframe section of this model was designed to be closed. As revolution axis of this model the single line which closed the wireframe was selected. Then, the regional surface of the solid volume was created. This surface was created as a revolved surface. As generative curve for this operation the section's profile was selected (Picture 2.7.2.2). The final solid geometry was constructed by using the "*close surface*" command (Picture 2.7.2.3).



Picture 2.7.2.1: Gearshift's wireframe model.



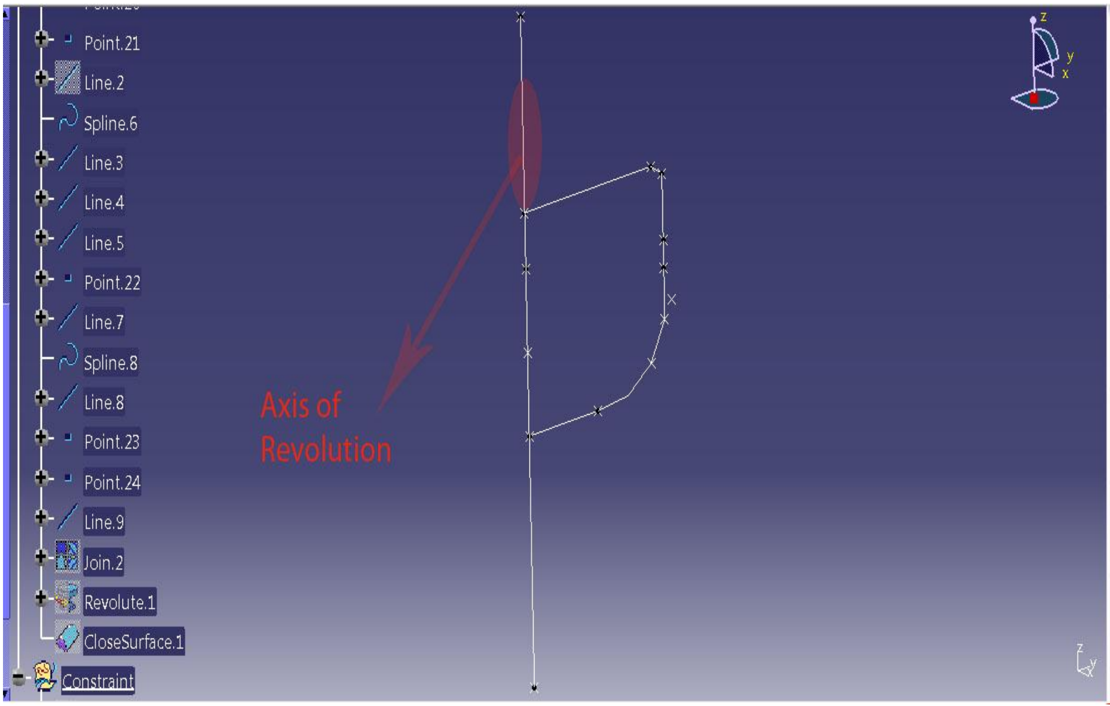
Picture 2.7.2.2: Design of gearshift's external surface.



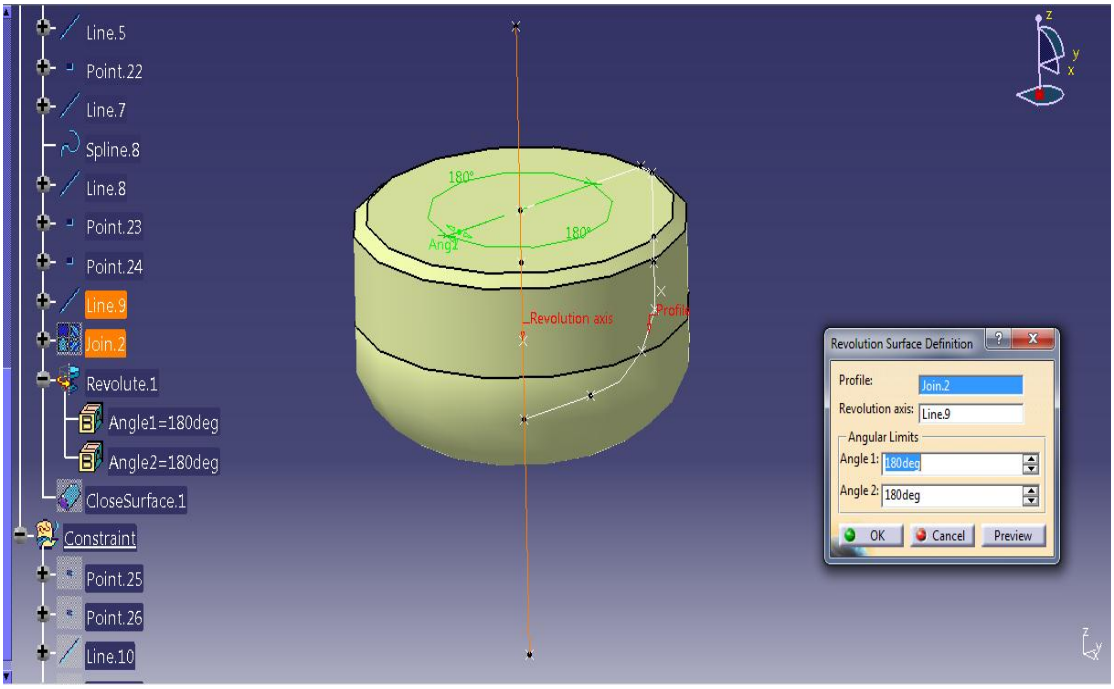
Picture 2.7.2.3: Gearshift's solid volume.

2.7.3 DESIGN OF SHIFT'S LINKAGE BUSHING.

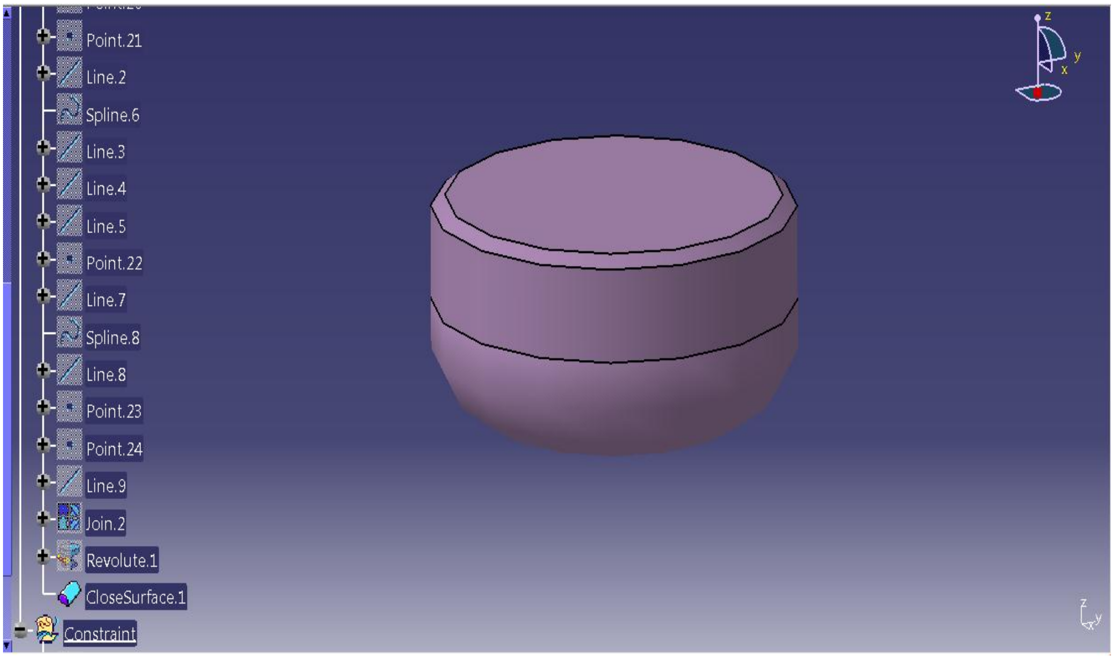
This part is the piece that connects the knob unit with the gearshift unit. Just like the previous part, this one is also a revolved volume part. For the definition of the solid volume of the part, the wireframe model was created first (Picture 2.7.3.1). The wireframe model was defined as a closed section. This closed section consisted of five lines and two B-Splines. Moreover, an additional line was created in order to be used as an axis of revolution. Then the revolved surface was created, using the "*revolve*" operation. As generative curve for the implementation of the operation the section's profile was used (Picture 2.7.3.2). Then the final solid volume was created using the "*close surface*" operation (Picture 2.7.3.3).



Picture 2.7.3.1: Shift's linkage bushing wireframe model.



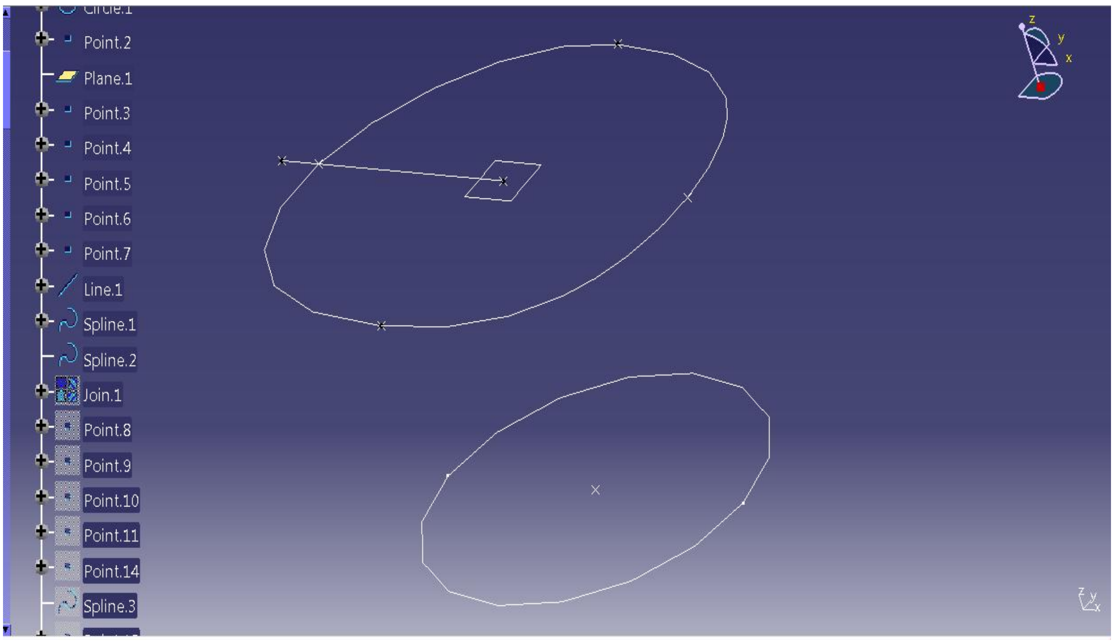
Picture 2.7.3.2: Shift's linkage bushing surface of revolution.



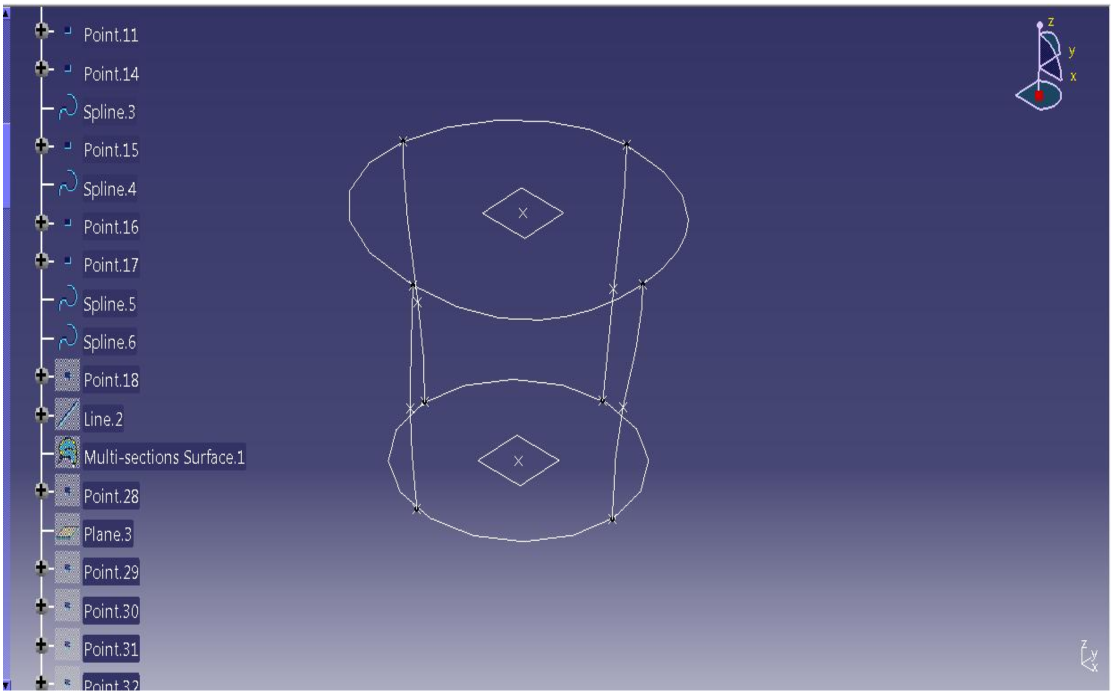
Picture 2.7.3.3 Shift's linkage bushing solid volume.

2.7.4 DESIGN OF KNOB PART

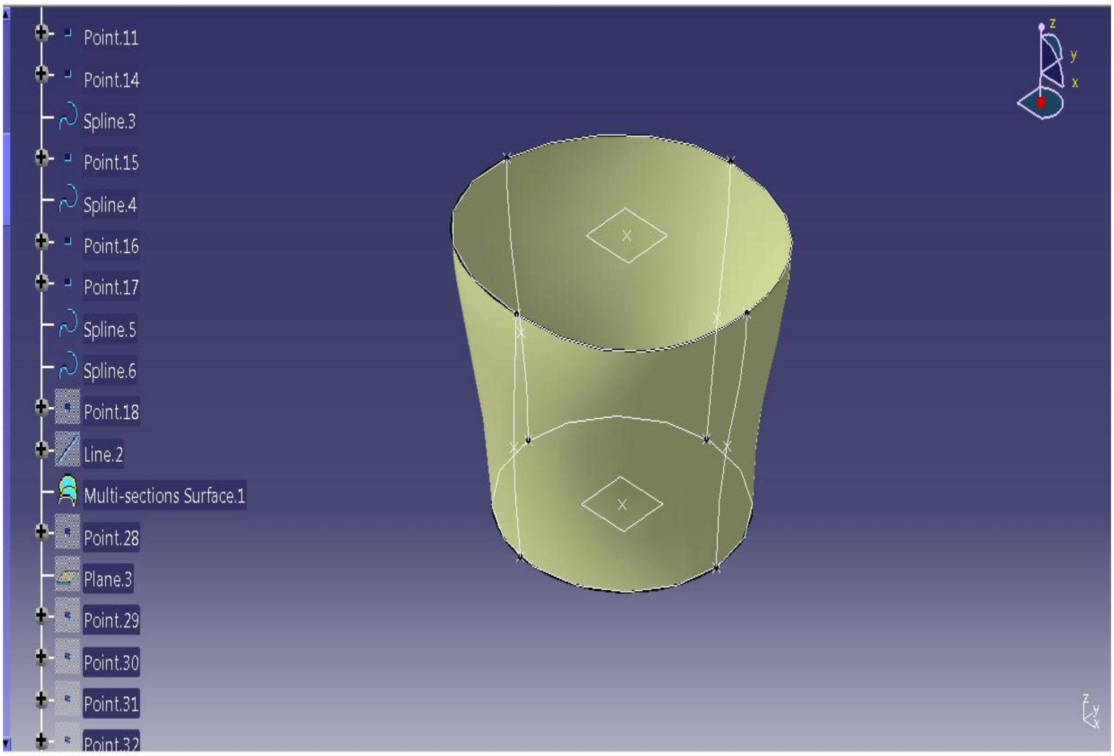
For reasons of simplicity and time saving, this part was designed only as a surface. At first, the wireframe model was determined (Picture 2.7.4.1). Two circular sections were designed, with a distance of 17.5 mm distance between them. These two sections were connected together with four additional B-Spline curves (Picture 2.7.4.2). The regional surface of the wireframe was constructed as a multi section surface. Inside the command definition, the two wireframe sections were defined as generative curves. The four spline curves that connected the two sections were selected as guide curves (Picture 2.7.4.3). Furthermore, an additional oval - shaped wireframe geometry was constructed, 30 mm above the first circular section. Following the same methodology as before, the oval and circular sections were connected together with four spline curves (Picture 2.7.4.4). In the exact same way the peripheral surface was created (Picture 2.7.4.5). Next in the design sequence followed the creation of an additional circular section. This circular section was defined 12.5 mm above the oval section (Picture 2.7.4.6). The transitive surface between the two sections was created with the "*sweep surface*" operation. For the implementation of this command a curvilinear spline was used as a generative curve while the two section wireframes were used as guide curves (Picture 2.7.4.7). The gap at the top of the knob's surface design was filled with a surface, which was created as a filled surface (Picture 2.7.4.8) As boundary for the creation of the surface the circular profile was used at the peak of the design of the wireframe. Finally, for the creation of a rounding on the lower surface of the knob unit the unification of all the surfaces was needed. This unification was made with the use of "*join*" command while for the rounding on the lower surface of the knob the *"edge fillet*" command was used (Picture 2.7.4.9).



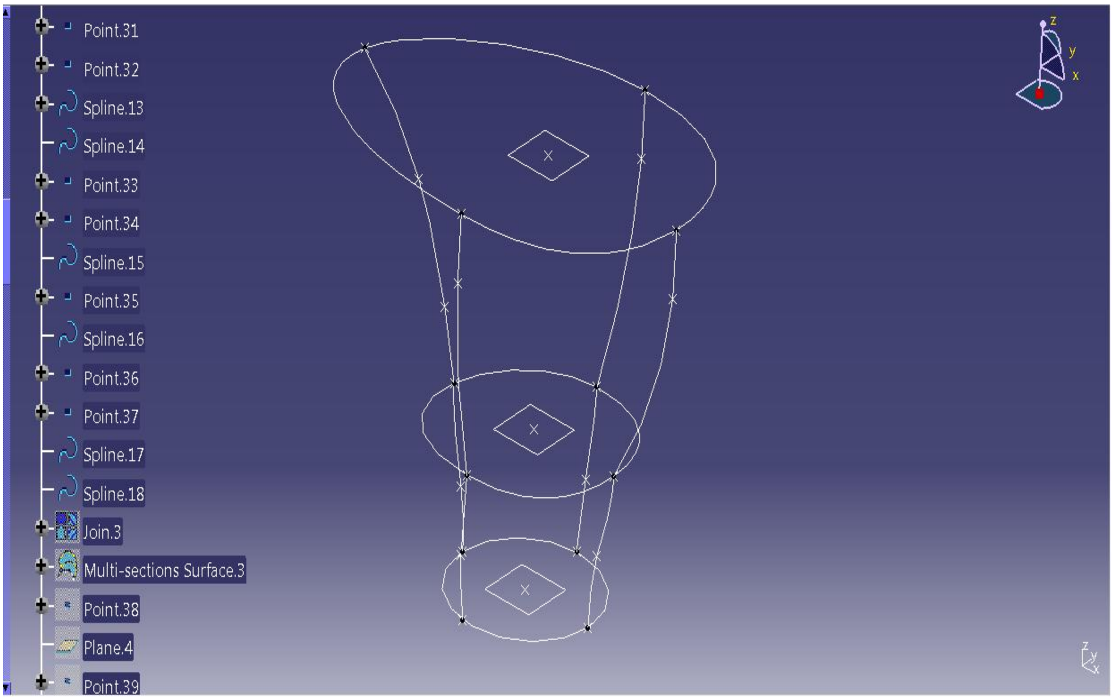
Picture 2.7.4.1: The first two circular wireframe sections of the knob part.



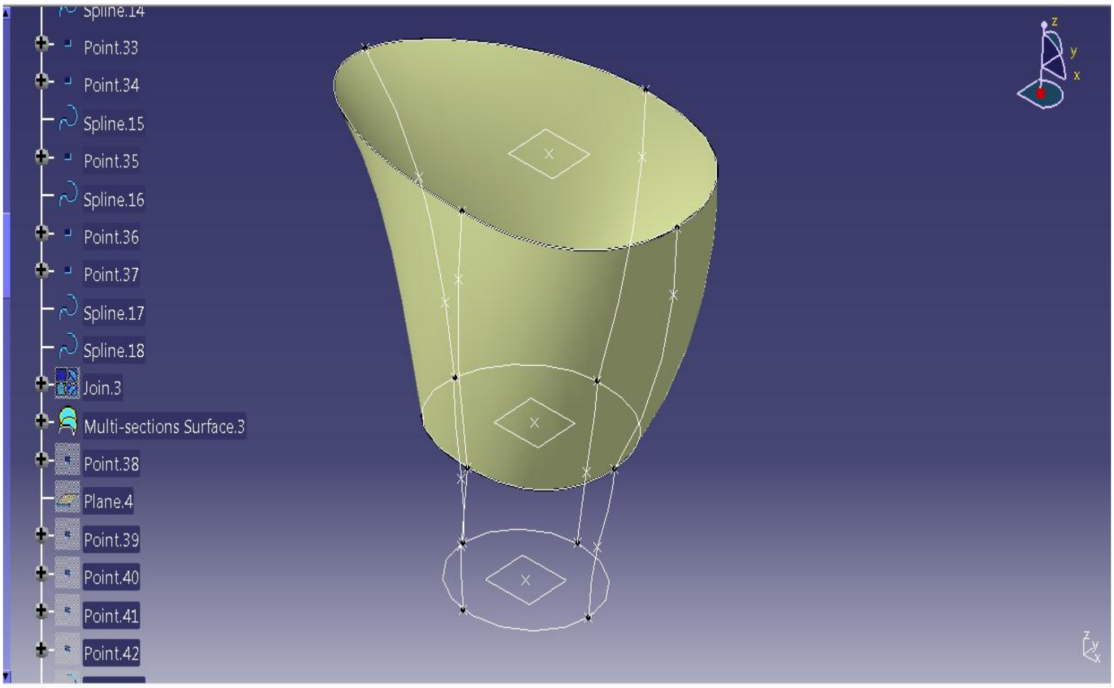
Picture 2.7.4.2: Definition of knob spline curves.



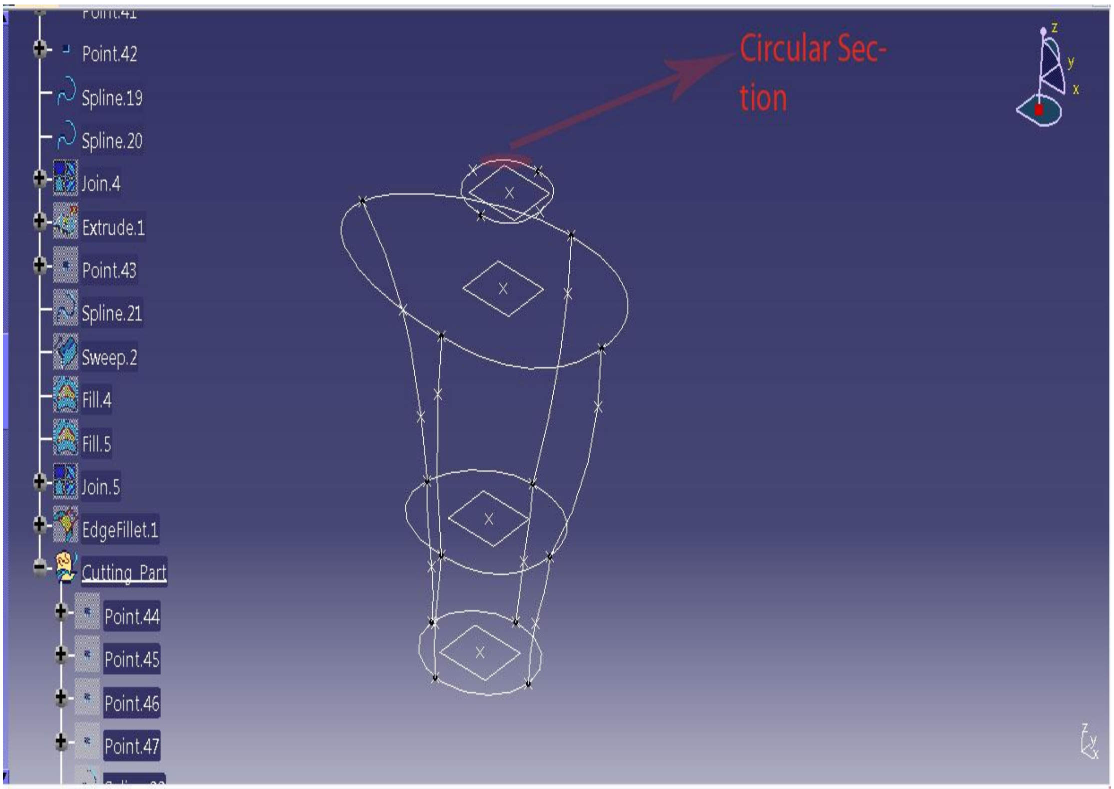
Picture 2.7.4.3: Multi section surface definition.



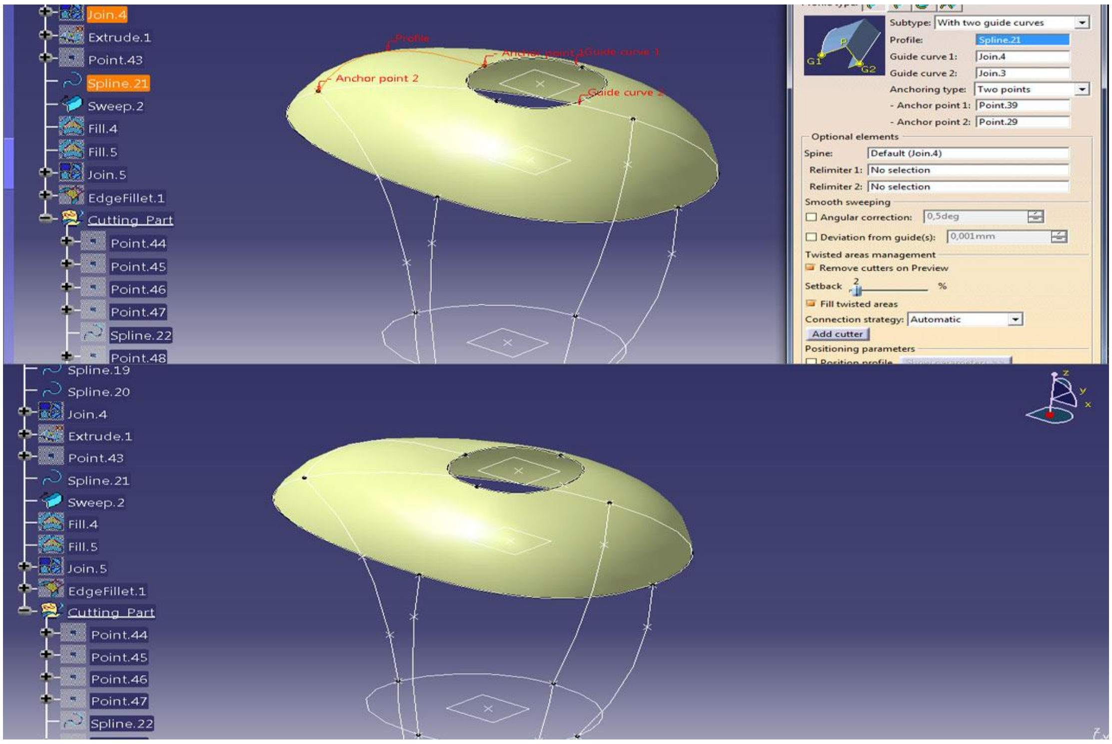
Picture 2.7.4.4: Knob's third wireframe section.



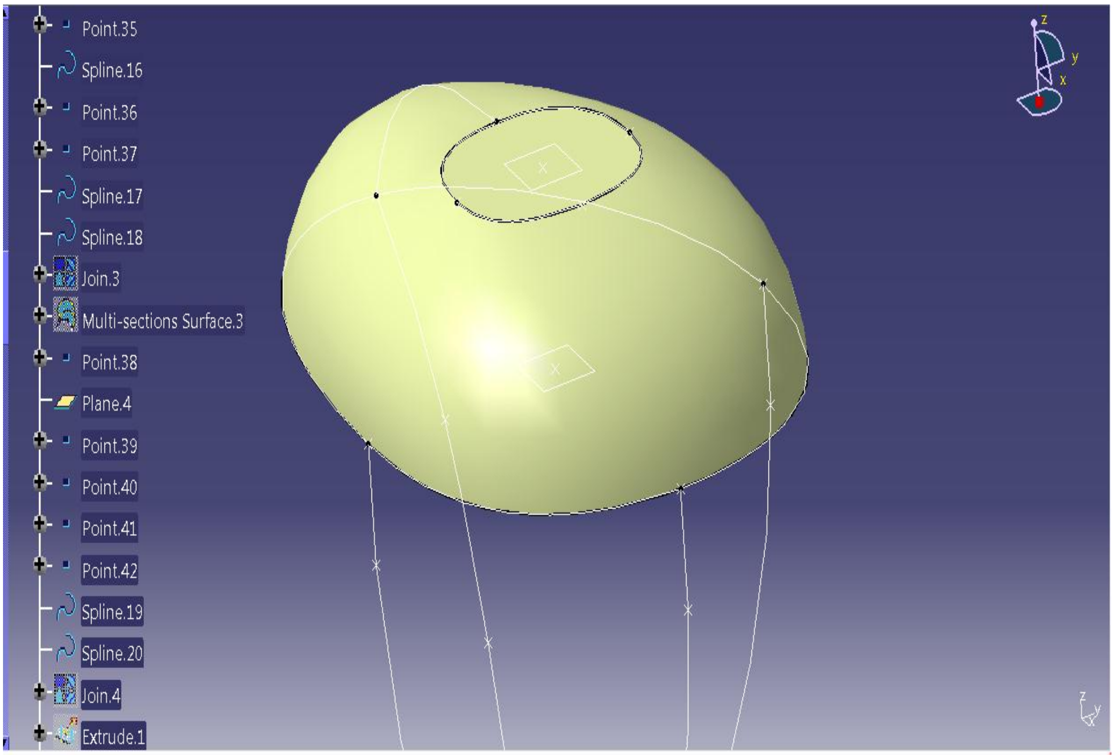
Picture 2.7.4.5: Creation of the second peripheral surface.



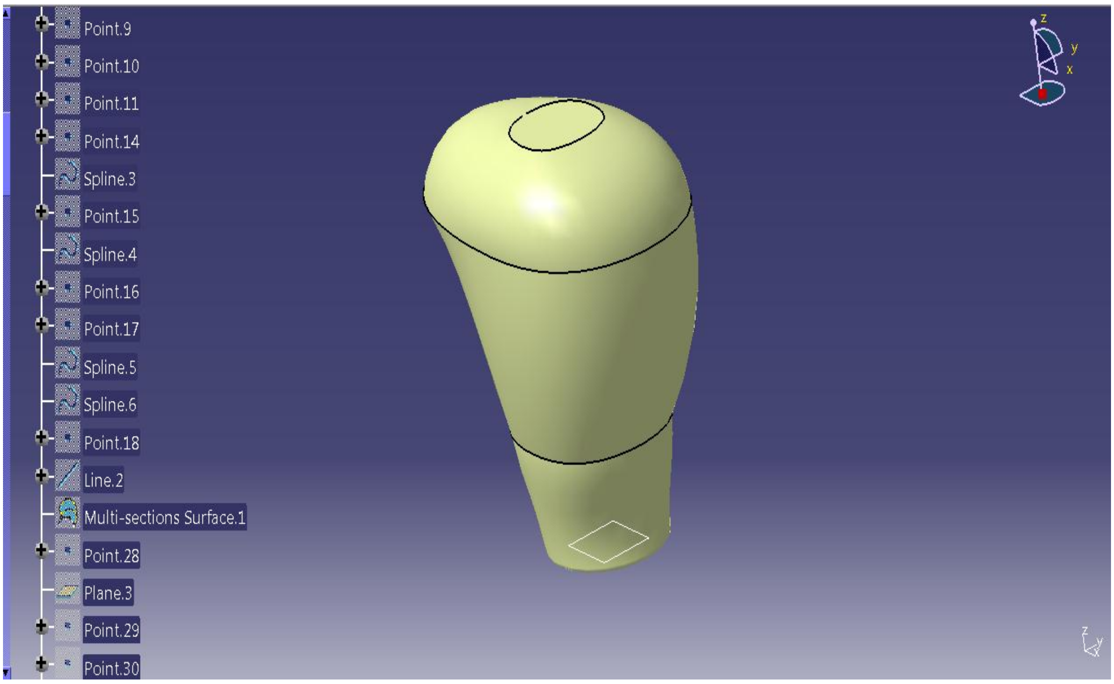
Picture 2.7.4.6: Circular section of knob's wireframe.



Picture 2.7.4.7: Knob's blended surface between the last two sections.



Picture 2.7.4.8: Knob's final fill surface.



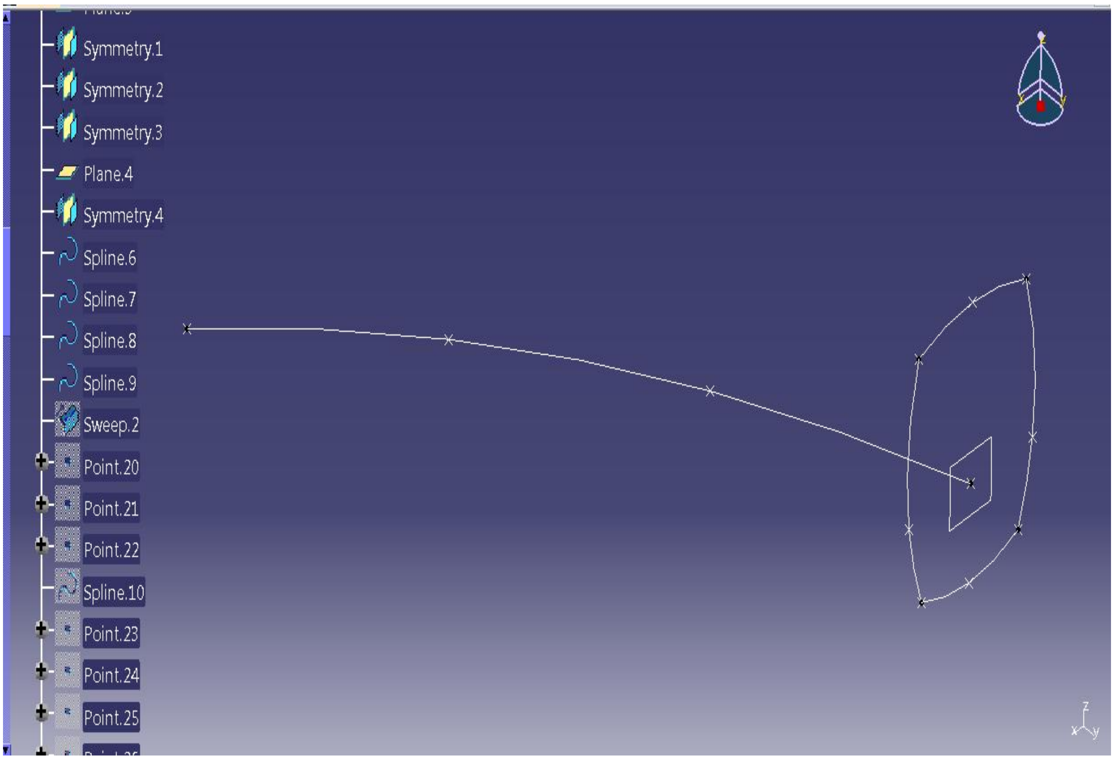
Picture 2.7.4.9: Knob complete part.

2.8 DESIGN OF HANDBRAKE LEVER

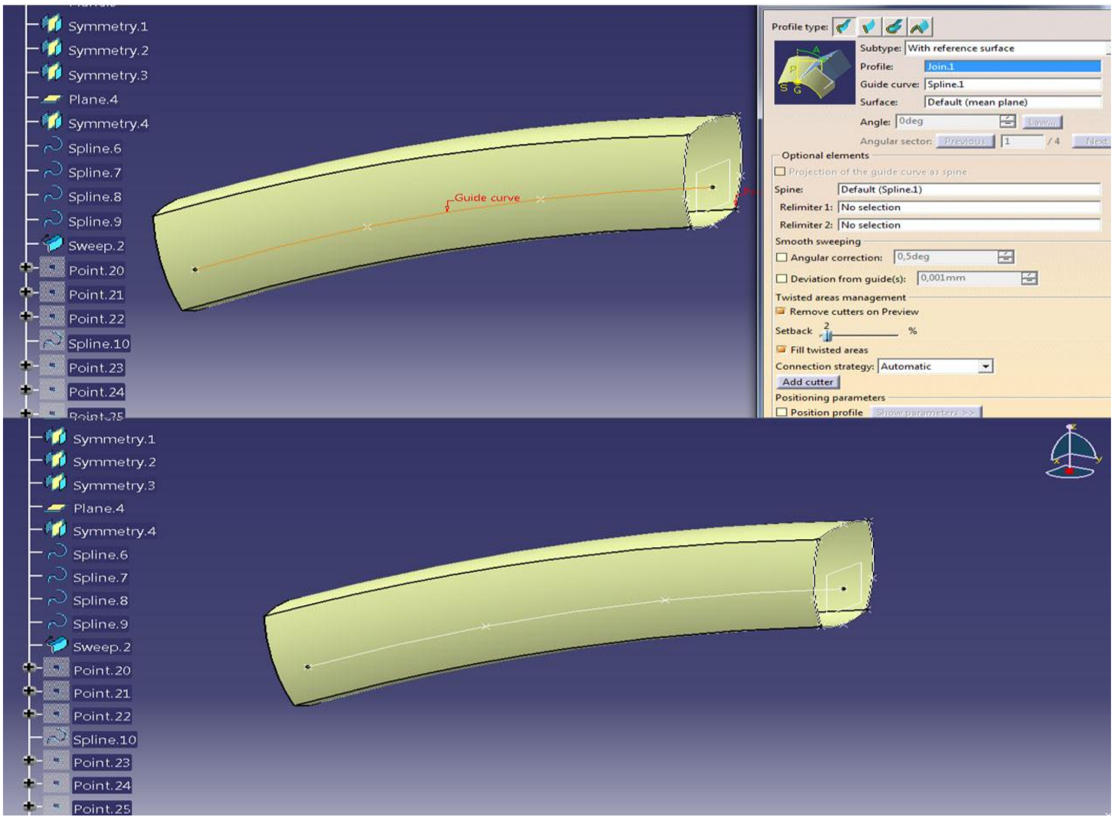
The handbrake part was an assembled part consisting of two elements. The first one was the primary handbrake lever with its boot attached on it. The second element was the knob that locks and unlocks the lever. Unlike the previous part, this one was created as solid.

2.8.1 DESIGN OF THE HANDBRAKE LEVER-PART ONE

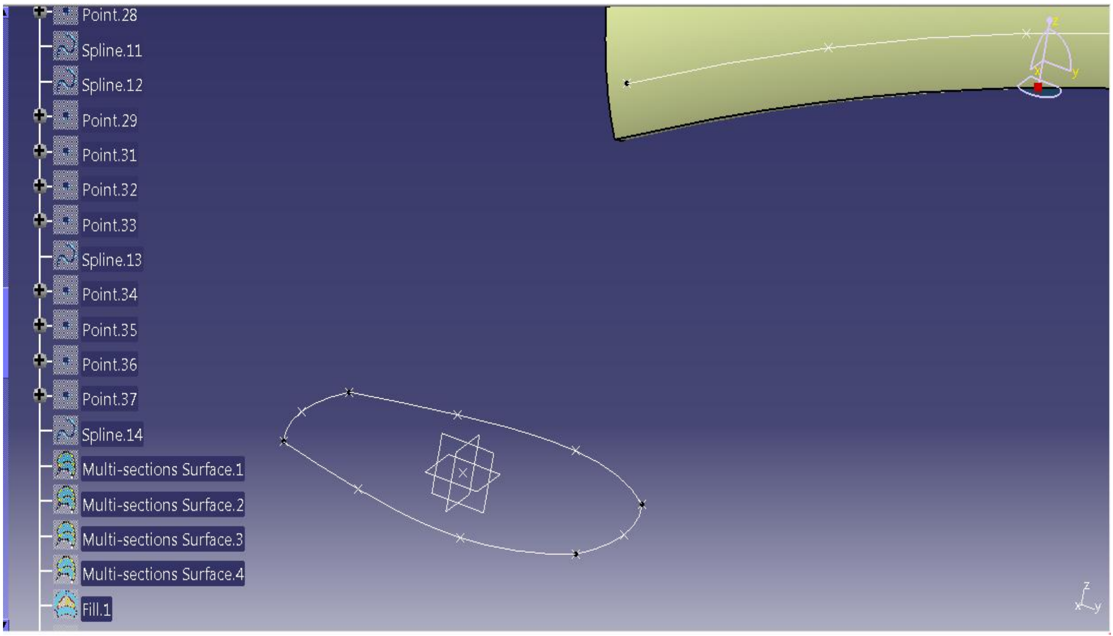
The design of the part started with the creation of the wireframe model (Picture 2.8.1.1). The first section consisted of four B-Splines and a single B-Spline curve. This single B-Spline curve functions as a guide for the creation of the handle's external surface. The external surface of the handle was created after the wireframe's creation (Picture 2.8.1.2). The surface was created as a sweep surface. For the execution of the "sweep" operation, the section's wireframe was used as a profile and the single spline curve was used as a guide curve. The handbrake's boot wireframe section was designed next. This section also contained four B-Spline curves and formed a closed section (Picture 2.8.1.3). The base section was connected with the handle's external surface with four Spline curves (Picture 2.8.1.4). The peripheral surface between the handbrake's boot and the handle's external surface was constructed as four separate multi-section surfaces. For the creation of each surface, two connective curves between the boot's wireframe and the handle's surface layout were designed as generative curves each time. As guide curves a segment of each wireframe of the boot's and handle's surface were imported respectively. The limits of each one of these segments were defined by using one of the two points of the generative curves that lied on each wireframe (boot's and handle's surface layout) as origin and as terminal respectively (Picture 2.8.1.5). An additional section was defined on the handle's surface, in order to approach an actual handbrake's geometry more accurately. This additional section was defined on a plane that was set with a certain degree of inclination (20 degrees). Same as for the other modules, this certain section was attached to the handle's surface with seven lines (Picture 2.8.1.6). Then, the peripheral surface between the new section and the rest handle surface was created as a multi-section surface (Picture 2.8.1.7). In order to close the gap that existed on the recently designed surface, a final section was designed. The shape of the final section was similar to a circular section (Picture 2.8.1.8). A ''blend'' surface was created between the two last sections, as shown in Picture 2.8.1.9. The gap that was formed on the last section was covered with a surface. This surface was created as a fill surface (Picture 2.8.1.10). After the completion of the surfaces, they were all unified by using the "*join*" operation command. The final solid volume was created by using the "*close surface*" command (Picture 2.8.1.11).



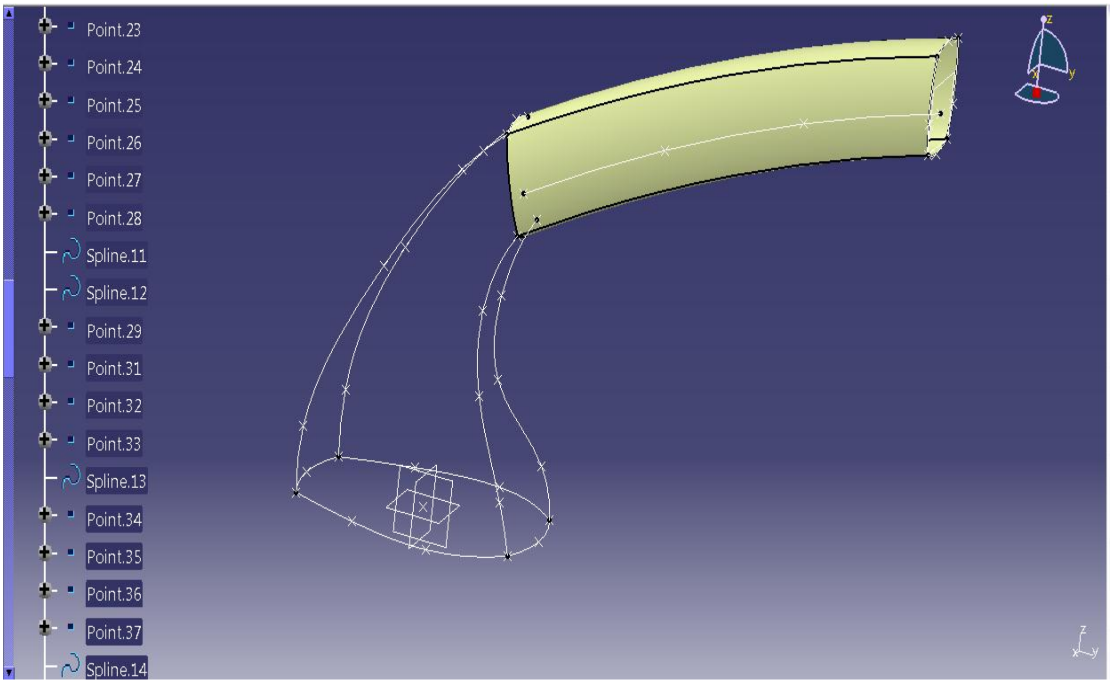
Picture 2.8.1.1: Handbrake's handle wireframe.



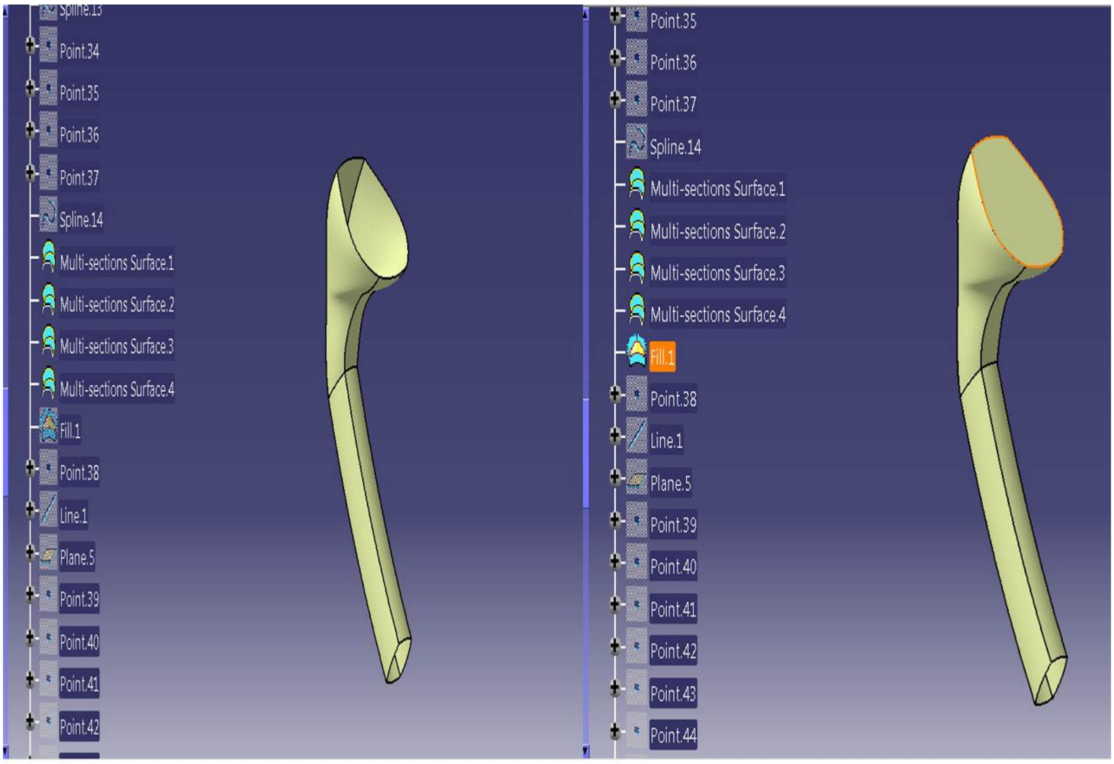
Picture 2.8.1.2: Handle’s external surface design.



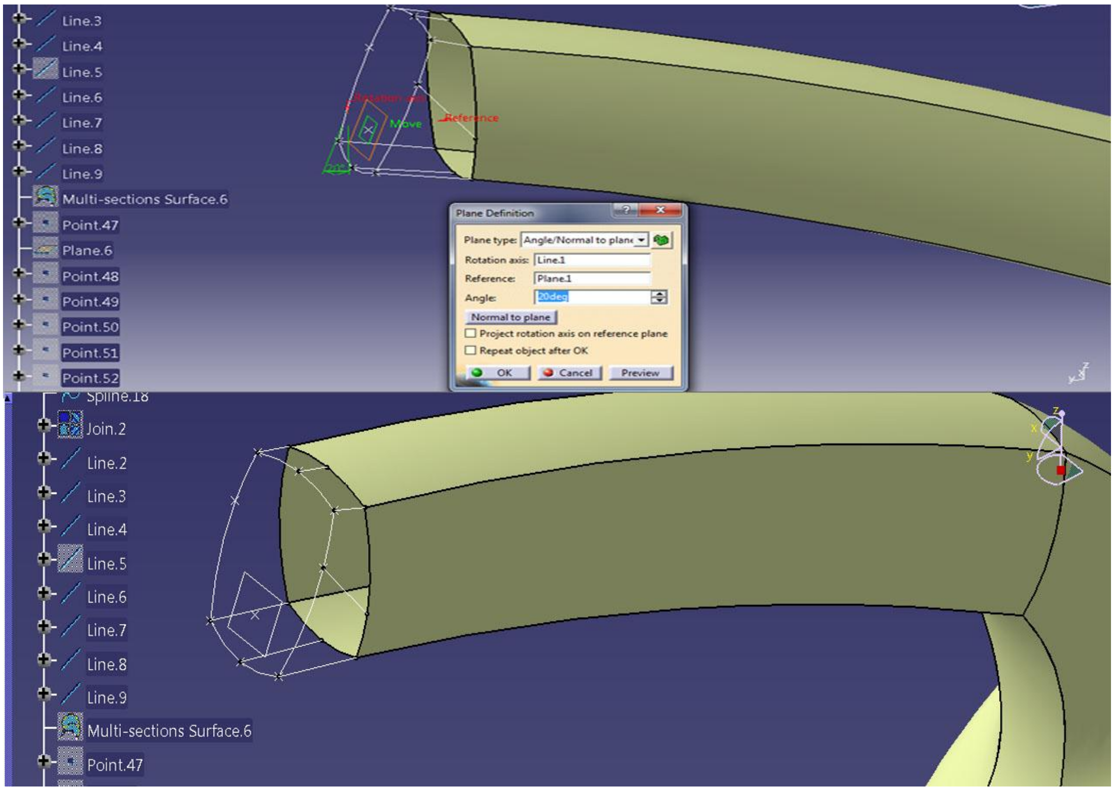
Picture 2.8.1.3: Handbrake’s boot wireframe.



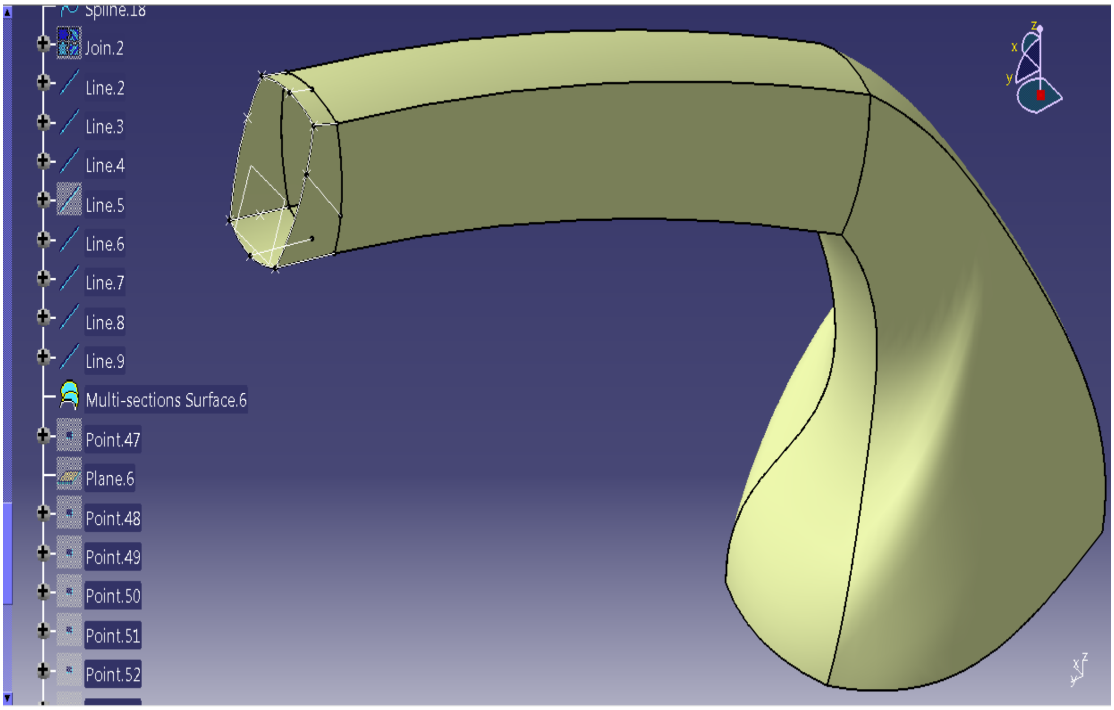
Picture 2.8.1.4: Definition of connective B-spline curves.



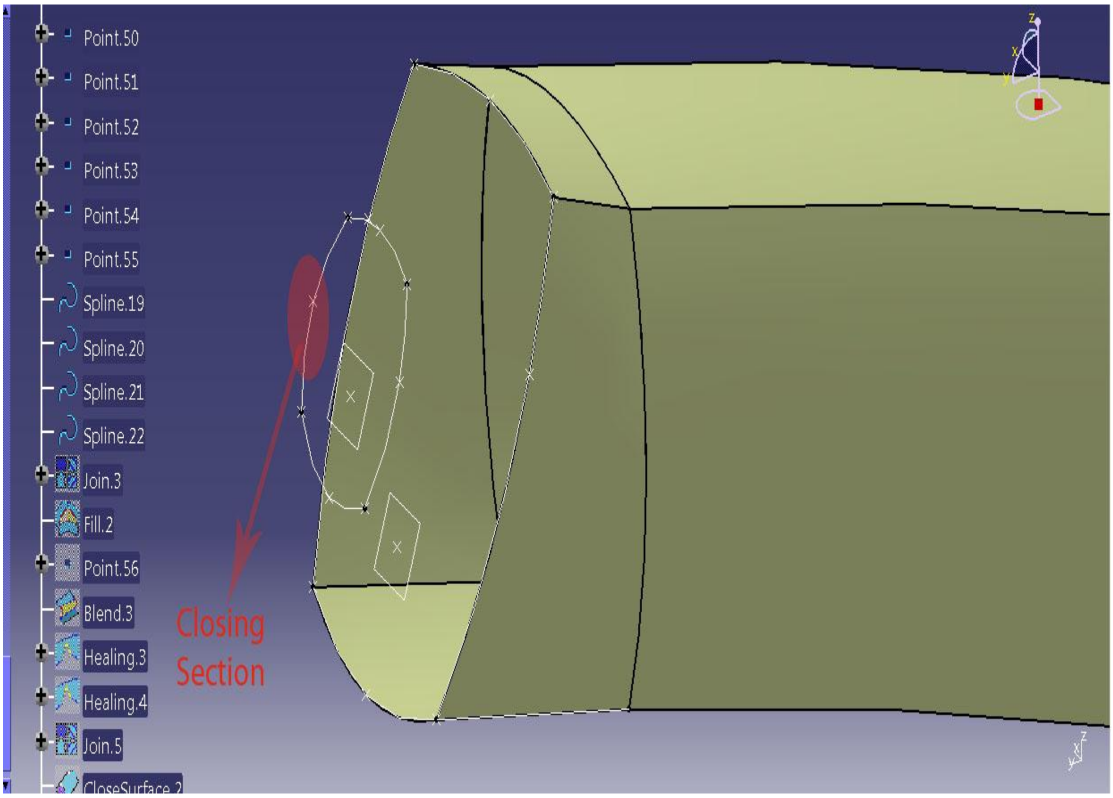
Picture 2.8.1.5: Creation of handbrake boot peripheral surface.



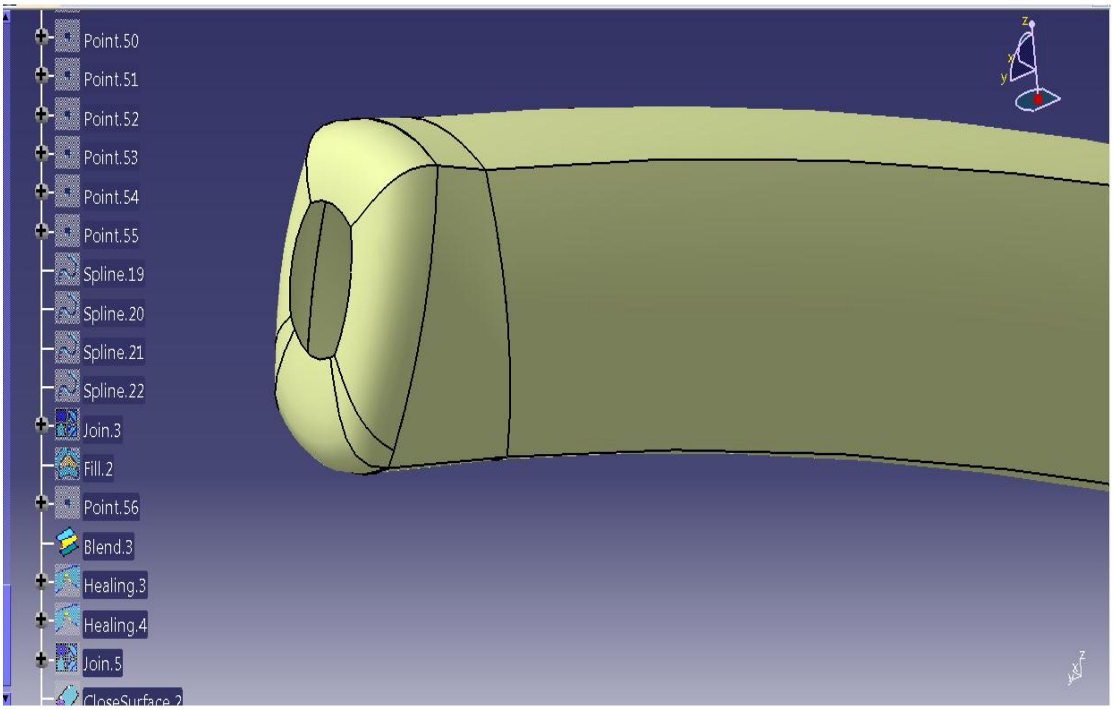
Picture 2.8.1.6: Handle’s additional wireframe.



Picture 2.8.1.7: Handle's additional multi section surface.



Picture 2.8.1.8: Handle's final wireframe section.



Picture 2.8.1.9: Definition of blended surface between the last two sections.

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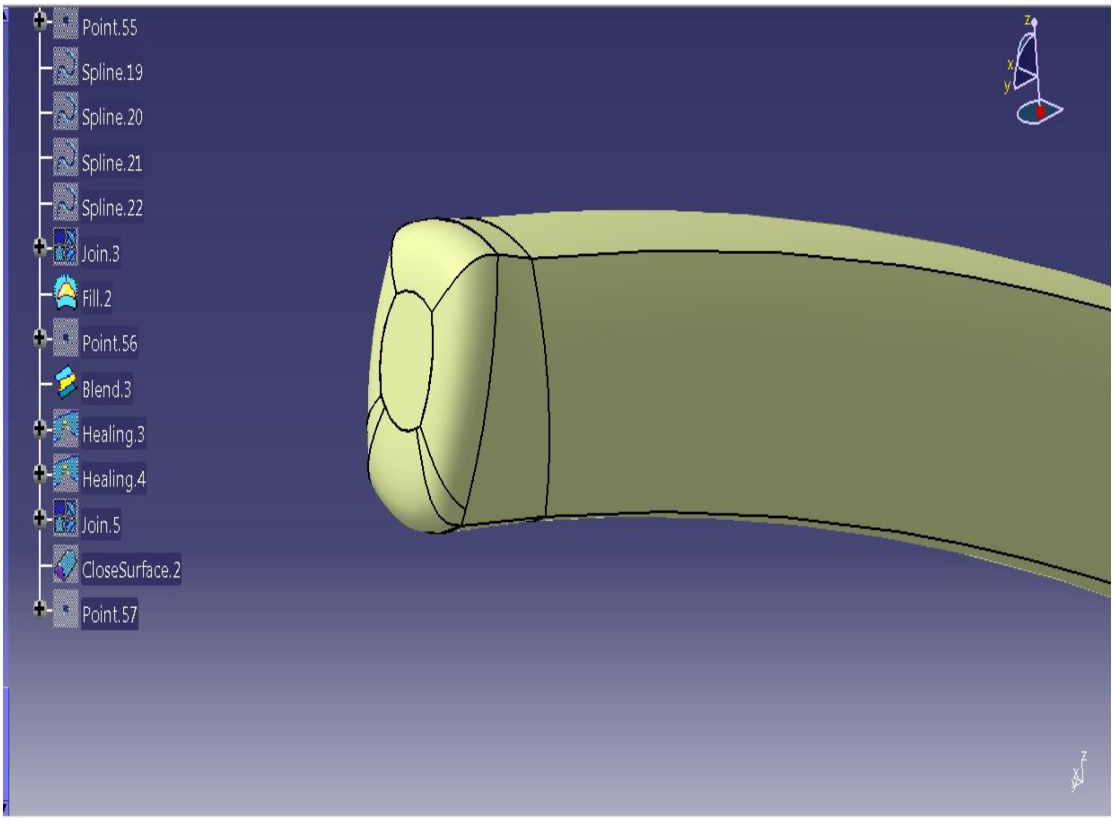
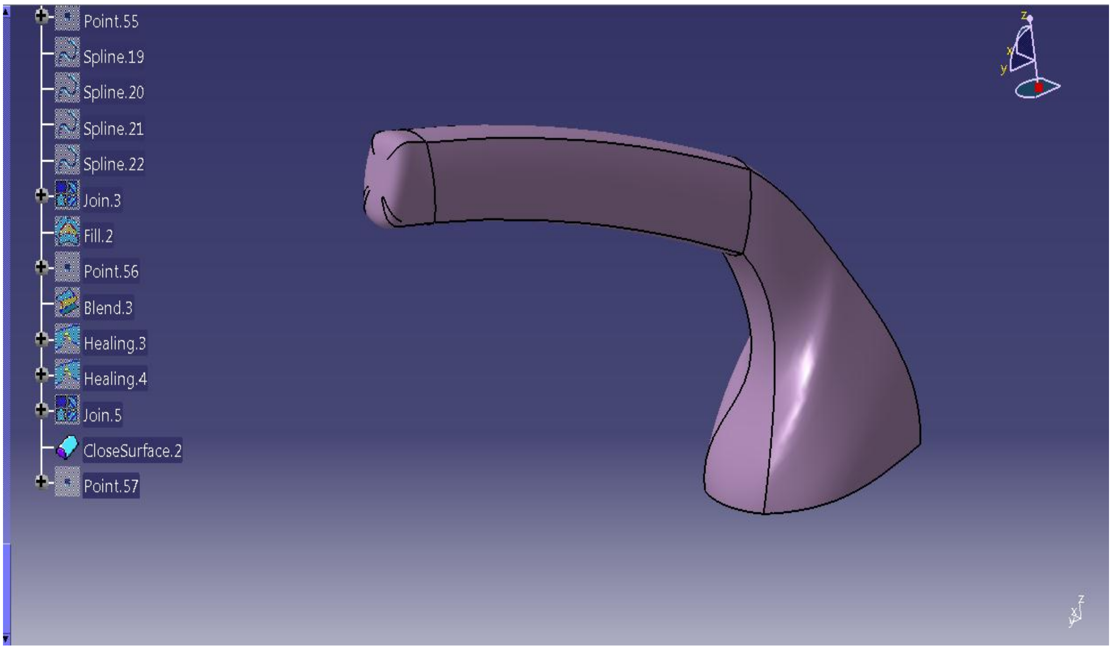


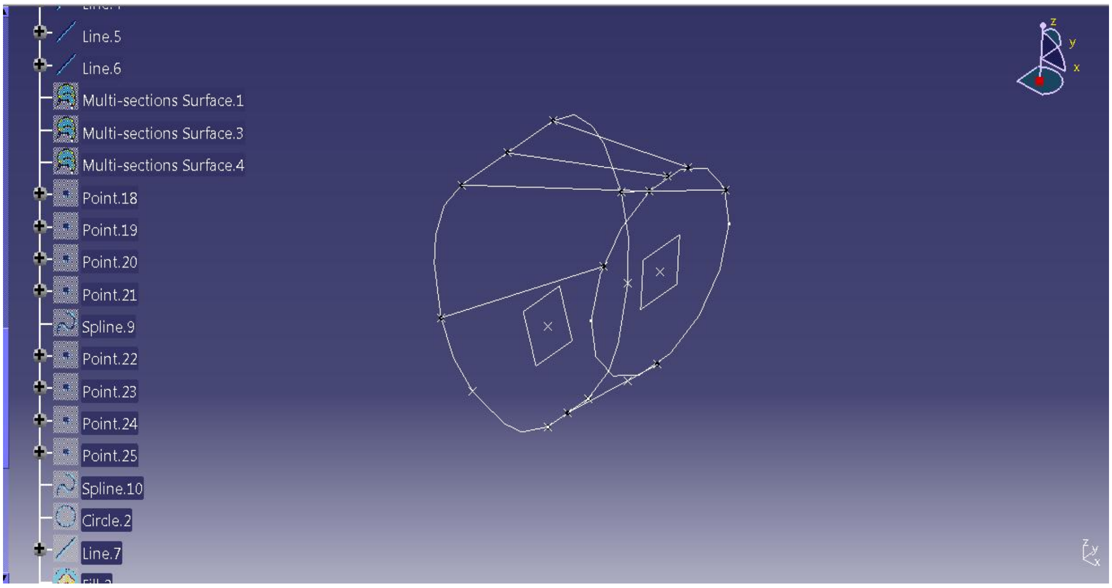
Picture 2.8.1.10: Complete handbrake's handle surface.



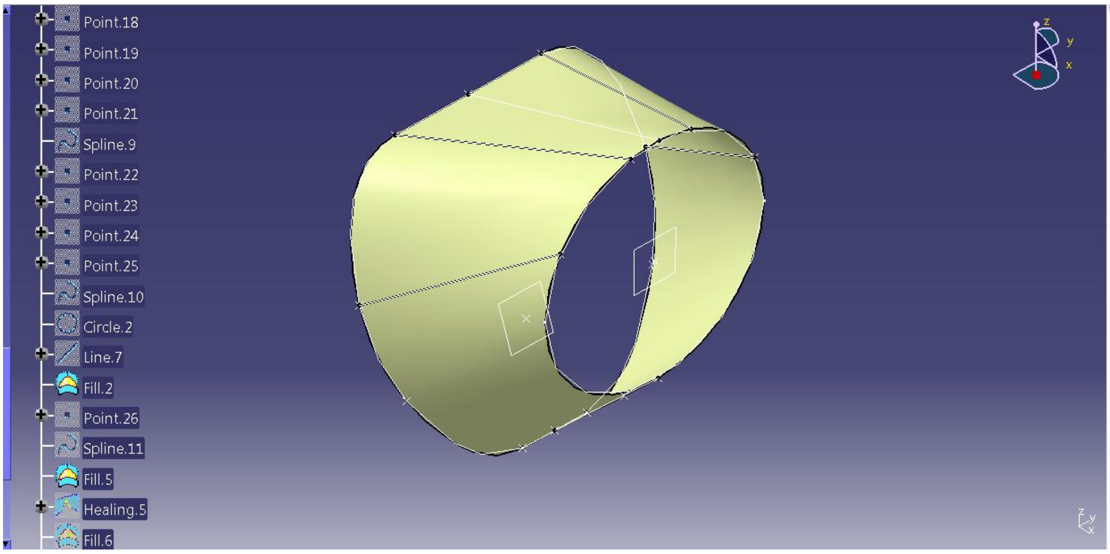
Picture 2.8.1.11: Handbrake’s solid volume.

2.8.2 DEFINITION OF HANDBRAKE’S ADJUSTING KNOB - PART TWO

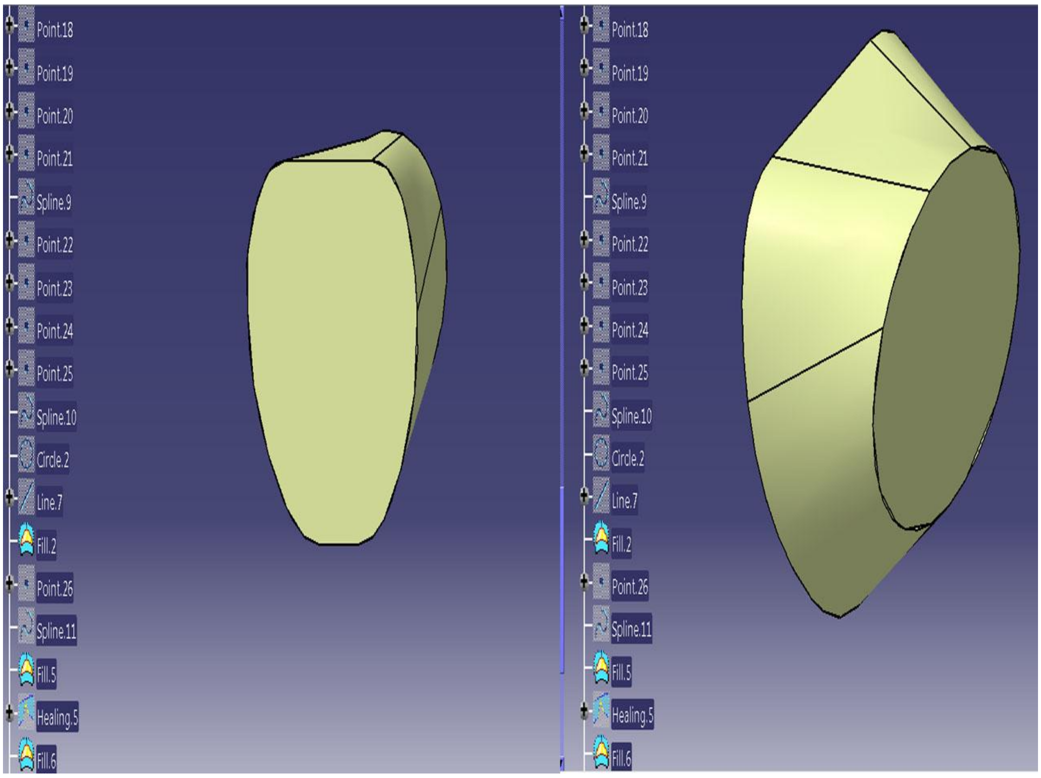
For the definition of this solid volume, the wireframe model was naturally created first. The model consisted of two sections, connected with each other with a certain number of lines (Picture 2.8.2.1). Firstly, the peripheral surfaces were created. Several segments were created as multi section surfaces and the rest as fill surfaces (Picture 2.8.2.2). The two gaps between the sections were closed using the “*fill*” surfaces command (Picture 2.8.2.3). Then, by using the “*close surface*” command, following the same process as for the other parts, the solid volume of the part was created (Picture 2.8.2.4). For the completion of the 3-D design two filletings were added on the sharp layouts of the front and back surface of the solid volume (Picture 2.8.2.5).



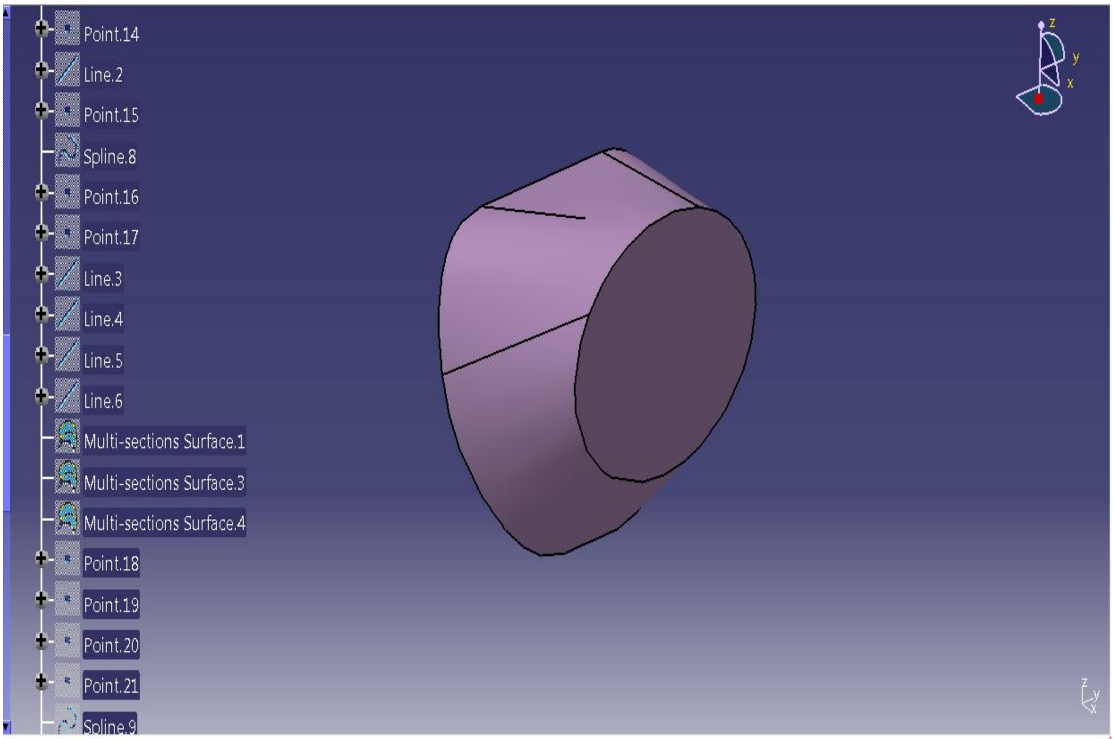
Picture 2.8.2.1: Handbrake's adjusting knob wireframe.



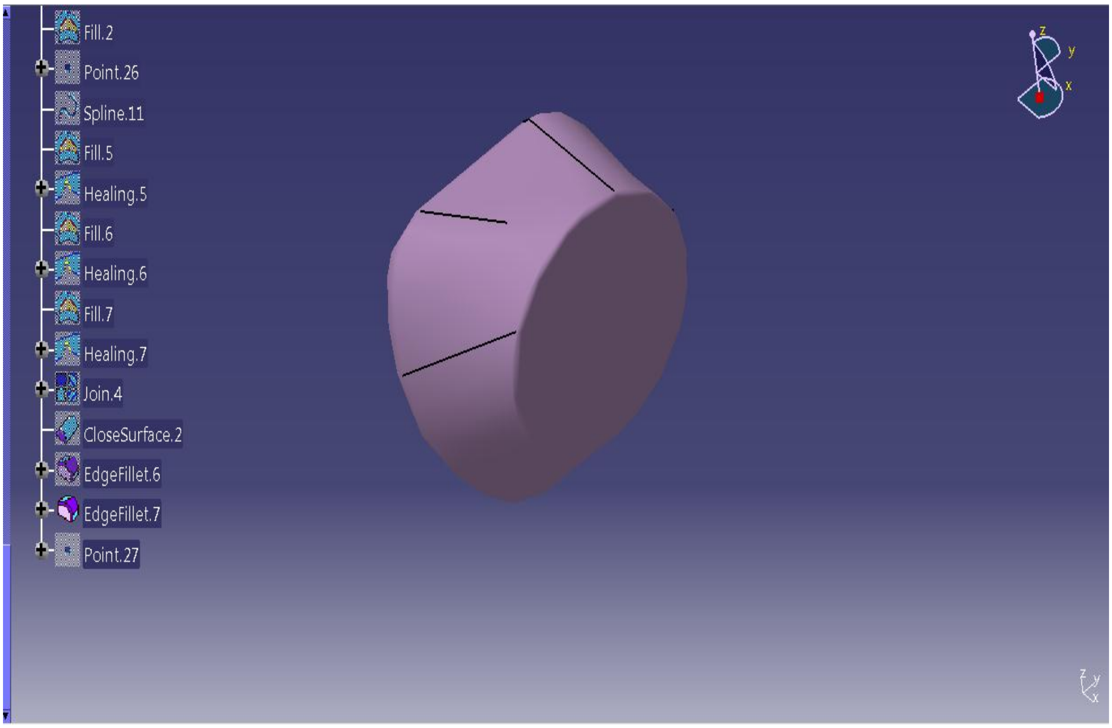
Picture 2.8.2.2: Handbrake's adjusting knob peripheral surfaces.



Picture 2.8.2.3: Handbrake’s adjusting knob complete set of surfaces.



Picture 2.8.2.4: Handbrake’s adjusting knob solid volume.



Picture 2.8.2.5: Adjusting knob final volume.

2.9 DESIGN OF THE THREE-DIMENSIONAL DOOR PANEL

This part functions as the primary vehicle insertion. It was designed by using real vehicle passenger door photos. As source of these photos official car manufacturer websites were used. Most of the material was also downloaded by using google search engine. As for the dimensions of the 3-D solid, they derived from measurements taken from a real urban vehicle passenger door.

Naturally, for the definition of the door solid volume, the wireframe model was created first, since it was the base for the implementation of the following stages of the design process. Firstly, a planar section was defined. The section consisted of two curvilinear B-Splines and two additive B-Splines, whose shape approaches the shape of a line (Picture 2.9.1). Two more rectangular sections, parallel to each other, were defined vertically to the first wireframe section. These rectangular sections were used as the base for the adjustment of the glass window frame (Picture 2.9.2). After the adjustment of the glass window frame base, the door panel wireframe was defined. The wireframe of the glass window frame lied on the same plane as the first wireframe section. The framework consisted of four B-Spline curves and two lines which function as a connection between this section and the panel's wireframe sections (Picture 2.9.3). Then a multi-section surface was defined on the first section of the wireframe model. As sections for the creation of the surface model the two curvilinear Splines of the wireframe model were used. As guide curves for the creation of the surface the two remaining Splines of the section were defined (Picture

2.9.4). Then, by joining the four B-Spline curves of the section with the use of "*Join*" operation, an additional extruded surface was created. The extruded surface was inserted by using the previous joined wireframe as profile, the xz - plane as reference plane for the definition of the extrusion's direction and by defining the length of extrusion as 30 mm (Picture 2.9.5). In order to fill the gap that was formed from the extruded surface, a multi-section surface similar to the first one was created. Two border curvilinear edges were used as generative curves while the remaining two edges were imported as guides (Picture 2.9.6).

Next in the design process was the creation of the glass window frame base surrounding surfaces. The six in total surfaces, were created as multi-section surfaces with the same methodology as before (Picture 2.9.7). Then, on the rear multi-section surface an additional close section was created. The points that constituted the wireframe model were used as points on a surface. Then four B-Spline curves were used to construct the close layout (Picture 2.9.8). Same as in the previous sections, the Splines of the current section were unified using the "*Join*" operation command in order to create a new unified curve. Then, by using the newly created close layout as input, an extruded surface was formed in the perpendicular direction of the zy plane (Picture 2.9.9). As a result of the creation of the extruded surface, an opening was formed on the front face of the geometry. For the purpose of covering this opening, an additional Multi-Section surface was defined. As inputs for the execution of the command the marginal curves that formed the previously created extruded surface's layout were used (Picture 2.9.10).

For purposes that will become clear later in the analysis, the extruded surface created two stages before was extended. This extension was constructed with the use of "*extrapolate surface*" command. For the execution of this command the surface's boundary, that was going to be extended, and the surface itself had to be defined. In this case, the upper straight curve of the extruded surface layout that was adjacent to the door's base surface was inserted as boundary (Picture 2.9.11). At the ''extrapolated'' frame of the extrapolation command window, the previously formed extruded surface was inserted. The last important command feature, that was needed to be defined, was the type of extrapolation. The only options required from the user by the command is the type of extrapolation and its length. The reason why the command doesn't require the definition of the extension's direction is because it uses by default the direction of the initial extrusion. The direction at the command's definition stage is represented by a tiny green arrow that appears right after the selection of the ''extrapolated'' surface (Picture 2.9.12). The extrapolation's length was defined as 5 mm. Then, the extrapolated surface was ''trimmed'' by the perpendicular surface that was intersected by the first. For the execution of the command, the two intersected surfaces had to be inserted into the command's window. Additionally, the side of the material to be removed had to be defined. The side chosen to be removed was the overriding part of the previously created extrapolated surface and was interior to the second intersected surface (Picture 2.9.13). While at first sight the execution of the trim operation seemed to be meaningless, it was used because it made possible the unification of the set of surfaces, except for those that belonged to the glass window frame. This technique is going to prove very useful later in our analysis, and especially at the stage of the solid volume creation.

After the unification of the surfaces two additional wireframe models were created on the glass window frame: a similar and coplanar one and a second one, parallel to the initial. The second wireframe model was defined using the ''parametric technique''. Each point belonging to the section had as reference the corresponding point of the initial wireframe while the distance between the two points was 25 mm (Picture 2.9.14). This technique enabled us to manipulate the alteration of the frame's dimensions more easily and effectively. Next in sequence was the creation of a final wireframe structure. The final wireframe model was parallel to the smaller section, on the plane where the initial section was designed (Picture 2.9.15). The technique used was similar to the one used for the previous section. The distance between the reference section and the current one was 45 mm (Picture 2.9.15), while the regional sides of the four sections were covered by surfaces. The total number of surfaces designed using the curves of the wireframe models was five multi-section and one fill surface (Picture 2.9.16).

An additional stage of surfaces was designed on the already existing one. Naturally, a wireframe model was designed first, consisting of three lines and being perpendicular to the yz plane. Furthermore, one more wireframe was defined to be adjacent to the door's exposed surface. Then a sweep surface was created, using as a profile the perpendicular wireframe model and as guide curve the previously created wireframe direction (Picture 2.9.17). As shown in Picture 2.9.18, the tube trim, that derived from the sweep operation, had two openings. These two openings were closed by two multi-section surfaces (Picture 2.9.19). By using the edges of the previously created sweep surface a number of extruded surfaces was also defined. The direction of the surfaces was perpendicular to the yz - plane and their length was 100 mm (Picture 2.9.20). As before, the opening that derived from the construction of the four extruded surfaces was covered with a fill surface. As inputs for the execution of the command the contour edges of the extruded surfaces were used (Picture 2.9.21). Then, by using the same ''trim'' technique as before, the surfaces of the third stage were able to be connected to the set of the remaining surfaces. At this point it is important to note that the tube shaped surface of the third stage was not connected to the rest of the surfaces (Picture 2.9.22).

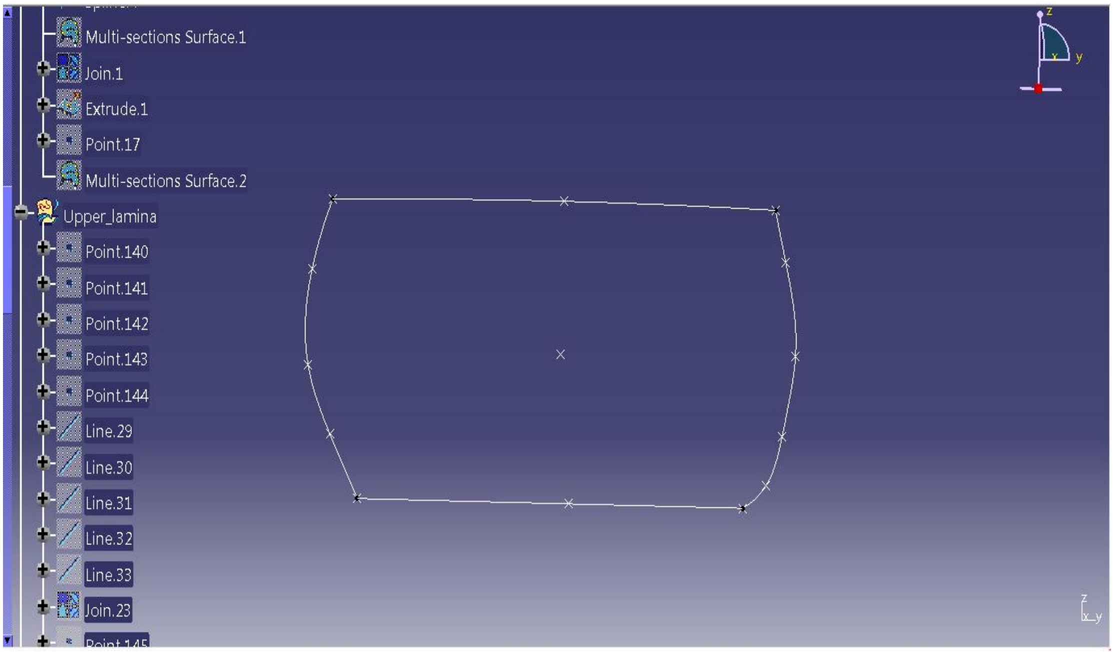
As it is shown from the door's complete surface model, the 3-D design is completely planar (Picture 2.9.22). Thus, in order to approach a real passenger three dimensional door panel design, this planar structure should somehow be curved. For this certain procedure the Catia software includes a certain command, which is called '*'wrap curve*''. In order for this command to be executed it is needed that two wireframe models to be designed. These wireframe models correspond to the initial and final shape of the surface model (Picture 2.9.23). Firstly, the previously combined set of surfaces was deformed by executing the wrap curve command using as reference shape the line 54 and as target shape that of Spline 63 (depicted in Picture 2.9.24). By executing the same "*wrap curve*" command for the remaining tube-shape surface the door panel of the passenger door was completed (picture

2.9.25). Next, the same wrap curve command was executed on the glass window frame. This time as reference and target shape the line 53 and the circle 2 (circular segment) were selected respectively (Picture 2.9.26 below).

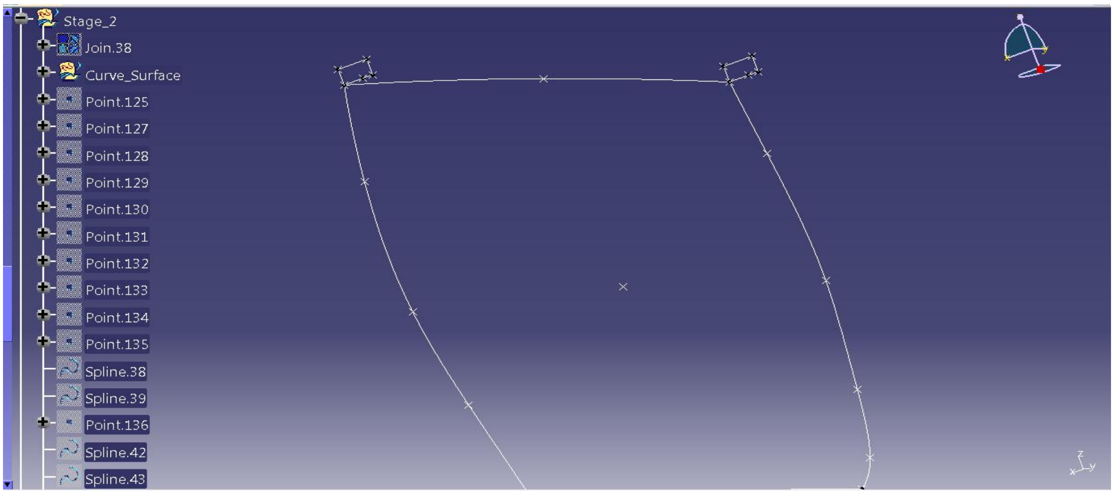
Finally, with the execution of "*close surface*" command, the final solid volume of the door design was created (Picture 2.9.27). Several edge fillets were inserted on the sharp edges of the door, primarily to fulfil the design's aesthetic criteria (Picture 2.9.28). The main dimensions measured on an existing passenger door were the door's total length, total height, as well as the window's framework height. The exact numerical values of the measured dimensions are displayed at the table below.

Table 2.9.1 Dimensions Measurements

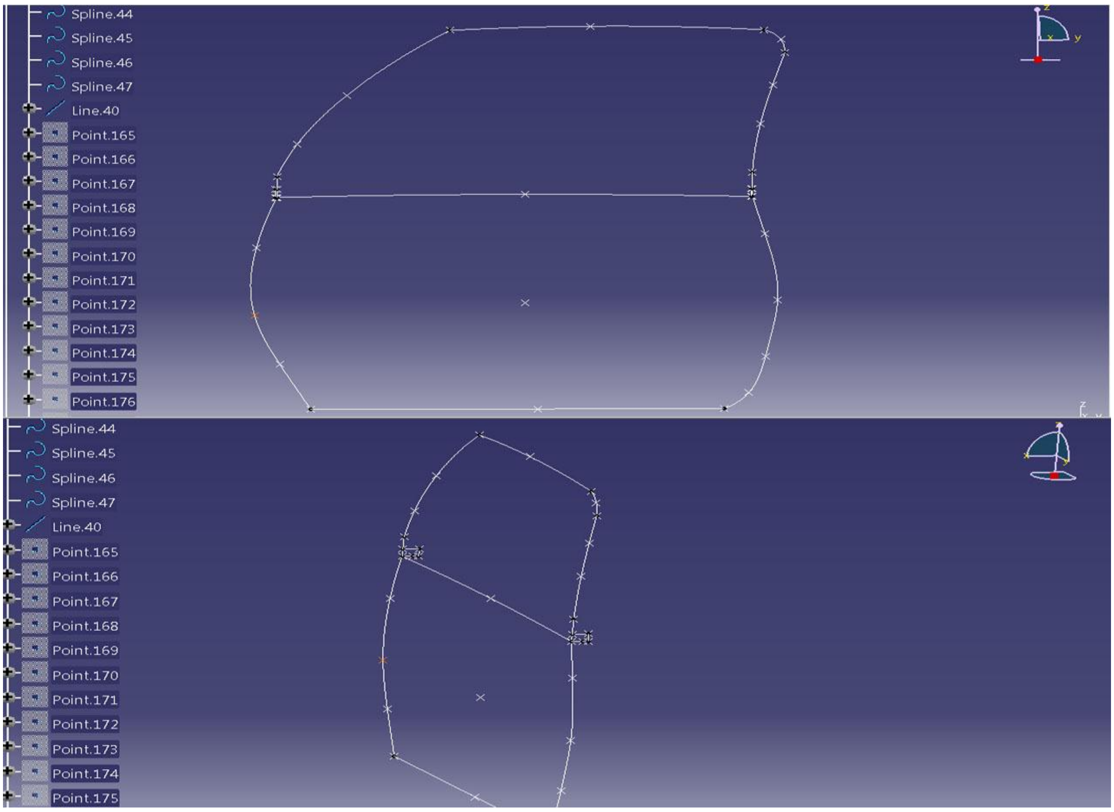
Dimensions	Numerical Values (mm)
Total Length	1160
Total Height	1120
Window Framework Height	520



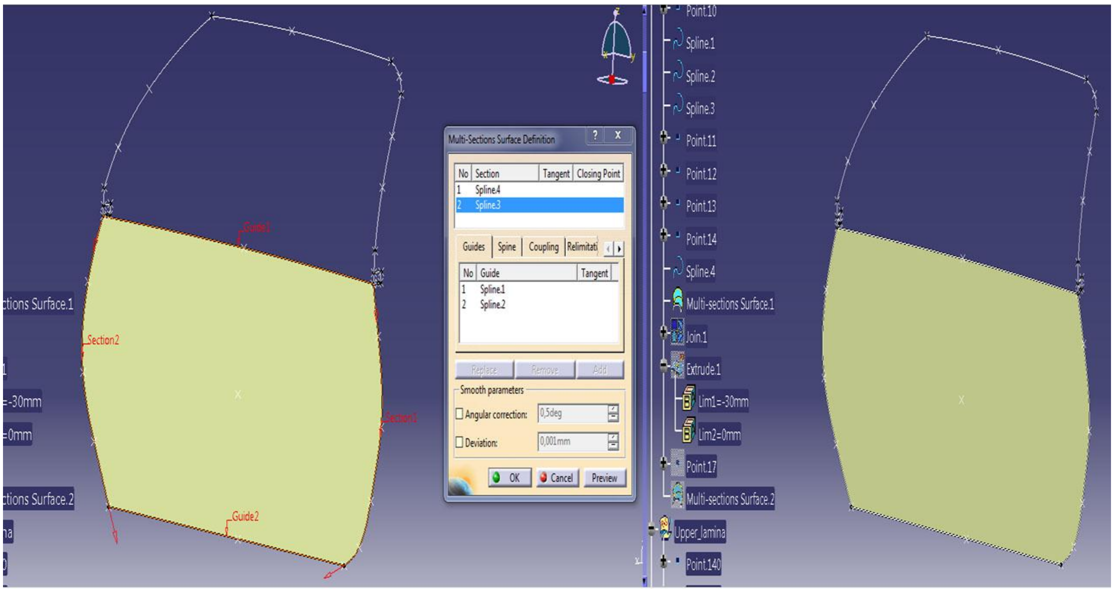
Picture 2.9.1: Passenger door panel initial wireframe.



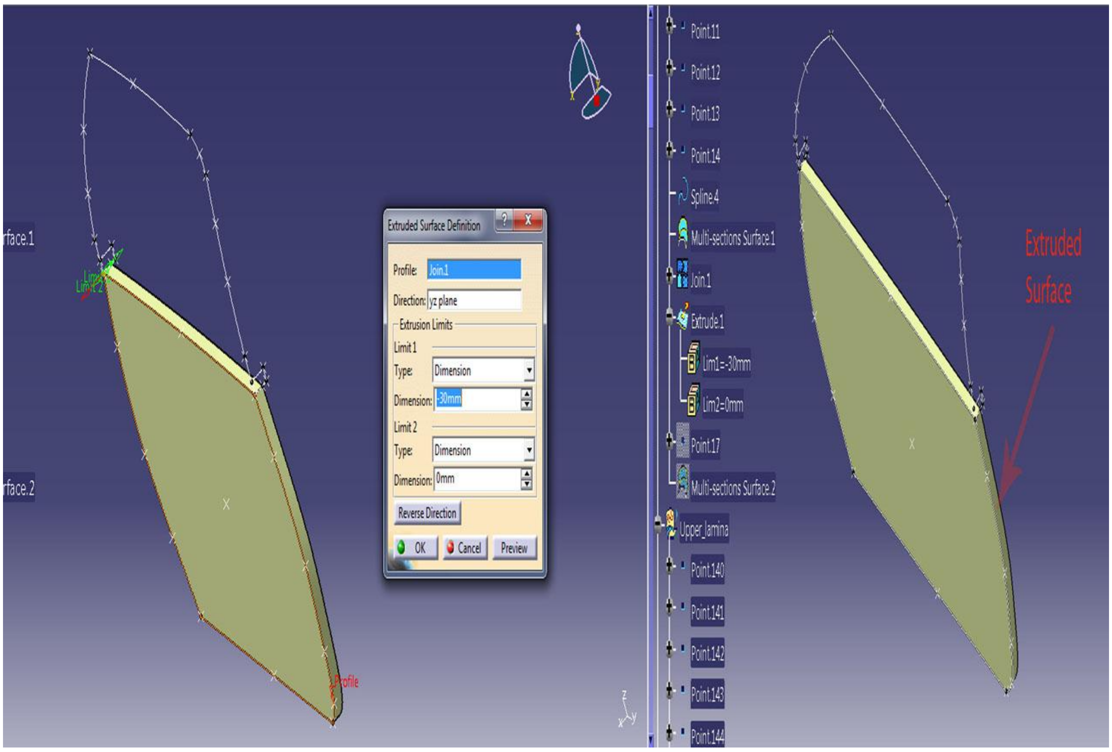
Picture 2.9.2: Part of door panel wireframe section that constitutes the base for the adjustment of the glass window frame.



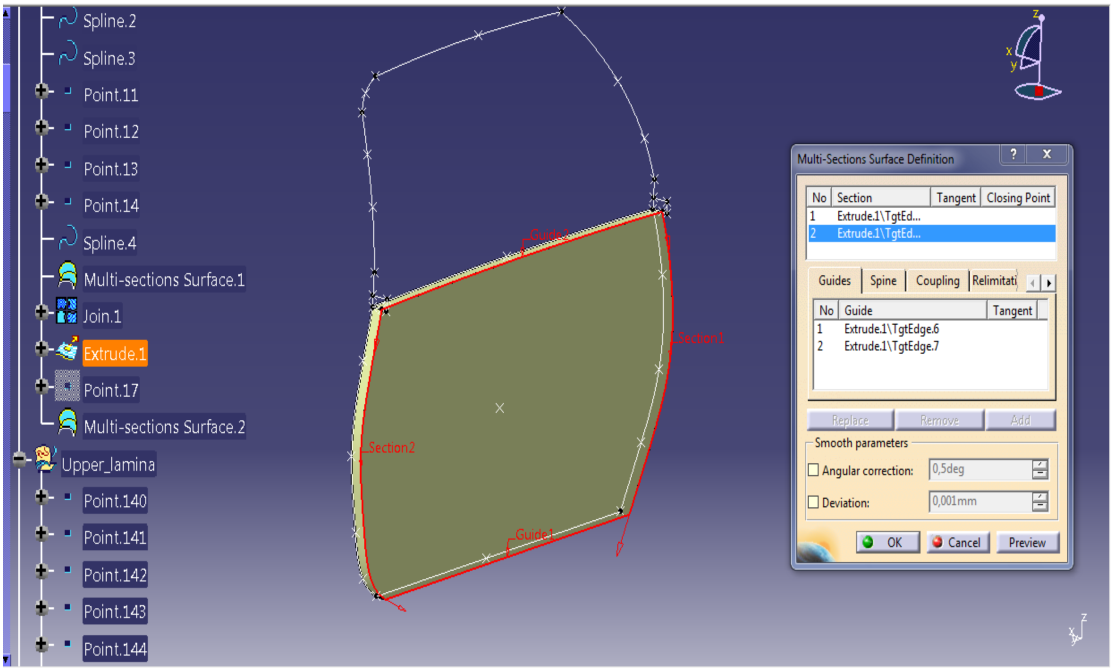
Picture 2.9.3: Passenger door panel complete wireframe.



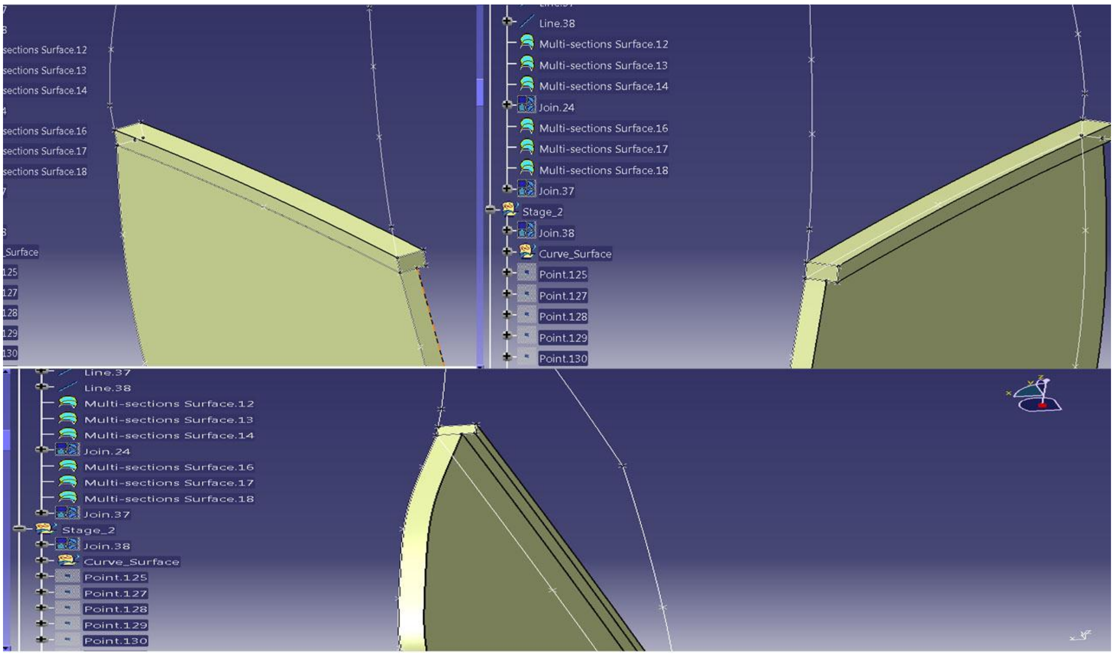
Picture 2.9.4: Creation of door panel first multi section surface.



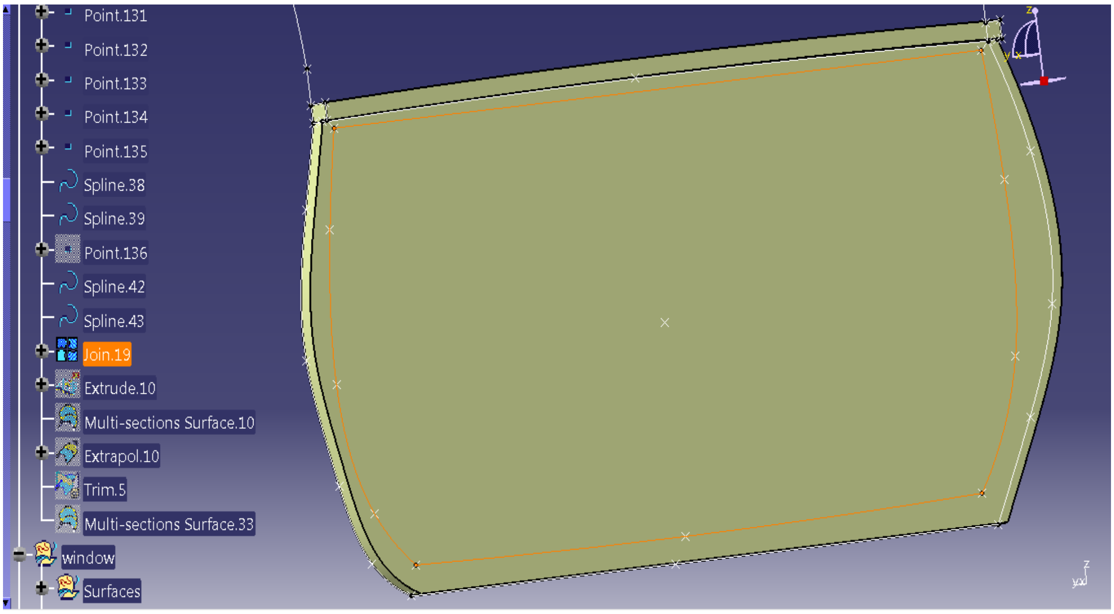
Picture 2.9.5: Door panel surface of extrusion.



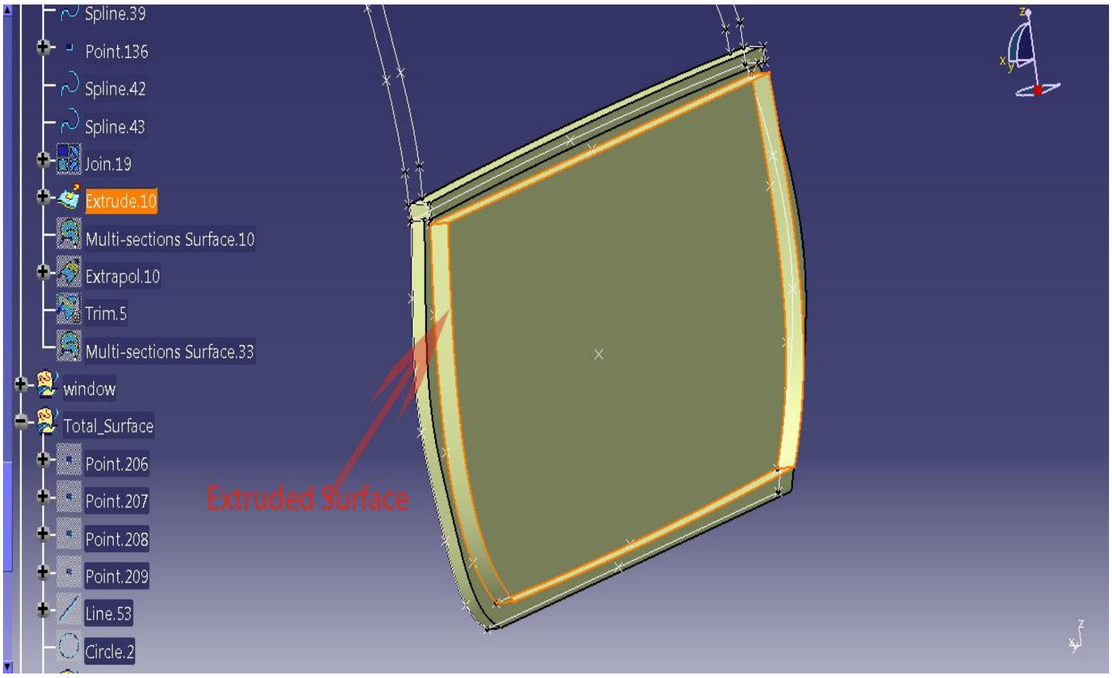
Picture 2.9.6: Door panel second multi section surface.



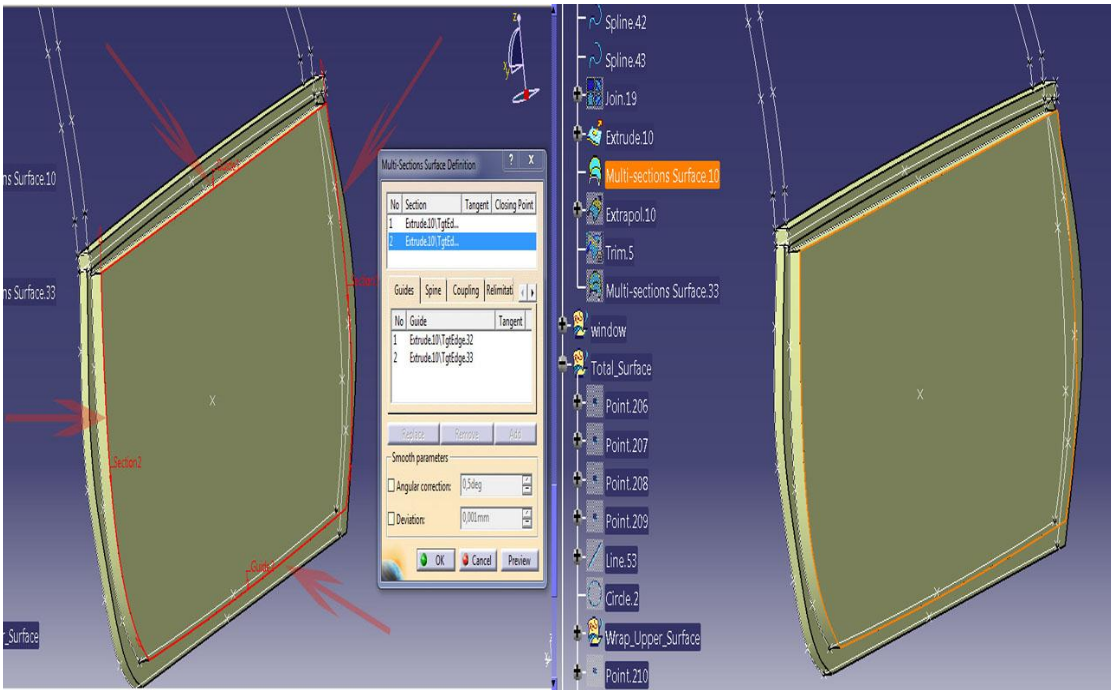
Picture 2.9.7: Panel surface that constitutes the base for the creation of glass window frame.



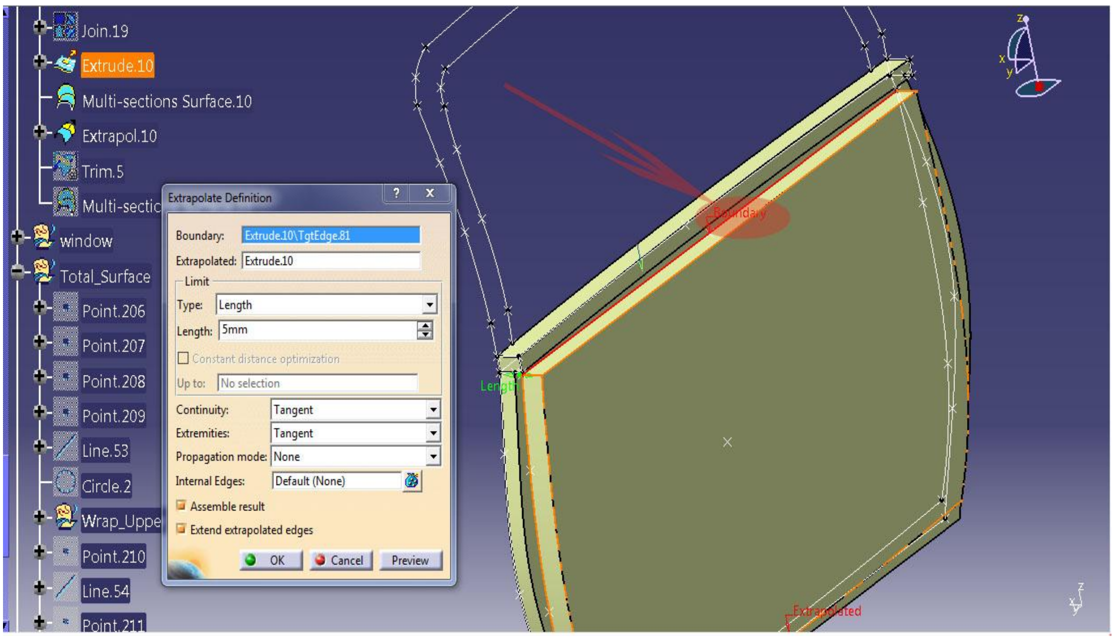
Picture 2.9.8: Additional door panel trim wireframe section.



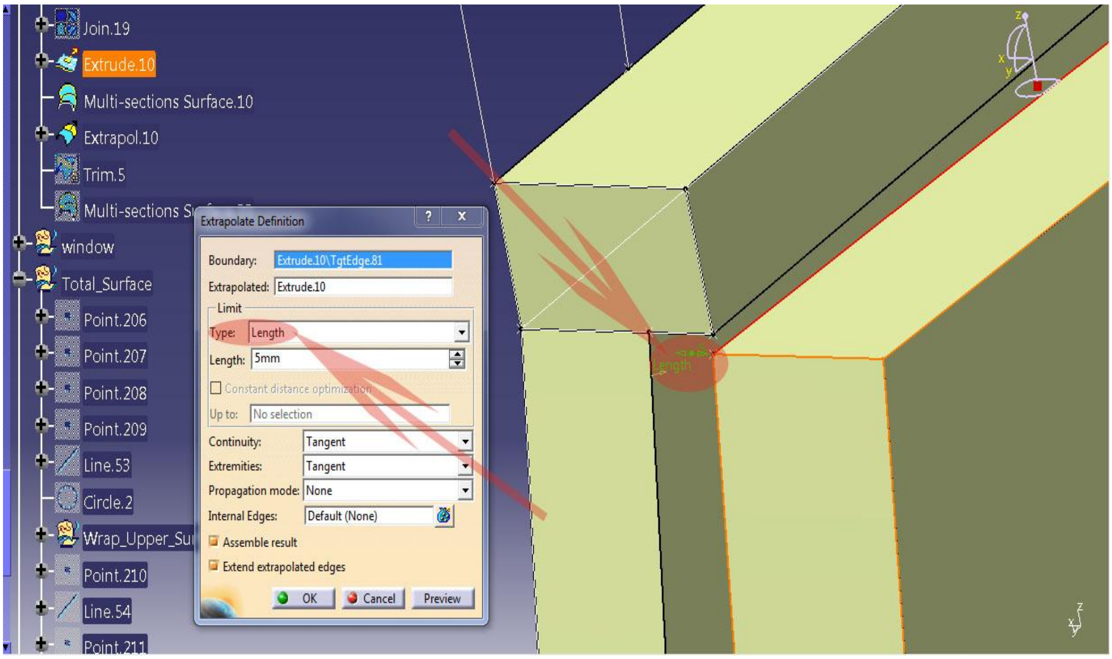
Picture 2.9.9: Creation of surface of extrusion.



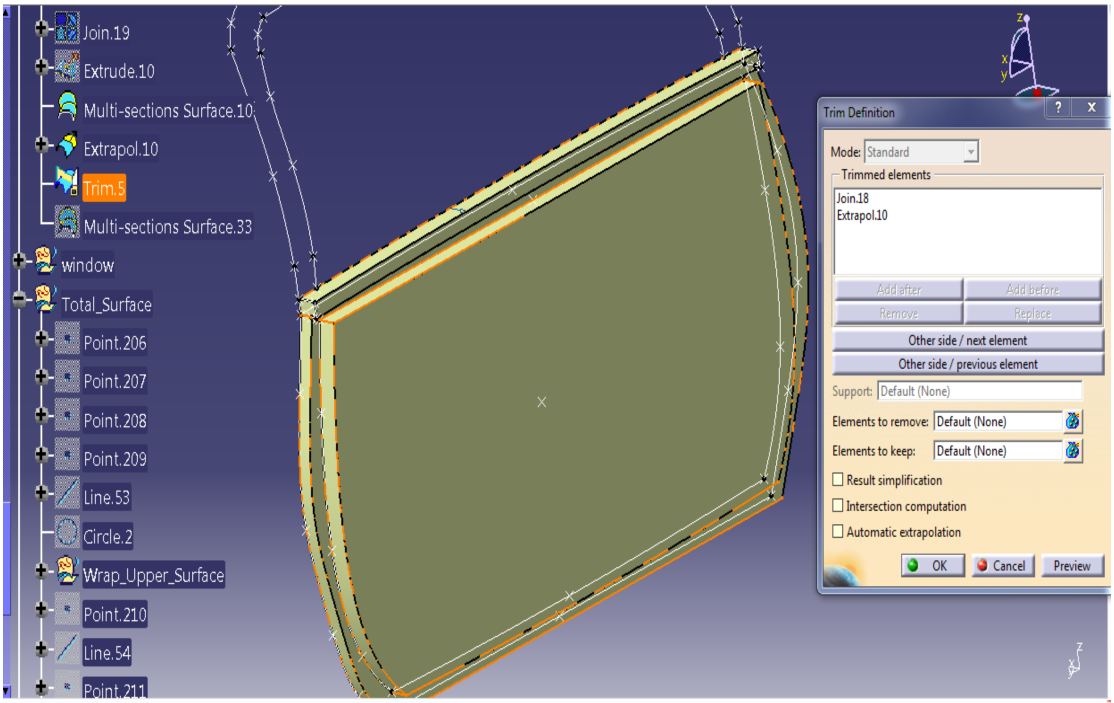
Picture 2.9.10: Creation of trim's multi-section surface.



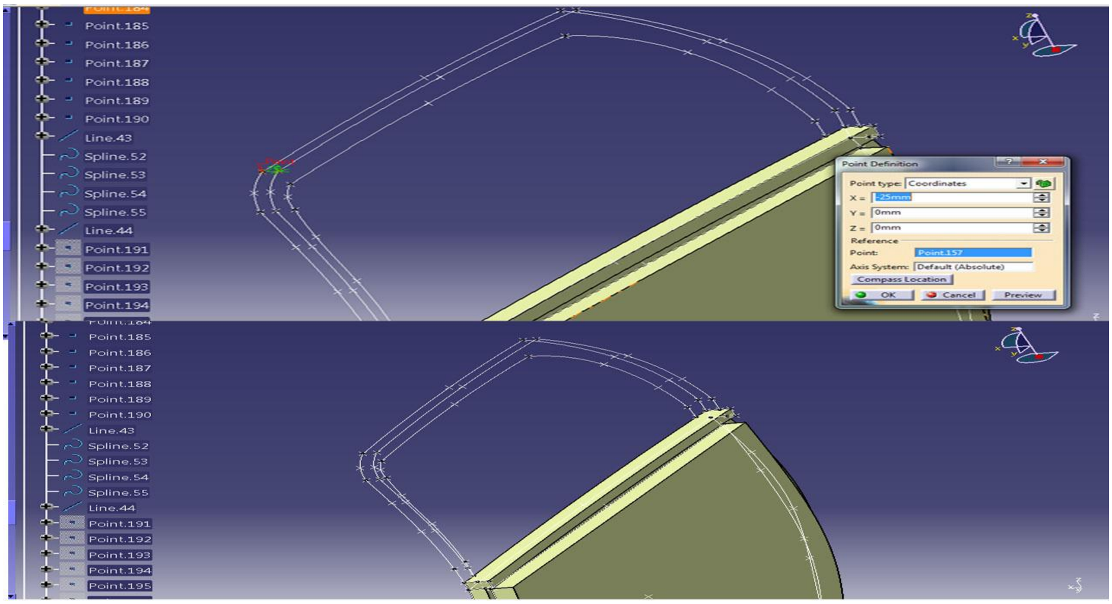
Picture 2.9.11: Execution of "*extrapolation*" command – boundary definition.



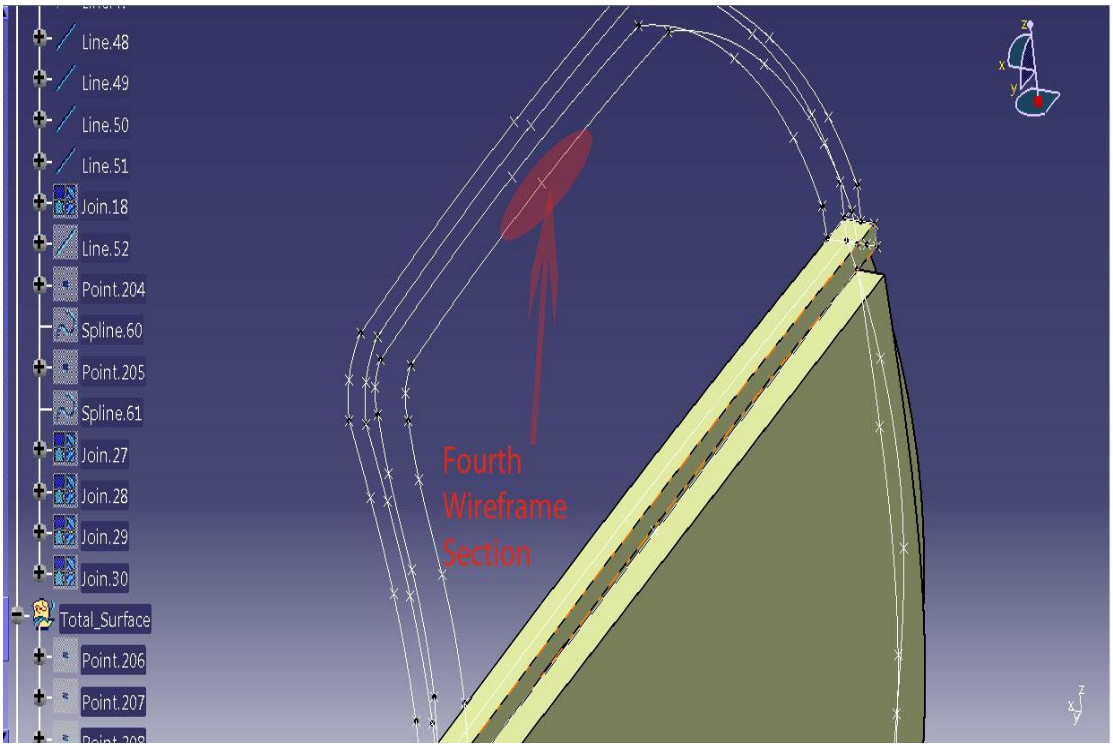
Picture 2.9.12: Execution of “*extrapolation*” command – definition of extension’s direction.



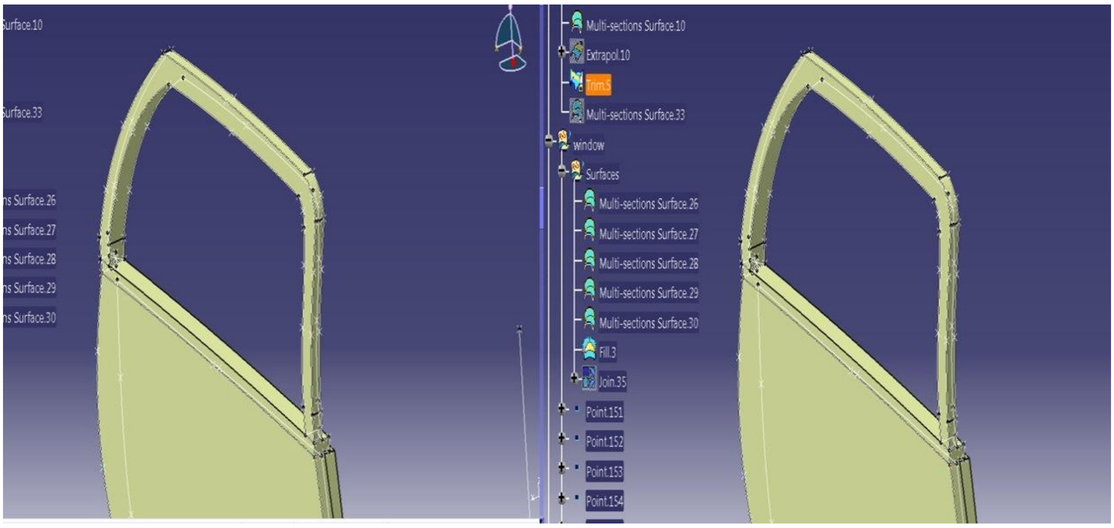
Picture 2.9.13: Definition of trim operation parameters.



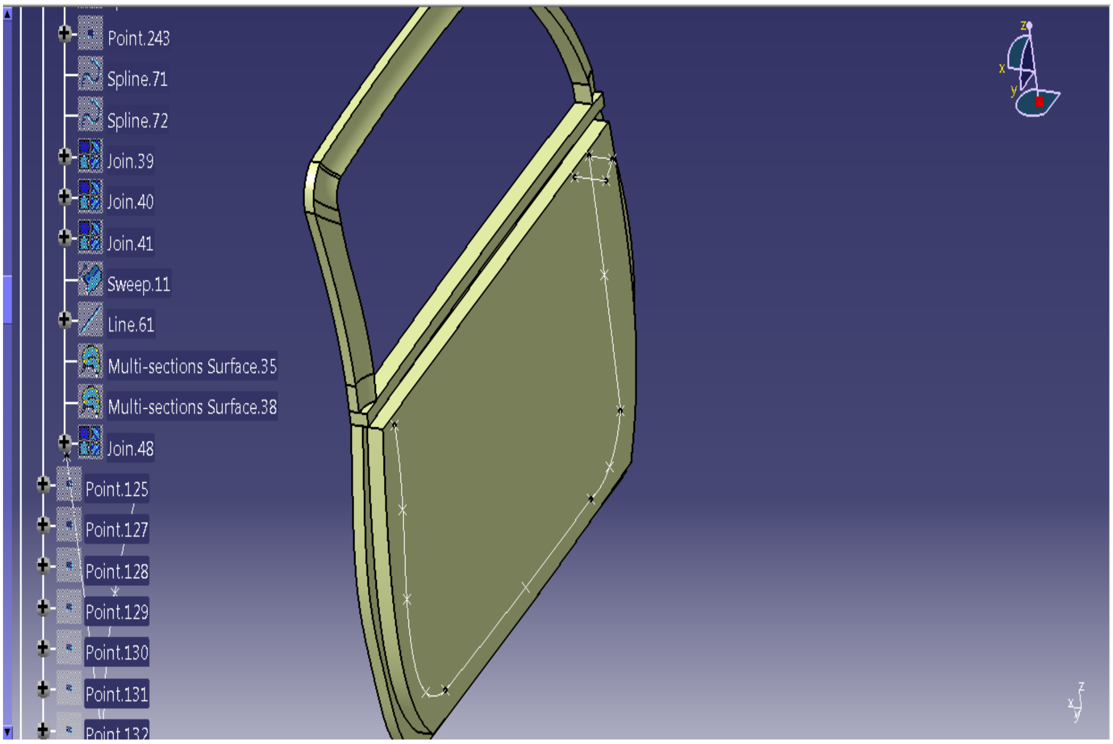
Picture 2.9.14: Adjustment of glass window frame wireframe model.



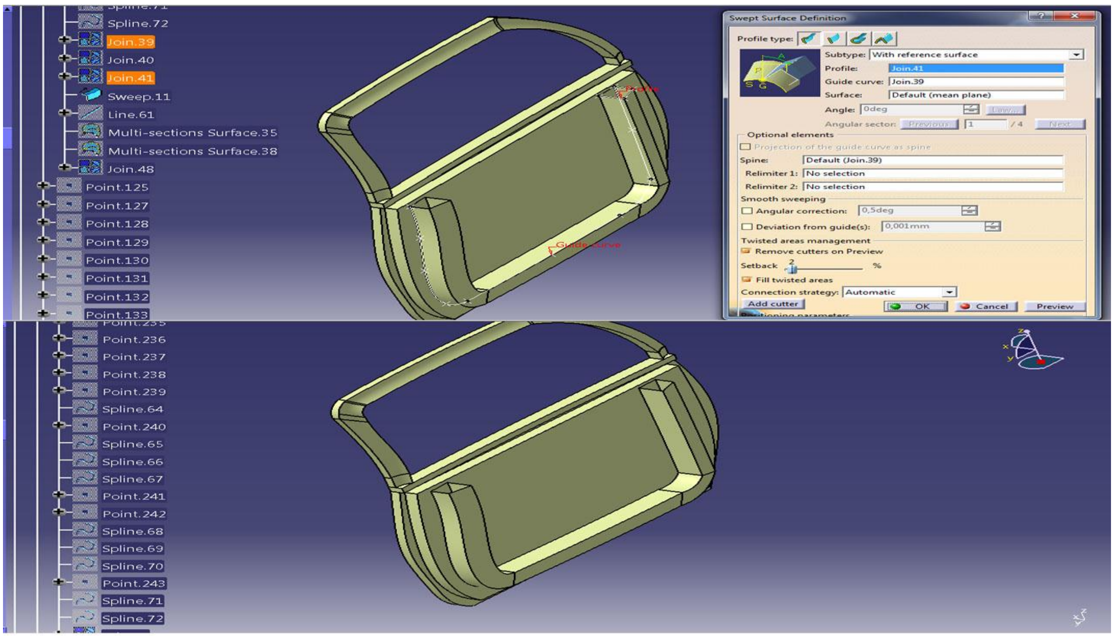
Picture 2.9.15: Additional section of glass window frame.



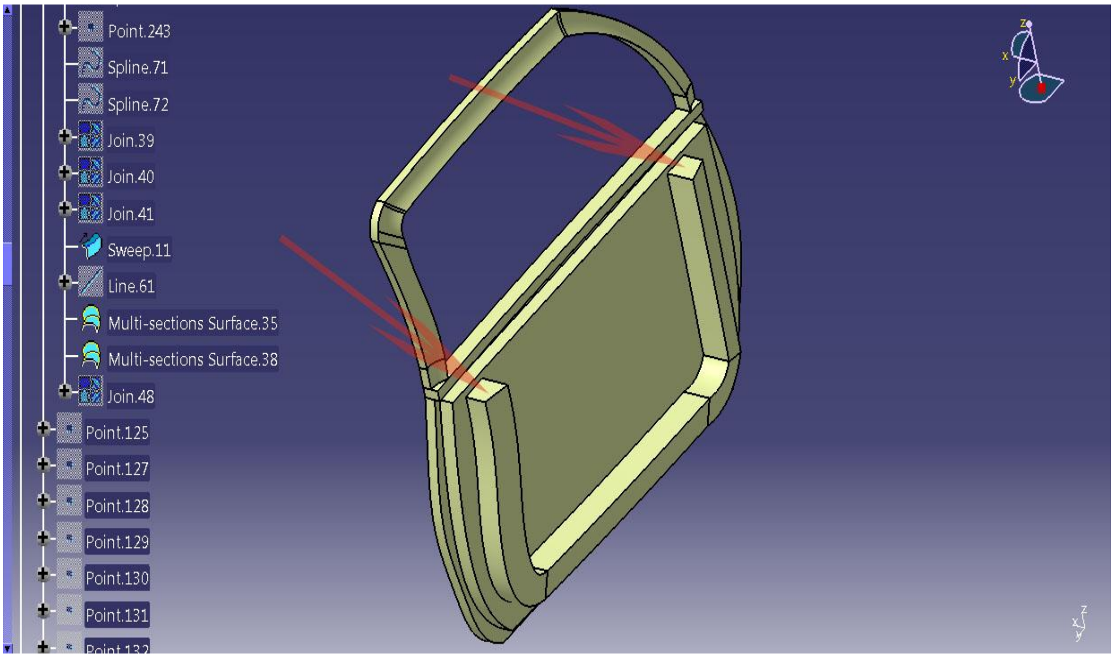
Picture 2.9.16: Creation of surface model of glass window frame.



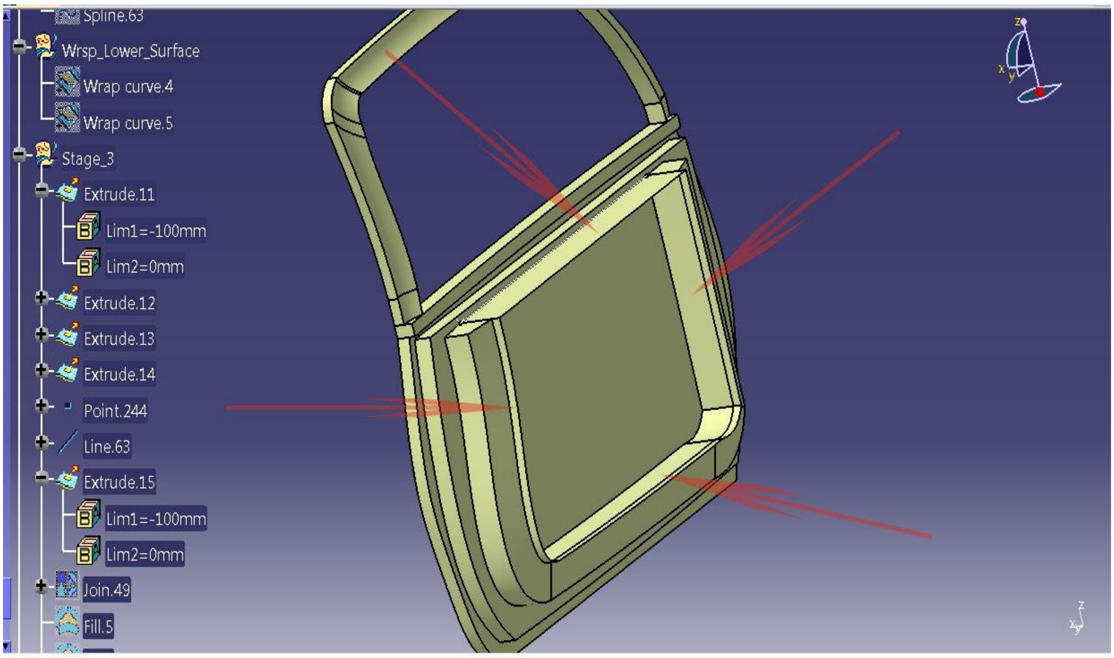
Picture 2.9.17: Wireframe model of door panel second trim.



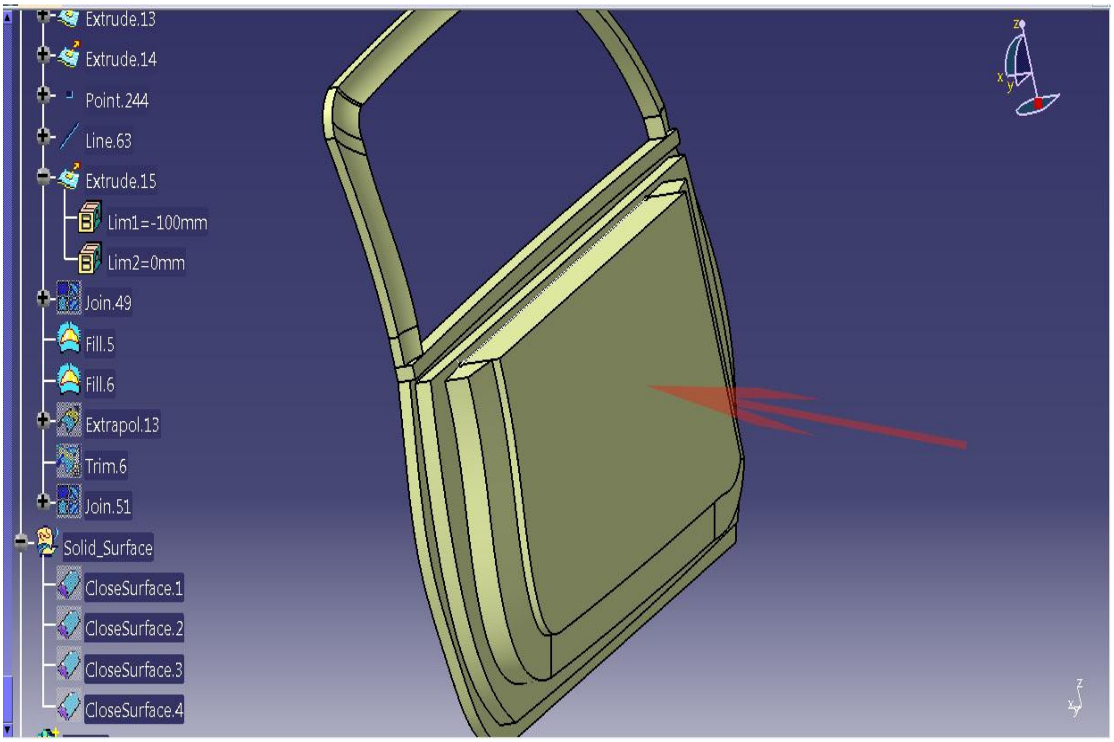
Picture 2.9.18: Creation of second trim sweep surface.



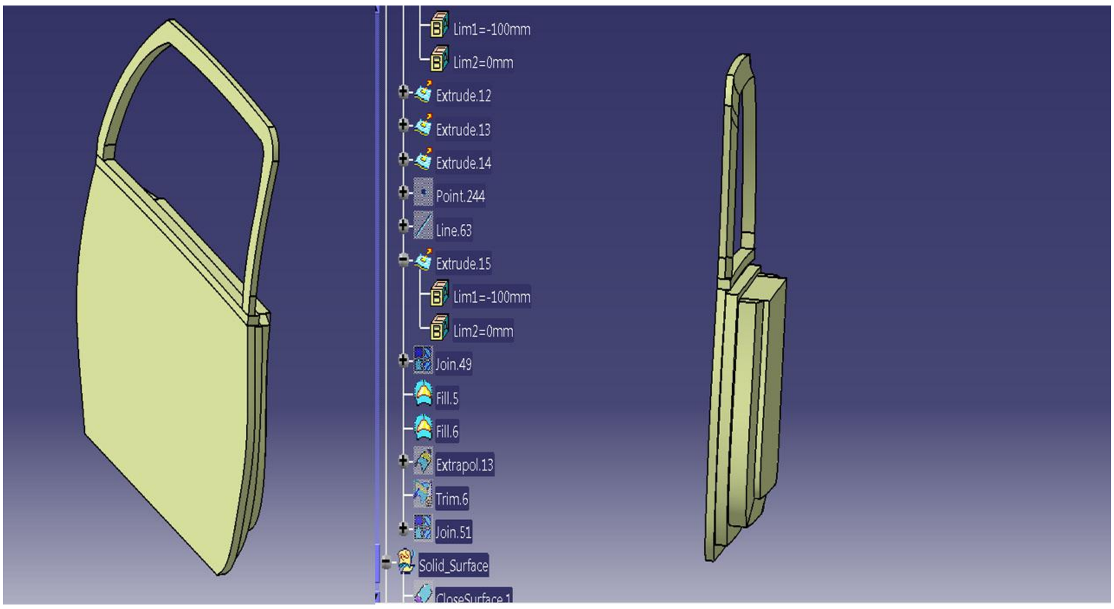
Picture 2.9.19: Cover of second trim openings with multi section surfaces.



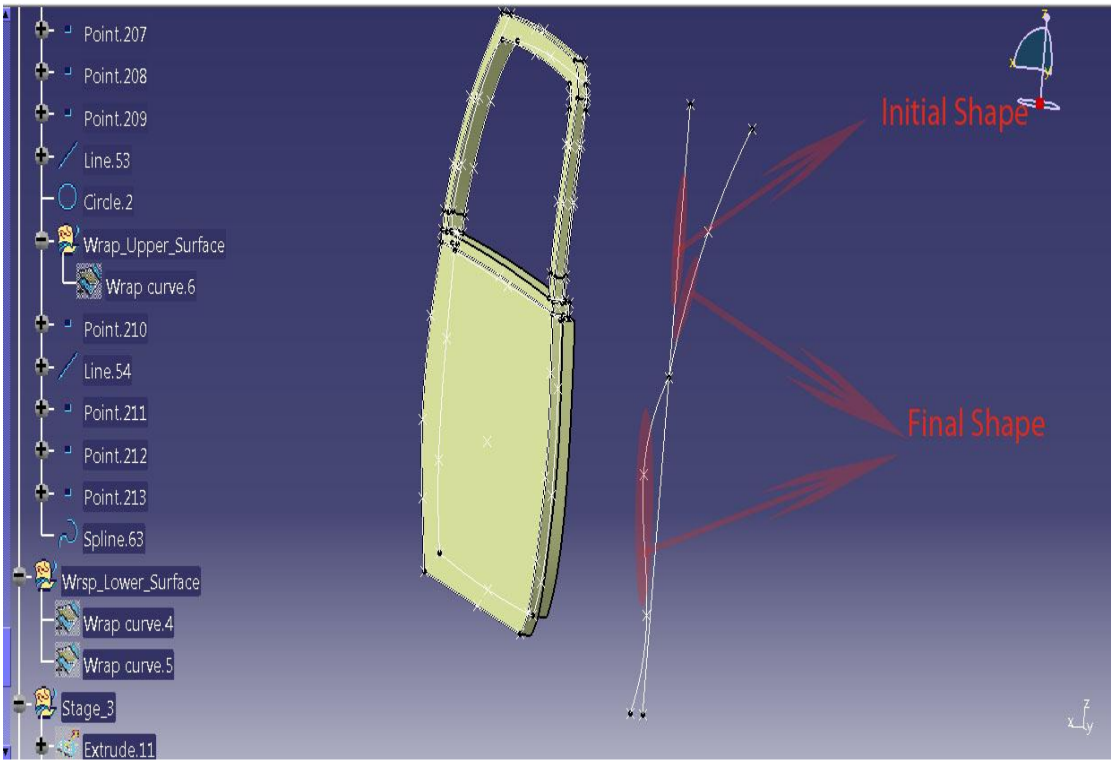
Picture 2.9.20: Door panel third trim set of surfaces of extrusion.



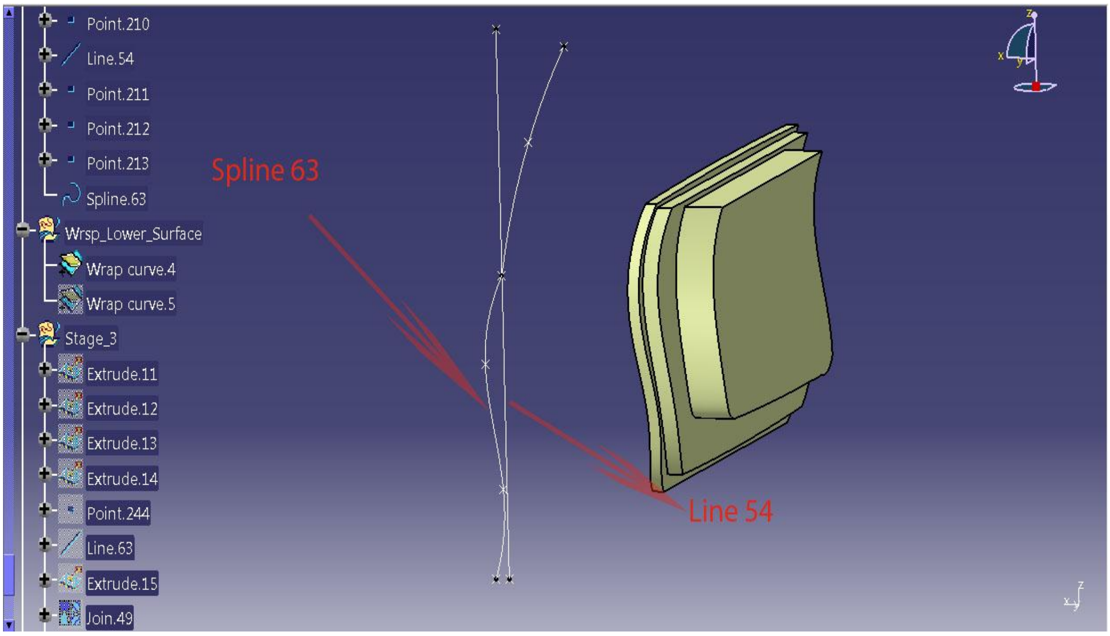
Picture 2.9.21: Third trim final surface.



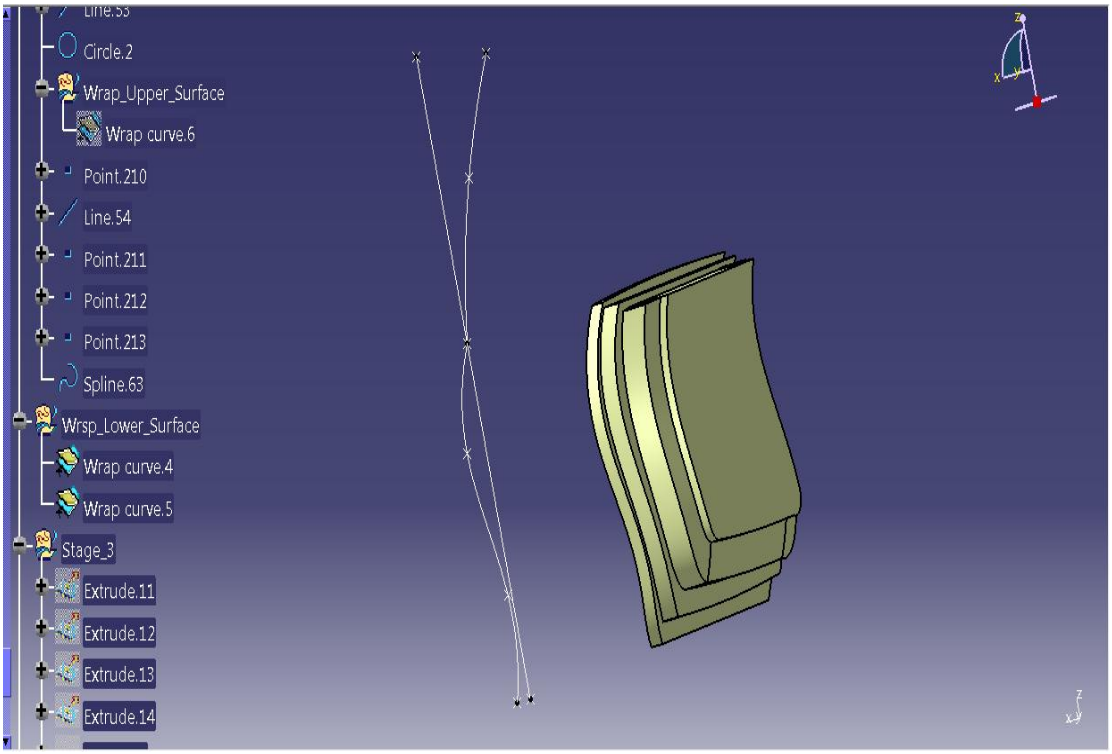
Picture 2.9.22: Complete door panel surface model.



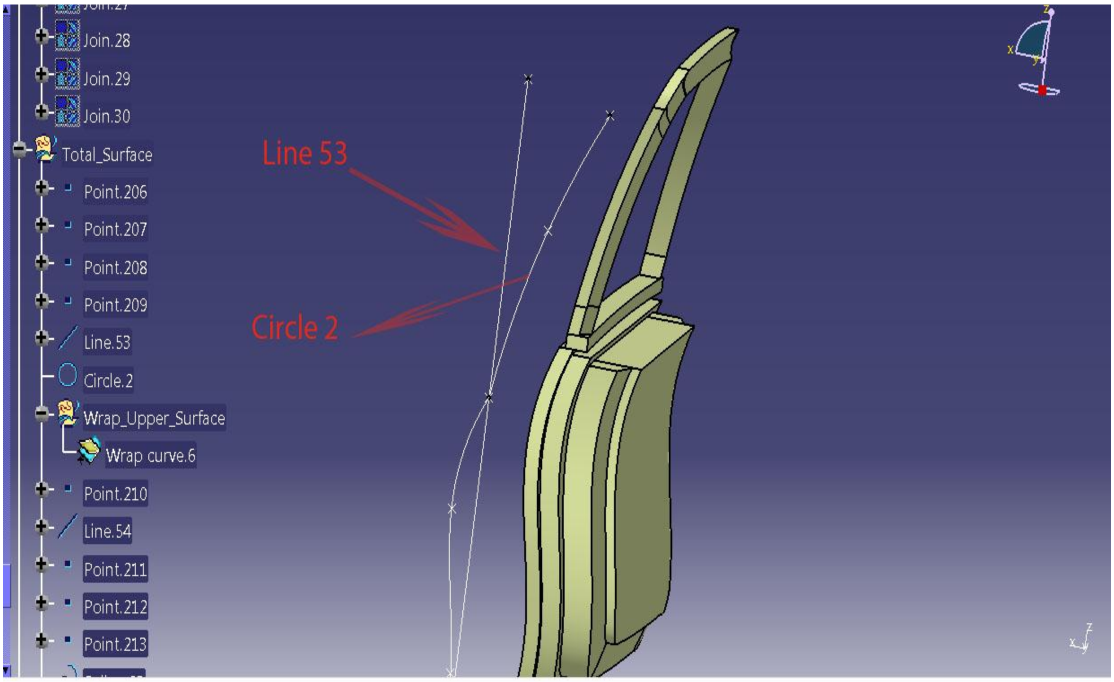
Picture 2.9.23: Wireframe representation of door panel initial and final shape.



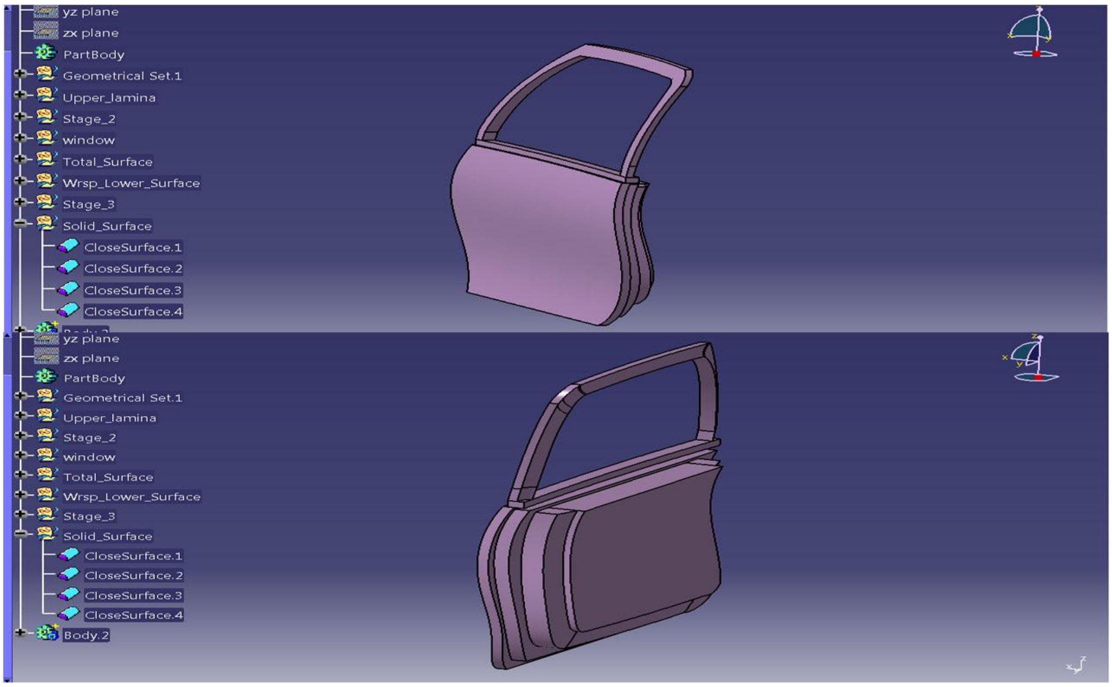
Picture 2.9.24: Execution of “*wrap curve*” command.



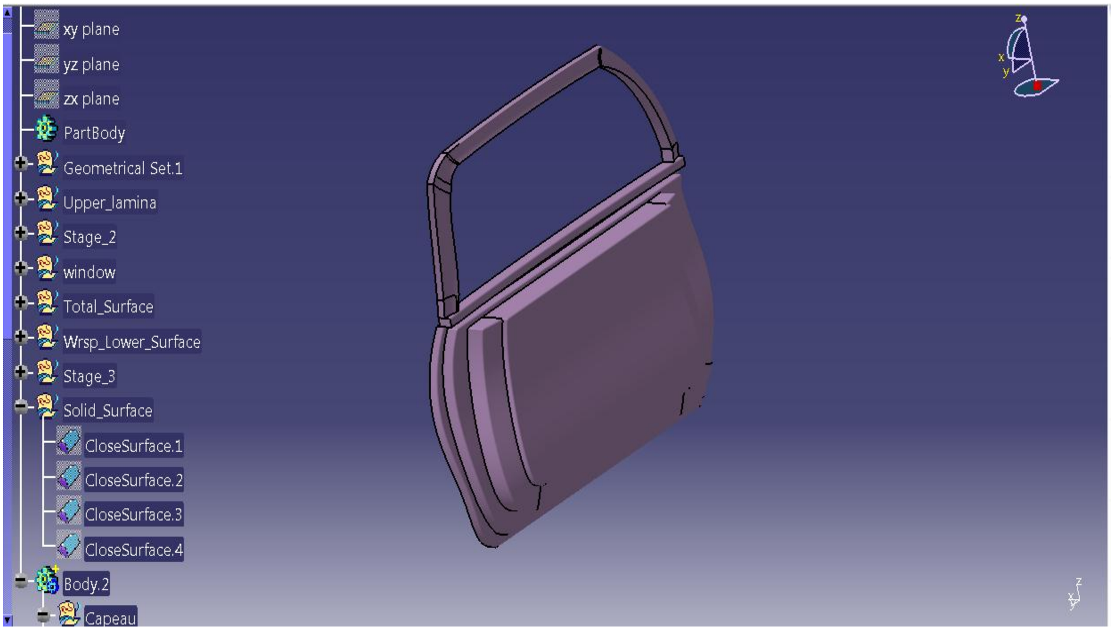
Picture 2.9.25: Execution of second “*wrap curve*” command.



Picture 2.9.26: Execution of “*wrap curve*” command on the set of surfaces of glass window frame.



Picture 2.9.27: Three dimensional door panel initial volume.



Picture 2.9.28: Three dimensional door panel final volume.

2.10 CREATION OF THE TRUNK LID PART

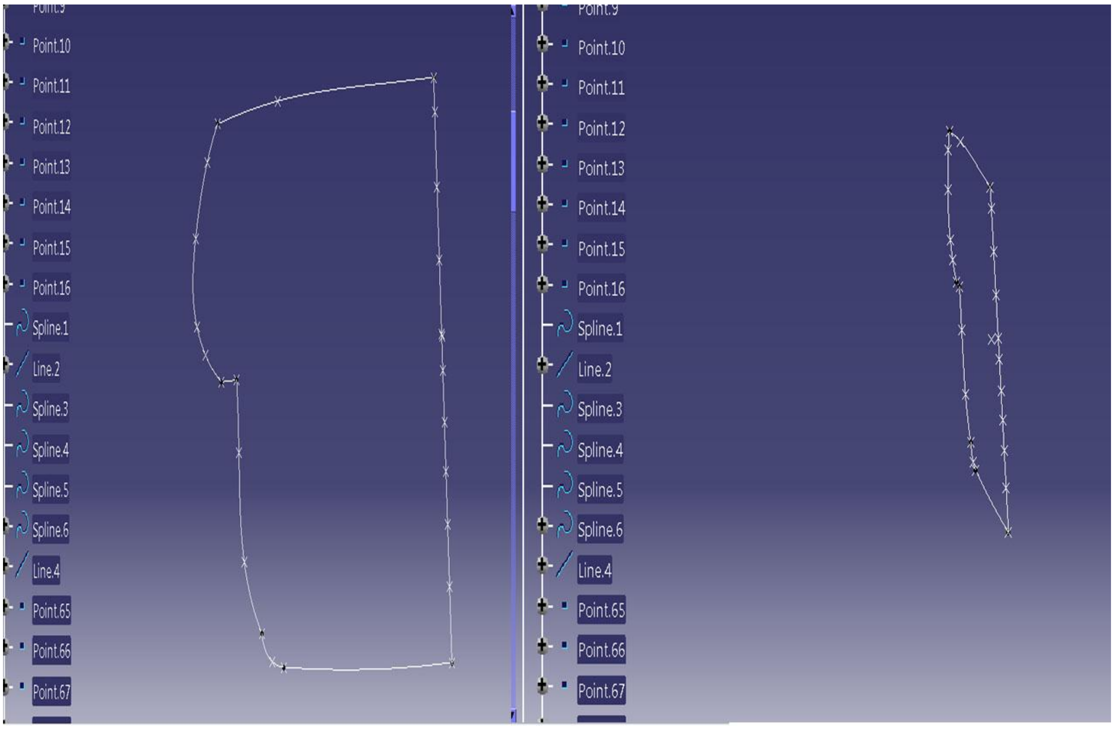
This part constitutes the second primary passage of an urban car vehicle. It is also an integral part for virtually every urban car design. Its primary function is to provide easy access to a void space inside the vehicle where equipment can be stored. The design of this particular part was far simpler than that of the passenger door. Just like the other parts, this was designed using a real urban car trunk lid as a reference. Sources for designs of trunk lids were images found inside manufacturers' official webpages, while some of them derived from private research facilitated by google search engine. As for the dimensions of the CAD model, these derived from measurements taken from an existing car trunk lid of an actual urban car. The part was symmetrical to the xz - plane, thus only half of the desired geometry was defined.

Firstly, the geometry's wireframe model was introduced as the base for the construction of the rest. The model consisted of seven curves, five B-Spline curves and two lines. An additional characteristic of the model is its planarity. The whole model is laid on the yz plane (Picture 2.10.1). The reason why the wireframe model was designed as planar, is the same with the passenger door 3-D design. The design of the wireframe model followed the unification of the seven curves that were included in the model. This unification was done using the "*join*" operation command. Then, a surface was created on the existing wireframe model. This surface was created with the use of "*fill*" command. As input for the execution of the command the unified curve that was previously introduced was inserted (Picture 2.10.2). Next in sequence followed the introduction of a plane. This plane was inserted 80 mm away from the existing planar surface in perpendicular direction towards the yz plane (Picture 2.10.3). On the plane an additional wireframe closed section was defined. The shape of the wireframe's geometry resembled the shape of the window's opening on the door (Picture 2.10.4). A unified curve was created then, using the "*join*" operation command. As boundary inputs the four curves of the wireframe model were inserted. After the unification, an extruded surface was created, using the unified profile as an input (Picture 2.10.5). As the two surfaces intersect each other, a "*split*" operation could be executed. By opting for the split operation it was made possible to remove the interior segment of the filled surface, which was enclosed in the extruded surface (Picture 2.10.6). In order to add some thickness to the 3-D design, an additional extrude operation was opted for. The length of the extrude operation was 25 mm (Picture 2.10.7). Furthermore, an additional feature was integrated inside the design. This feature is a toolkit storeroom, attached to the door's exterior surface. This feature was supplementary to the design and its functional purpose was to provide to the vehicle a place to store several kinds of small tool sets. The wireframe model was defined first. This wireframe formed a closed section that consisted of 6 curves in total (Picture 2.10.8). Just like in previous stages, the wireframe's curves were unified with the use of "*join*" operation. The resulting unified curve was used as generative curve for the construction of a peripheral sweep surface. As a guide curve a single line created after the unification of the section's curves was defined. The guide line was designed setting as its start point one that belonged to the previously created section. Also, the length of the guide line was 35 mm (Picture 2.10.9). As it becomes clear from the picture, a gap was formed as a result of the creation of the sweep surface. This gap was closed with a filled surface (Picture 2.10.10).

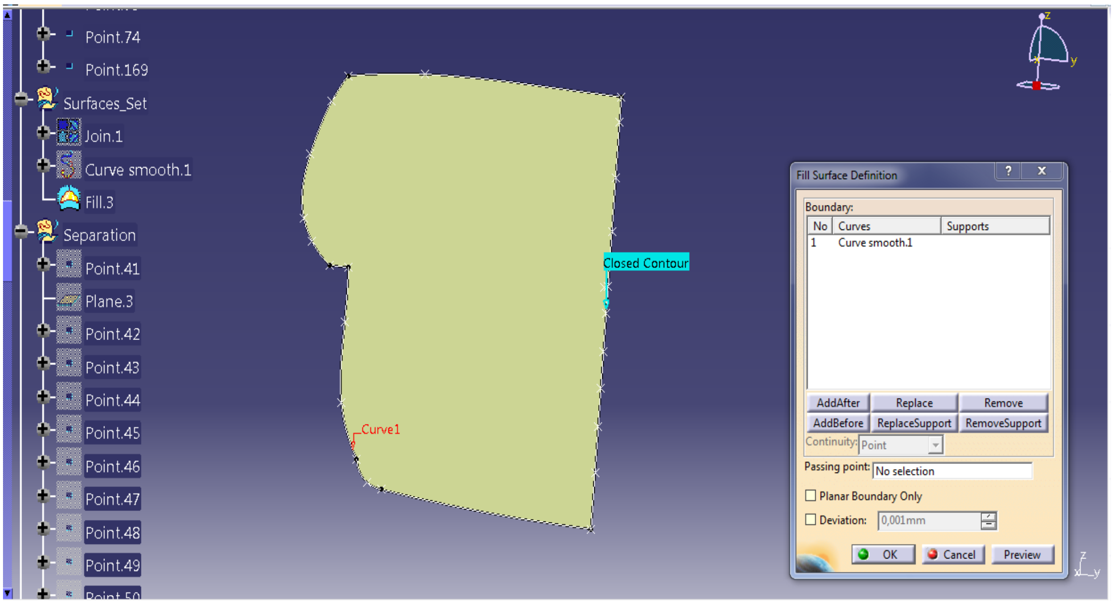
In order to approach the design of a real trunk lid more accurately, the current geometry had to be deformed. For the deformation of the 3-D design, as in the previous part, the "*wrap curve*" operation command was used. At first the wireframe models of the initial and final shape of the geometry were defined. The reason why the design of the real trunk lid is complex is because there was a need for a second curve wrap command to be executed in sequence. This second operation aimed at the design's horizontal deformation (Picture 2.10.12). The last stage that remained for the design of the solid part to be completed was the creation of the solid second symmetrical half. This geometry resulted from the execution of the "*symmetry*" operation command. As plane of symmetry for this operation the xz plane was selected. Two symmetry operation commands were executed in total. The first one was for the construction of the door's solid volume second half and the second was for the supplementary toolkit's solid volume second half (Picture 2.10.14). The primary dimension components that were measured on a real urban car vehicle were the total length and height, as well as the height of the window frame. The exact numerical values of the dimension components are depicted as measured on the table below.

Table 2.10.1

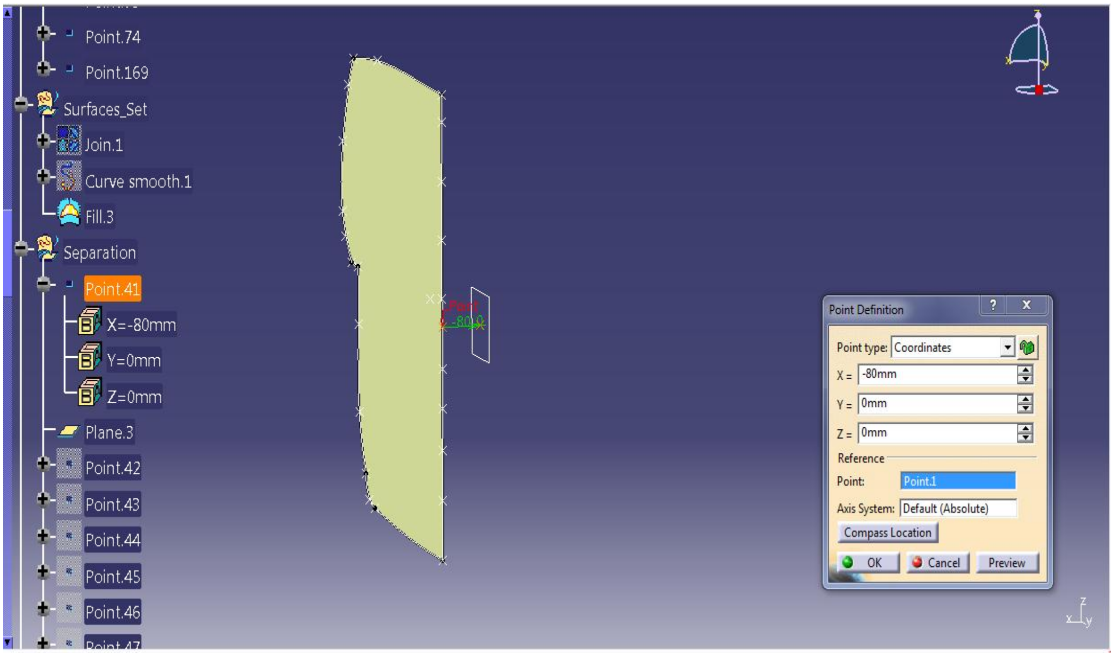
Dimensions	Measured Values (mm)
Total Length	1100
Total Height	830
Window's Frame Height	420



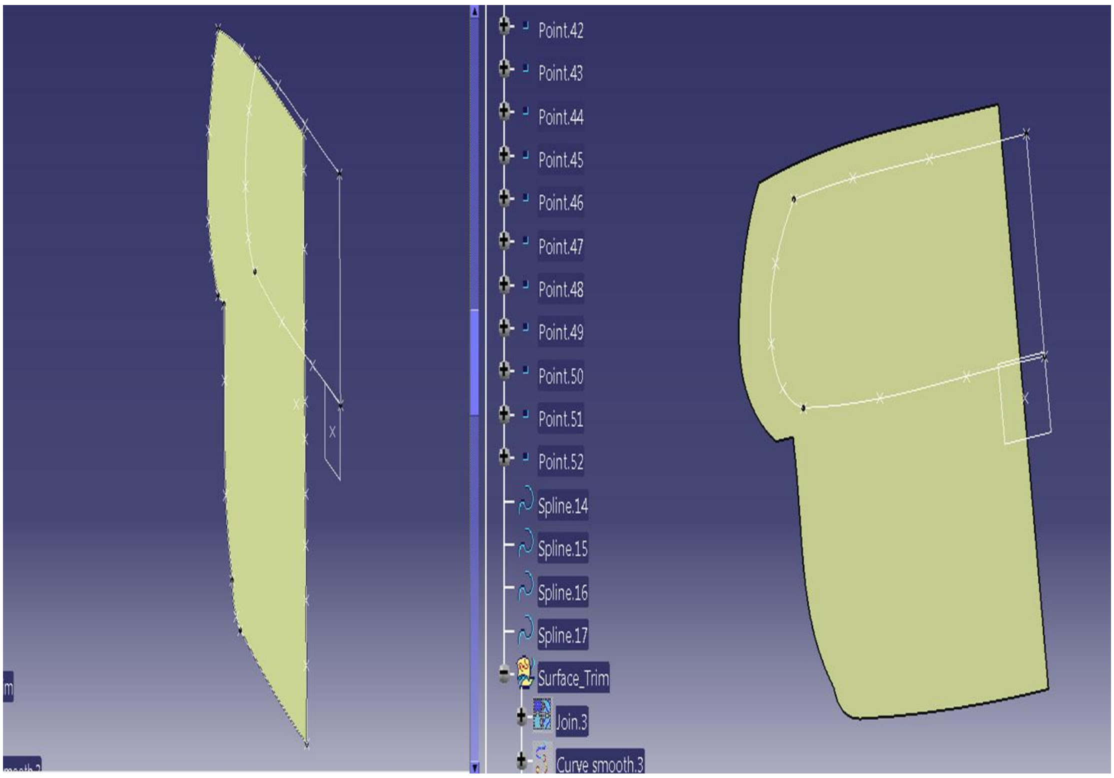
Picture 2.10.1: Trunk lid's wireframe model.



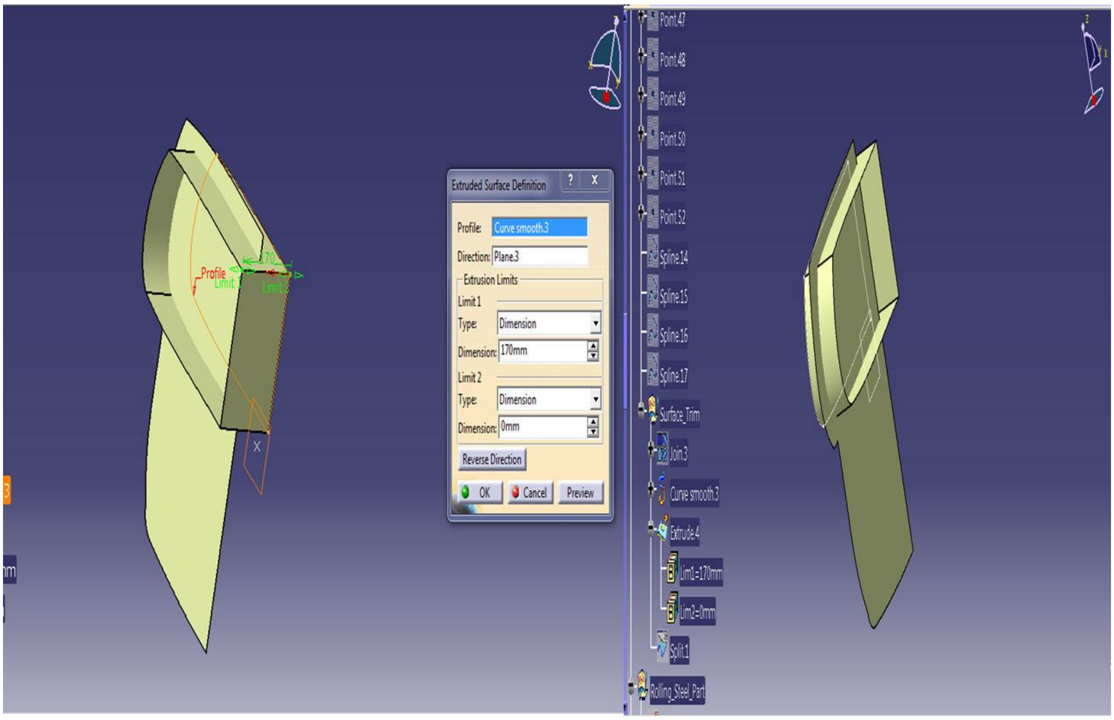
Picture 2.10.2: Creation of trunk lid's wireframe fill surface.



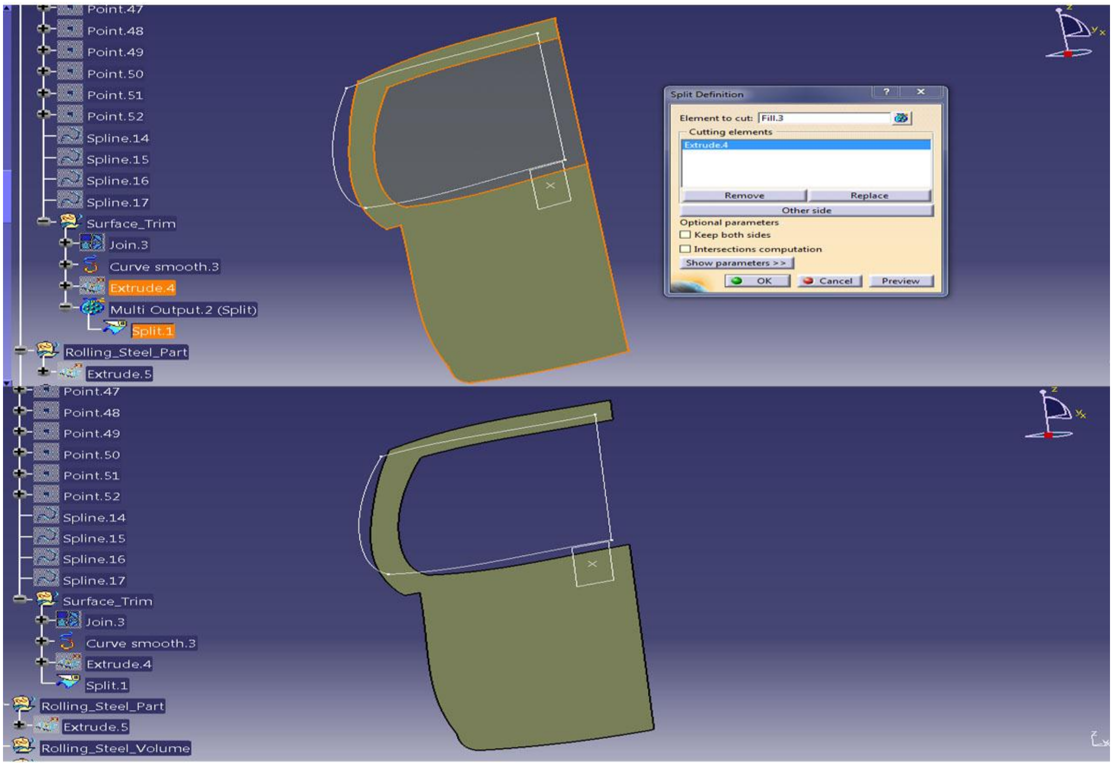
Picture 2.10.3: Introduction of plane.



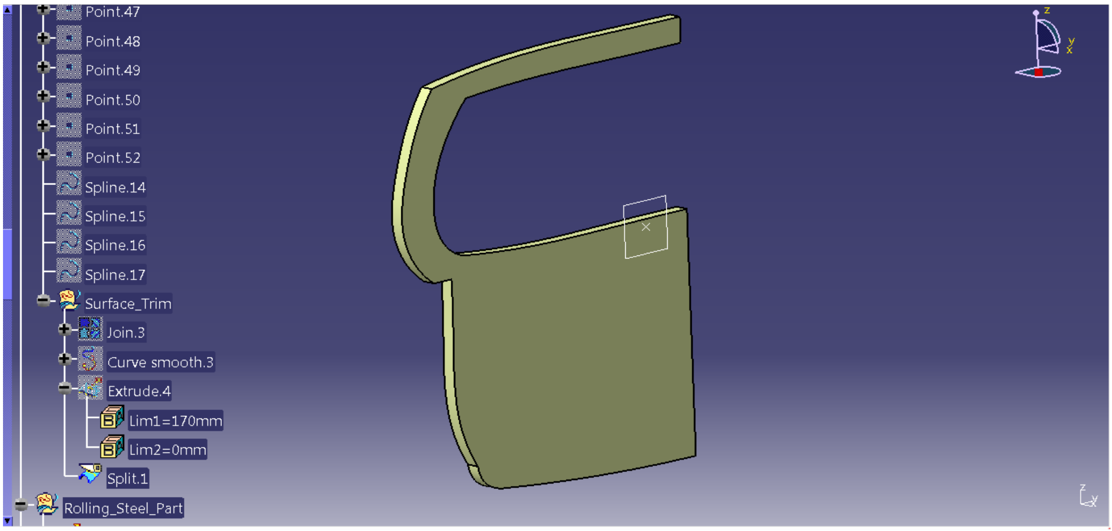
Picture 2.10.4: Trunk lid's additional wireframe for the creation of a window opening.



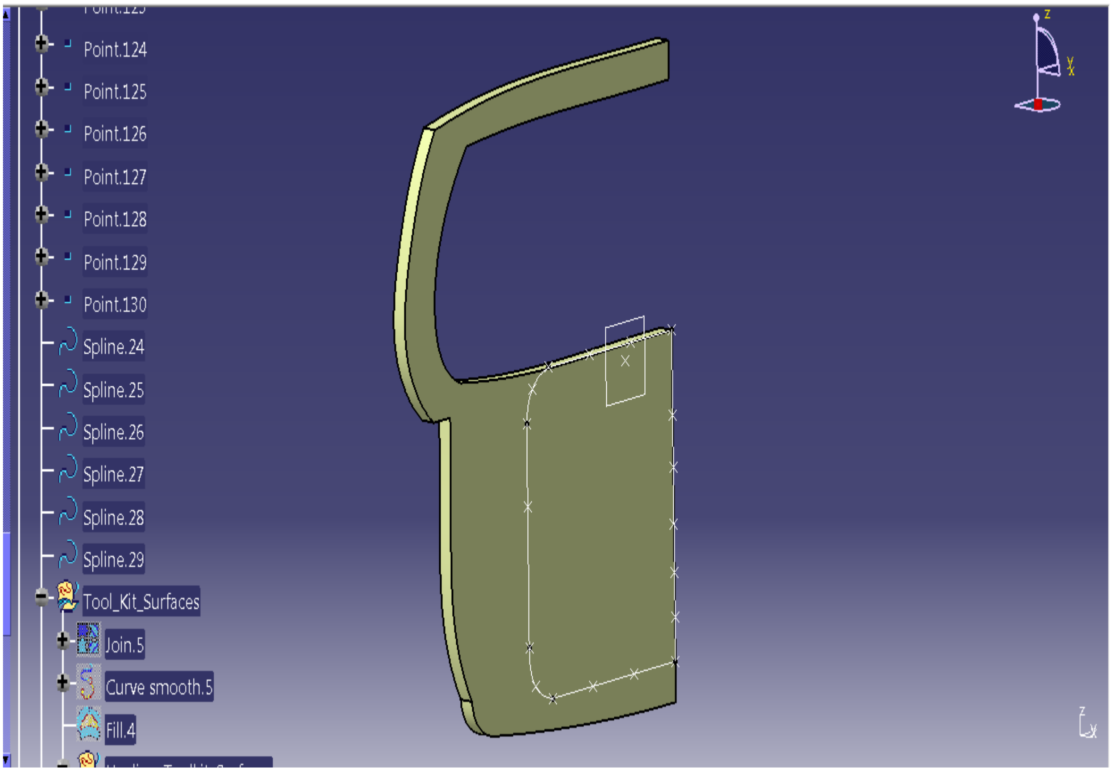
Picture 2.10.5: Creation of a surface of extrusion from the previous wireframe model.



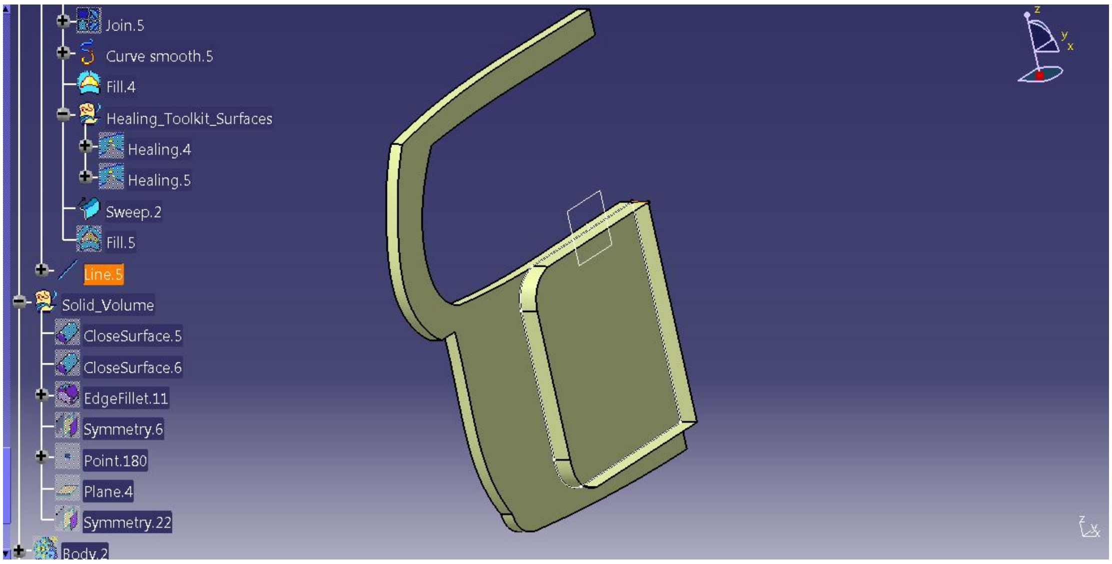
Picture 2.10.6: Execution of trim operation.



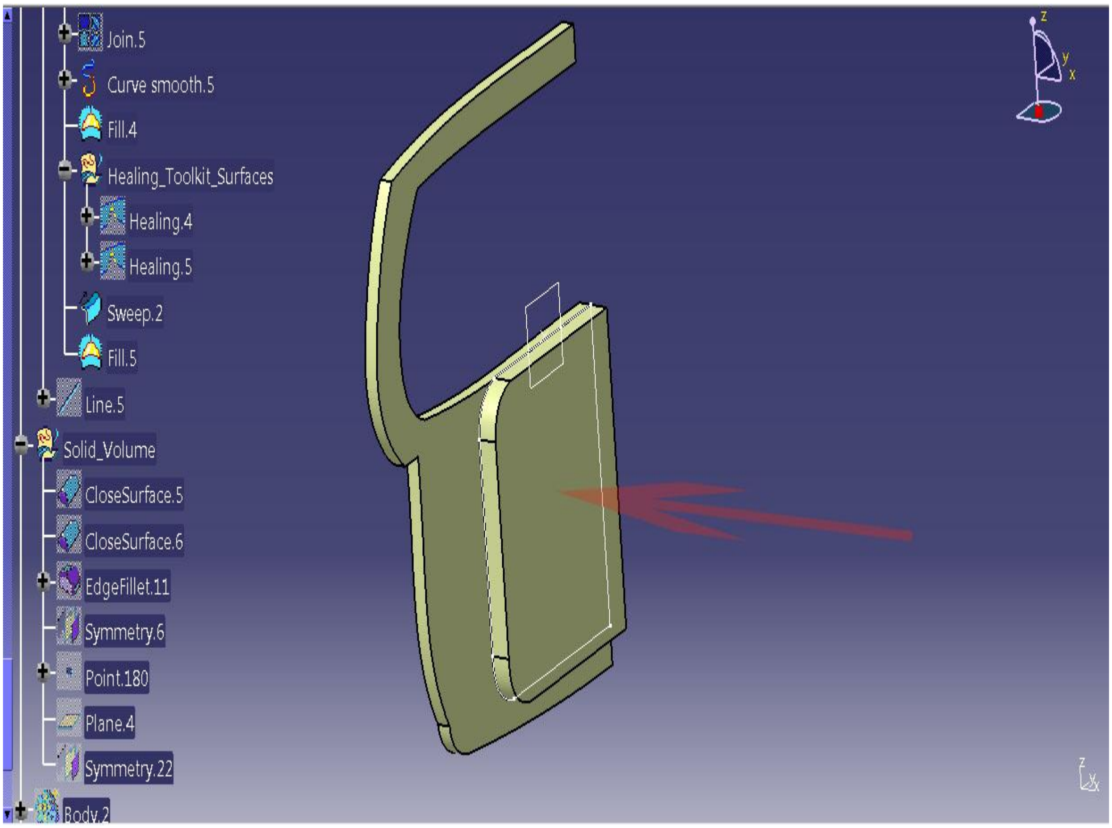
Picture 2.10.7: Execution of “*extrude*” operation on the trimmed surface.



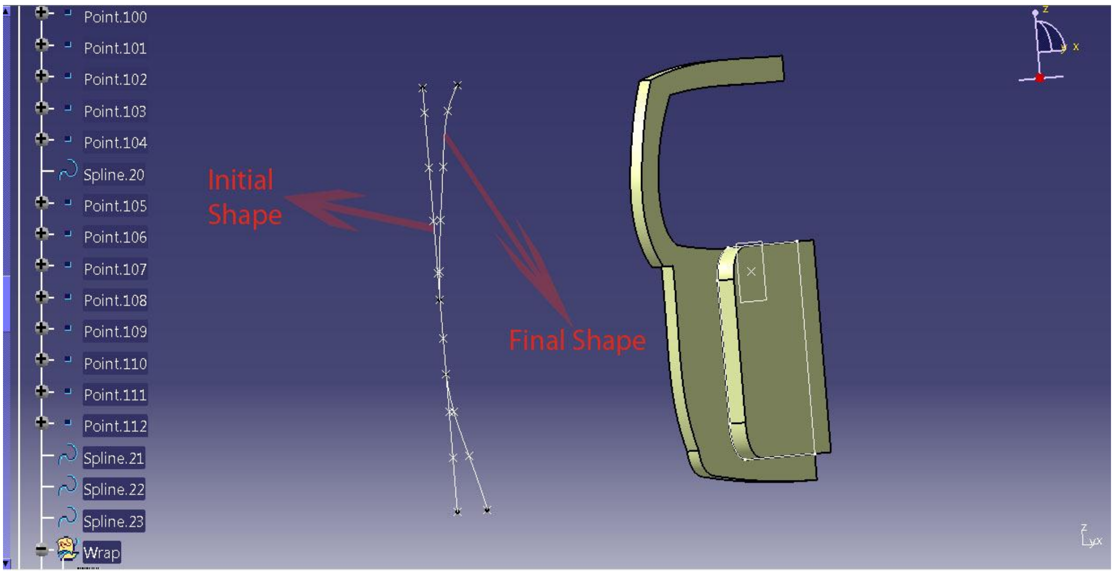
Picture 2.10.8: Additional toolkit's trim wireframe section.



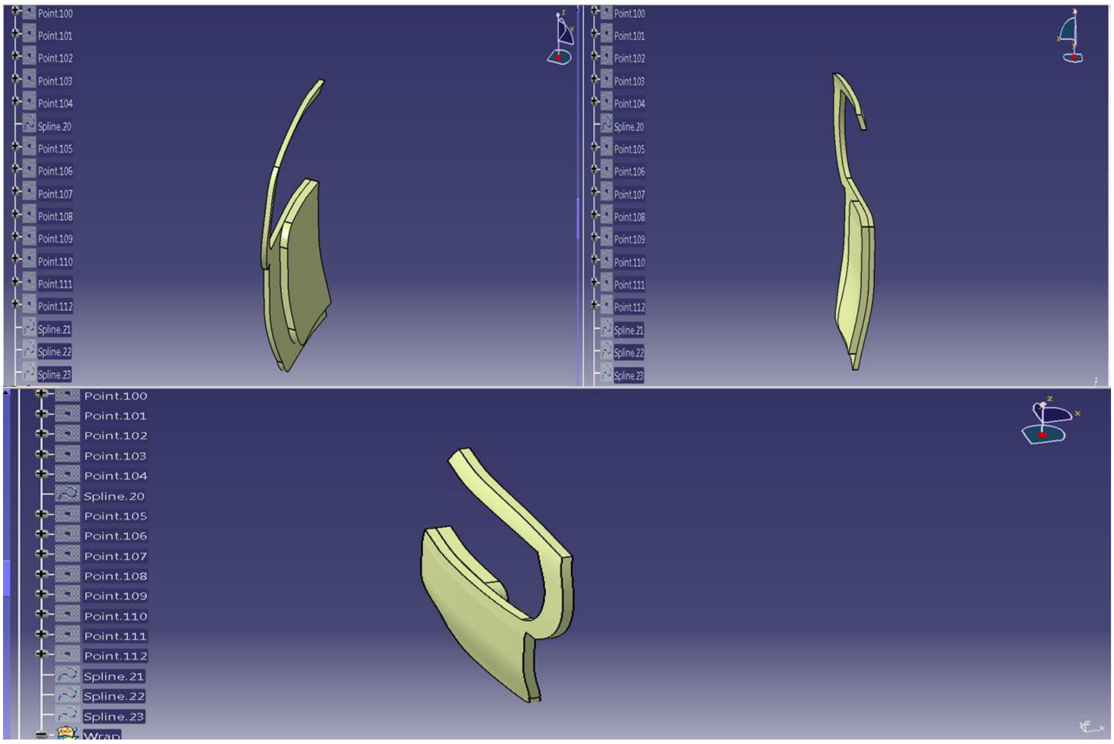
Picture 2.10.9: Creation of toolkit's trim peripheral surface as sweep surface.



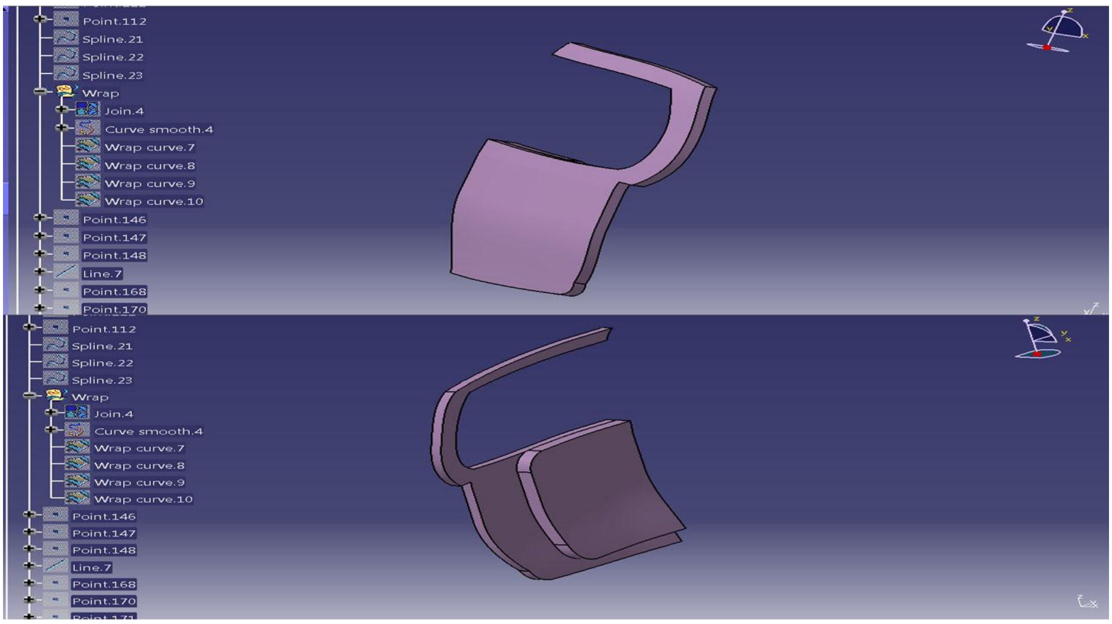
Picture 2.10.10: Creation of trim fill surface.



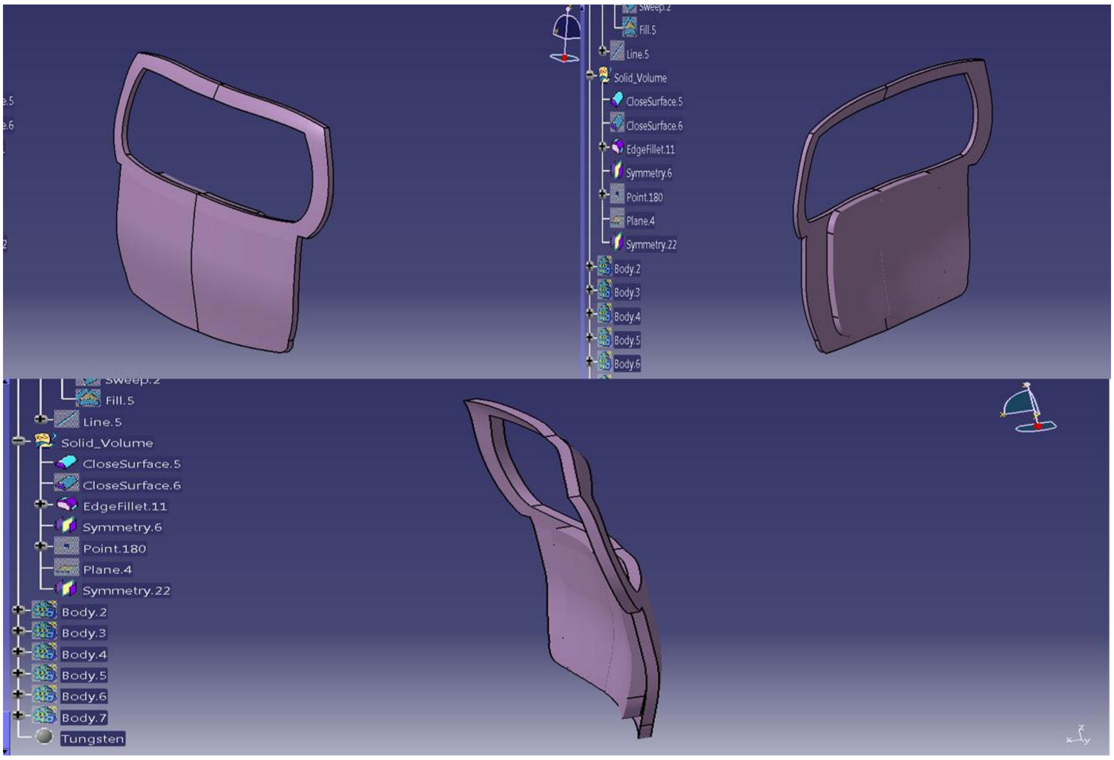
Picture 2.10.11: Wireframe representation of initial and final shape of trunk lid’s part.



Picture 2.10.12: Trunk lid’s part final shape.



Picture 2.10.13: Trunk lid’s final solid volume of first symmetrical half.



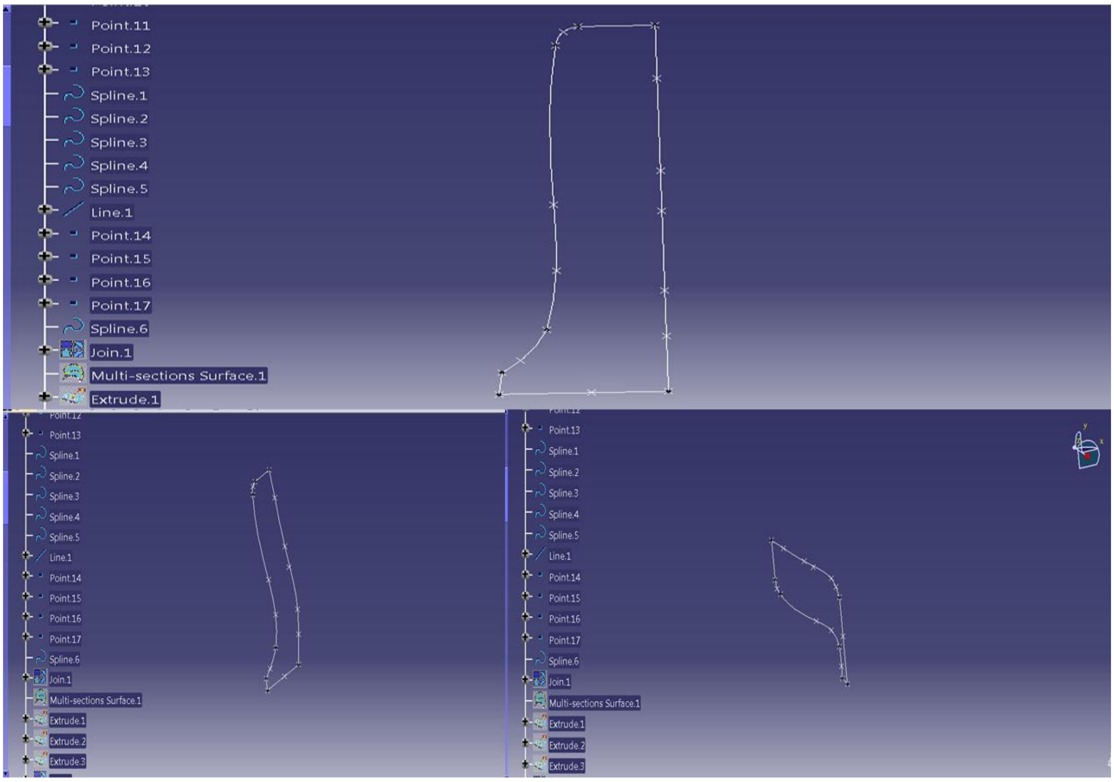
Picture 2.10.14: Trunk lid's complete solid part.

2.11 DEFINITION OF THE PEDAL PARTS

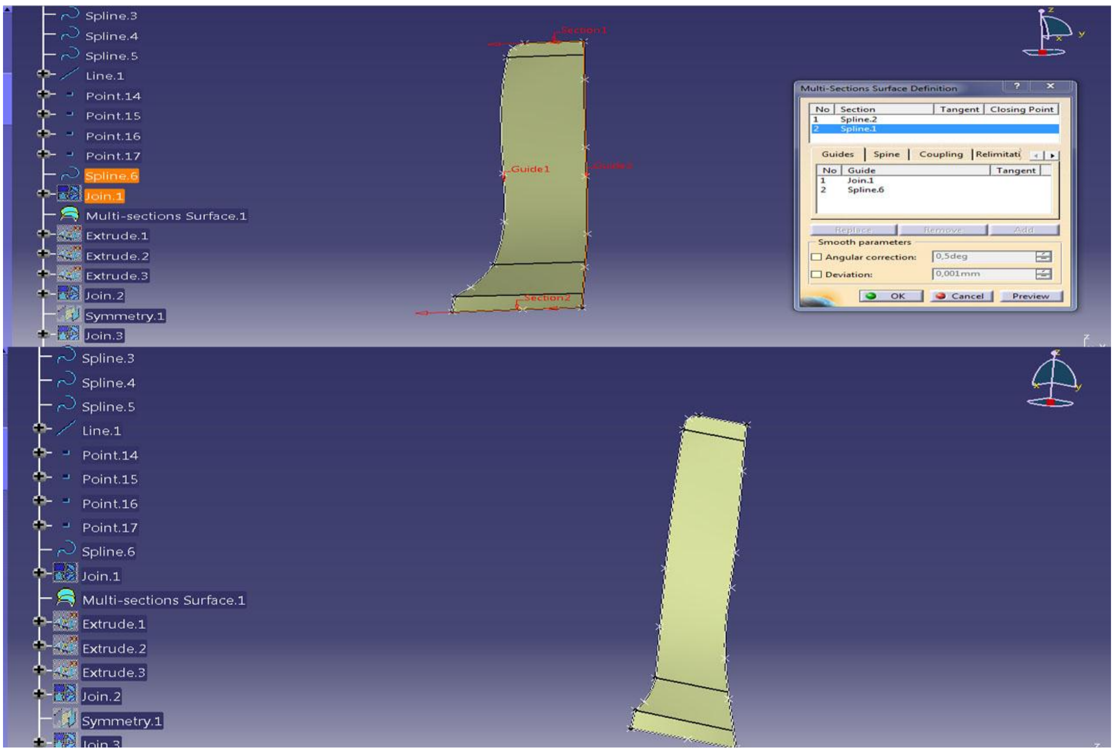
Just like with the parts of handbrake lever and gear clutch lever, the purpose of these parts was supplementary. Their design scope was to design a cabin cockpit that resembled an actual one as much as possible. The pedal parts that were constructed and placed on the vehicle's dashboard were the throttle pedal, the brake pedal and the clutch pedal. The complete pedal parts derived as an assembly of several independent parts. For the design of these certain 3-D models, the precise dimension numerical values were found with a different technique. This technique involved the insertion of the pedal parts inside the cabin's final assembly and then by a trial and error procedure it was verified whether the chosen dimensions of the parametric design were reasonable according to the cabin's final assembly.

2.11.1 DESIGN OF THE THROTTLE PEDAL PART

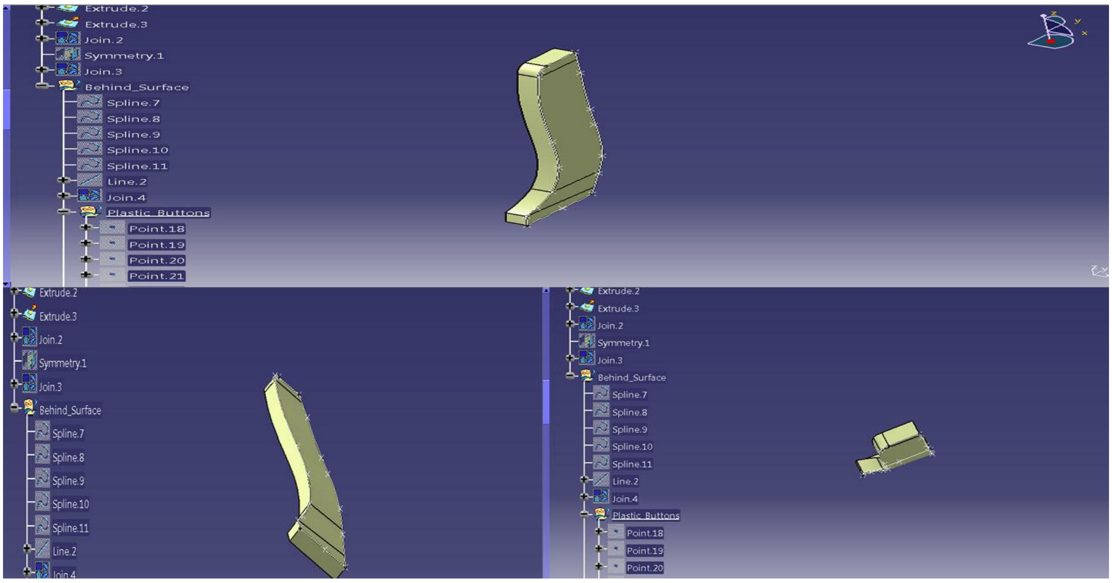
For the construction of the 3-d model design, its wireframe model was created first. As it can be seen in the picture of the wireframe design, the complete final geometry has a symmetry with respect to xz plane. As a result, only half of the geometry had to be determined. An additional important feature of the initial wireframe model is that the closed section is not planar. The wireframe closed section consisted of six B-Spline curves and one line (Picture 2.11.1.1). Next to the design process was a unification of three of the current section curves to a single entity with the use of *"join"* operation command. Then, the closed section was ''filled'' with a multi-section surface. For the creation of the multi section surface, the upper and lower B-Spline of the wireframe section were imported inside the command window as generative curves. Additionally, the two remaining curvilinear curves were used as guide curves (Picture 2.11.1.2). Moreover, three additional surfaces were created, using the current wireframe model as a base. These surfaces were defined as extruded surfaces. As profiles, inside the command window for each surface the two curves that were used as generative ones for the previously created multi section surface were inserted, while the unified guide curve was placed on the left of the section's profile (Picture 2.11.1.3). As a direction for the implementation of the *"extrude surface"* command, the direction of the unit vertical vector to the yz plane was chosen. Then, the four surfaces were unified using the "*Join*" operation command. After the unification, the second half of the symmetrical geometry was constructed with the execution of the "*symmetry*" command. As symmetrical element the previously defined unified symmetrical geometry was inserted, while as symmetry plane the xz plane was imported (Picture 2.11.1.4). Then the two separate symmetrical halves were brought together with the execution of an additional *"join"* operation command. On the current geometry, there was a gap on the back side of it, as shown in Picture 2.11.1.5. A multi-section surface was defined on the geometry's back side, using the same methodology as the one used for the design of the initial multi-section surface of the 3-D model (Picture 2.11.1.6). In order for the second symmetrical half to be filled with a surface, an additional symmetrical surface was constructed (Picture 2.11.1.7). Then, the set of the already existing surfaces was united, in order to form a single unified entity. For purposes of completeness, the geometry of the metallic pad knobs, that several throttle pedal pads feature in their design, was introduced. Firstly, an elliptical wireframe geometry was defined. Using the elliptical profile as a reference, an extruded surface was formed. This surface constituted the pad knob's peripheral surface (Picture 2.11.1.8). The gap which occurred from the pad knob's front face was closed with a fill surface. Naturally, the previously formed extruded surface contour was inserted in the command's boundary frame (Picture 2.11.1.9). Then, the remaining pad knobs of the geometry were formed using the "*user pattern*" command. The places where the pad knobs were going to be placed were defined by sketches. For the final geometry, two user patterns were created and consequently two more sketches were defined (Picture 2.11.1.10). Finally for reasons of simplicity, due to the fact that the pedals were not included in the set of volumes that were going to be inserted in the aerodynamical analysis of the vehicle's cabin, the solid volume of the pedals was not defined. Consequently, the part's design remained just a surface model.



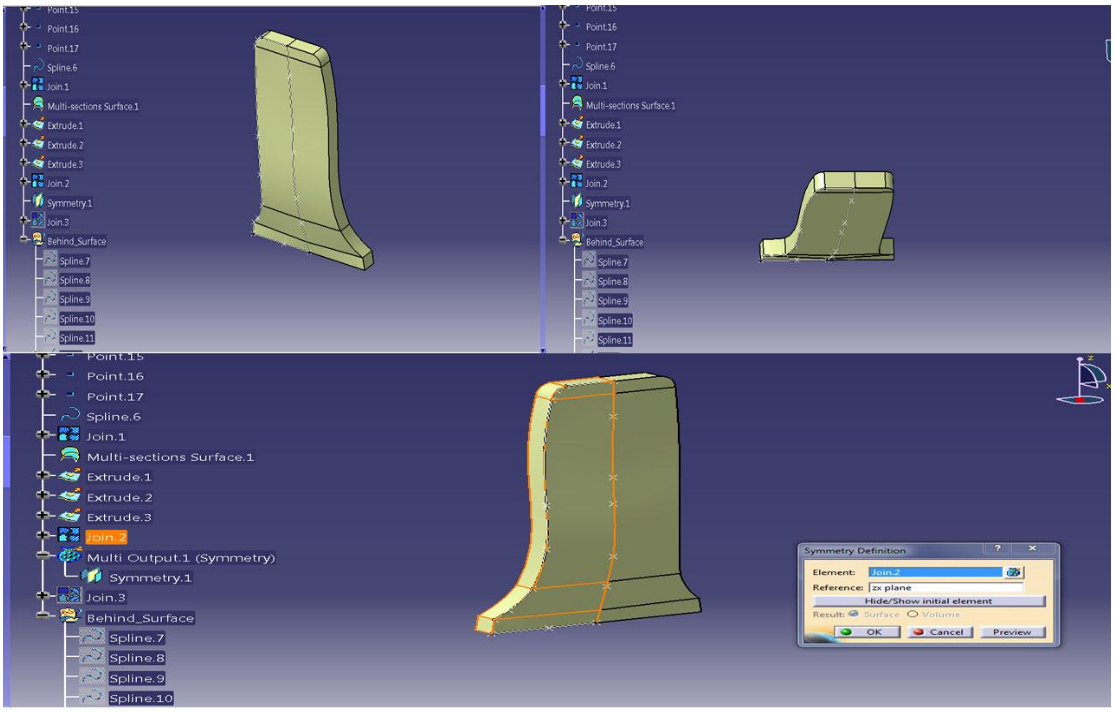
Picture 2.11.1.1: Throttle's pedal pad wireframe section.



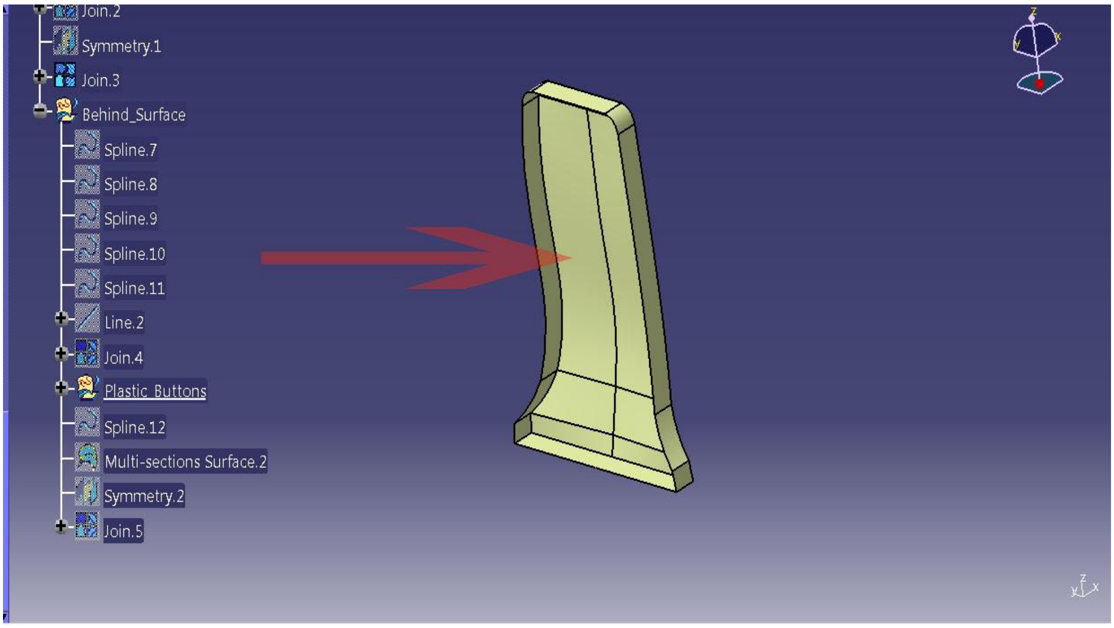
Picture 2.11.1.2: Creation of throttle's pedal pad multi section surface.



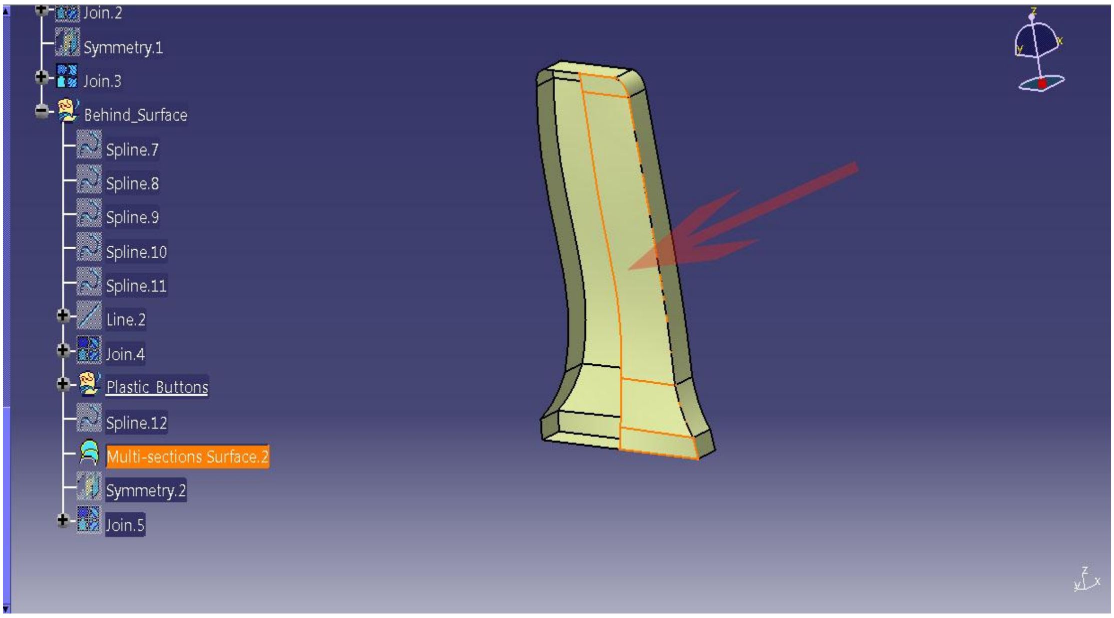
Picture 2.11.1.3: Creation of throttle's pedal pad peripheral surfaces.



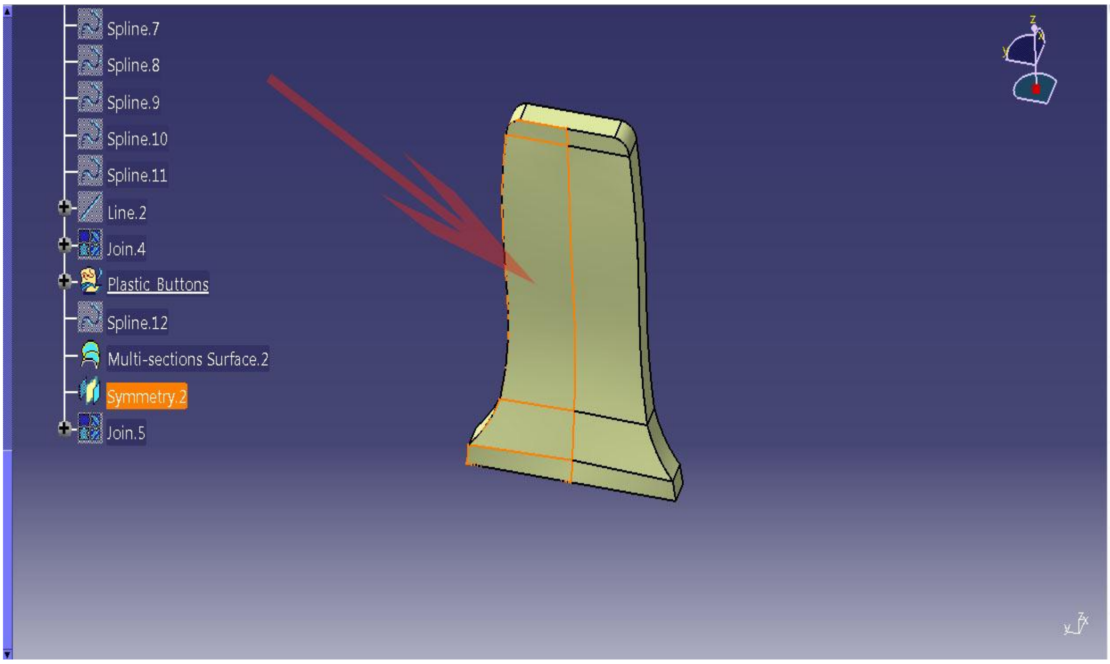
Picture 2.11.1.4: Creation of throttle's pedal pad second symmetrical half.



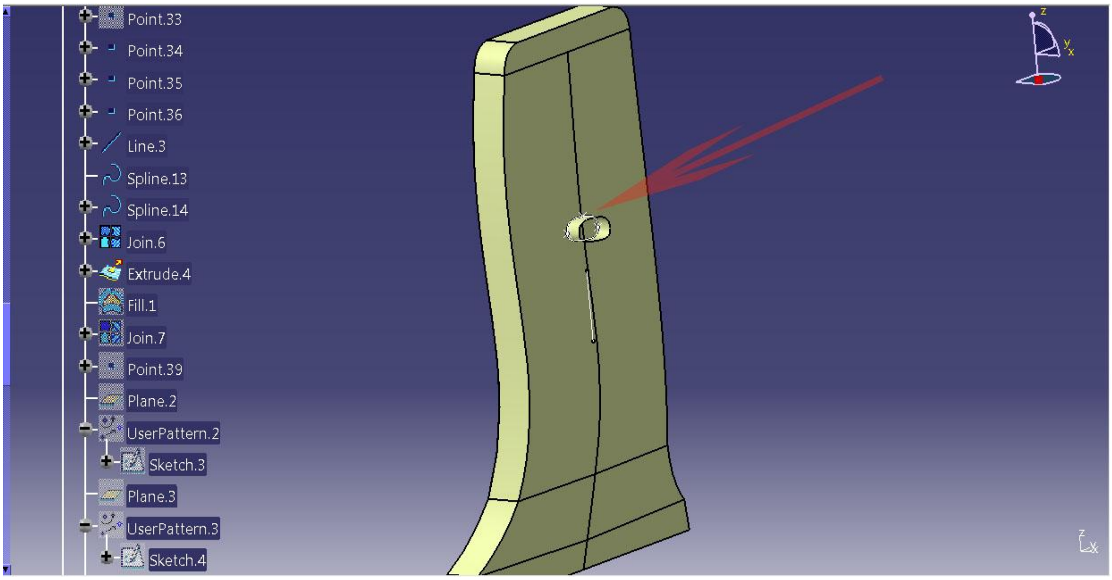
Picture 2.11.1.5: Gap on throttle's pedal back side.



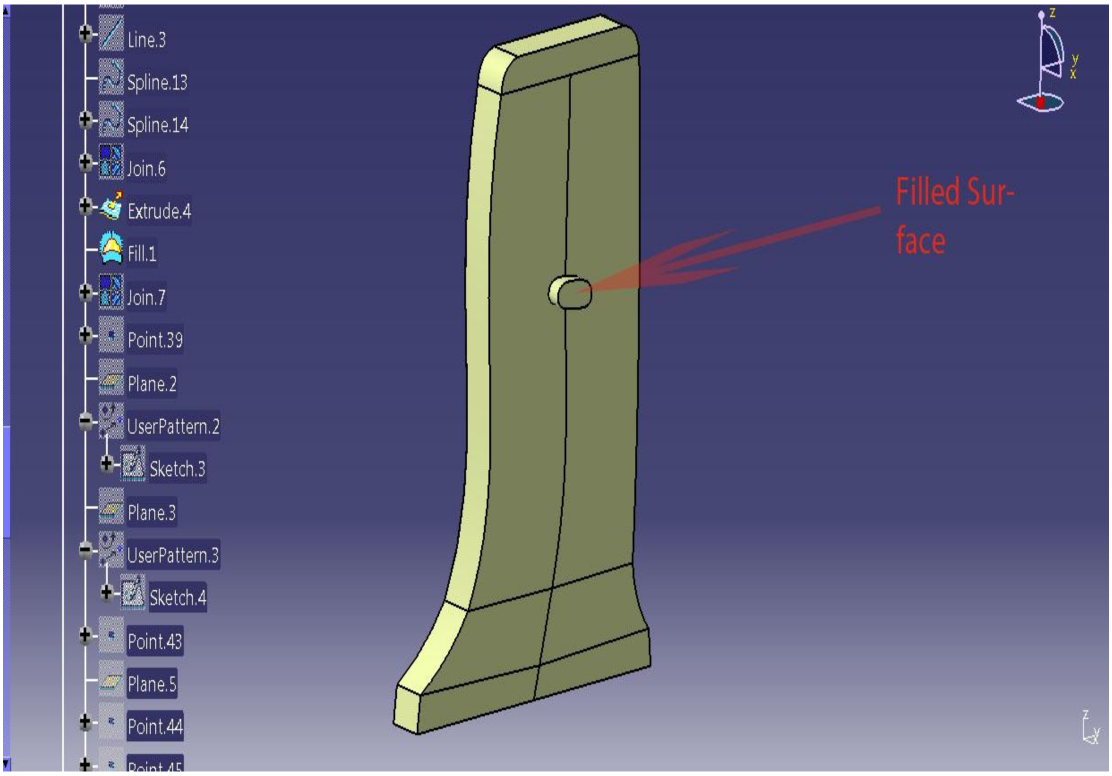
Picture 2.11.1.6: Covering of throttle's pedal pad back side with first multi section surface.



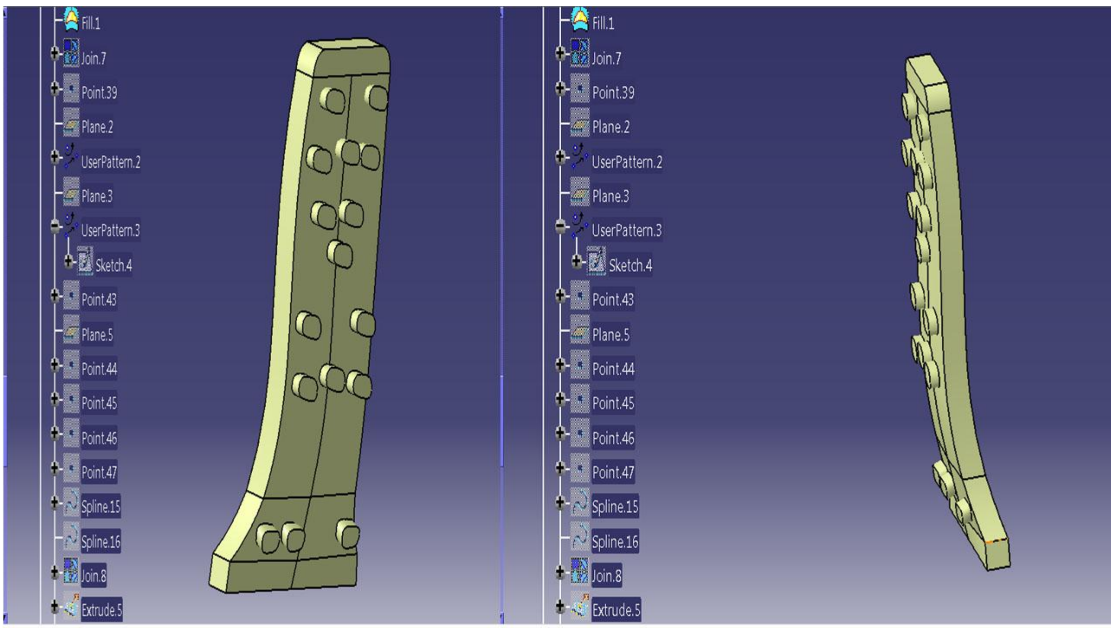
Picture 2.11.1.7: Creation of throttle's pedal pad second symmetrical surface.



Picture 2.11.1.8: Creation of pedal pad knob.



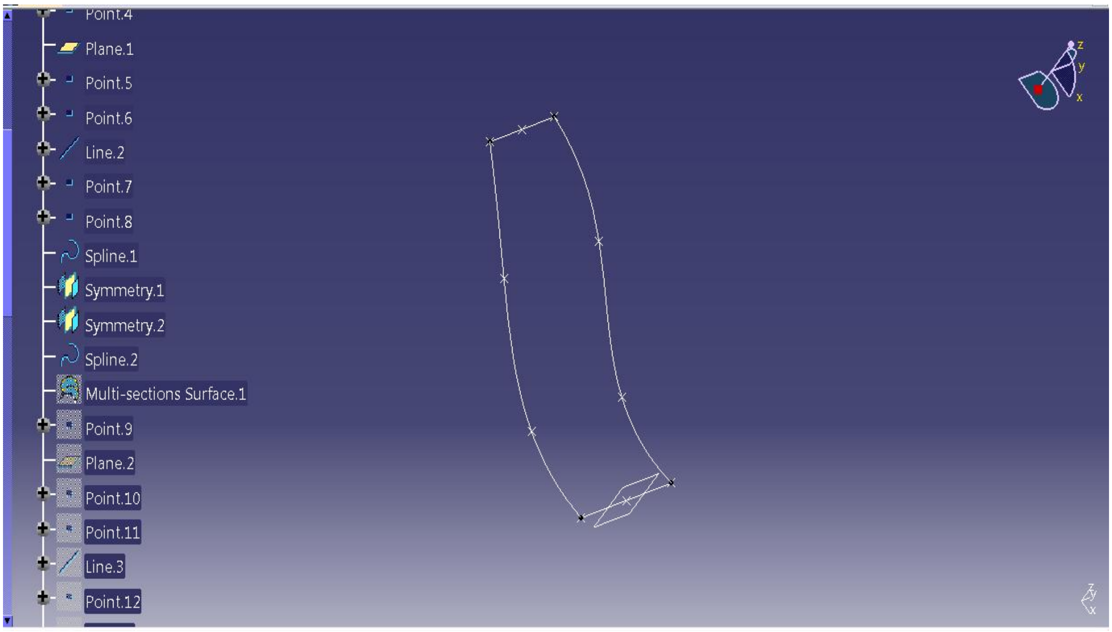
Picture 2.11.1.9: Creation of pedal pad metallic cylinder.



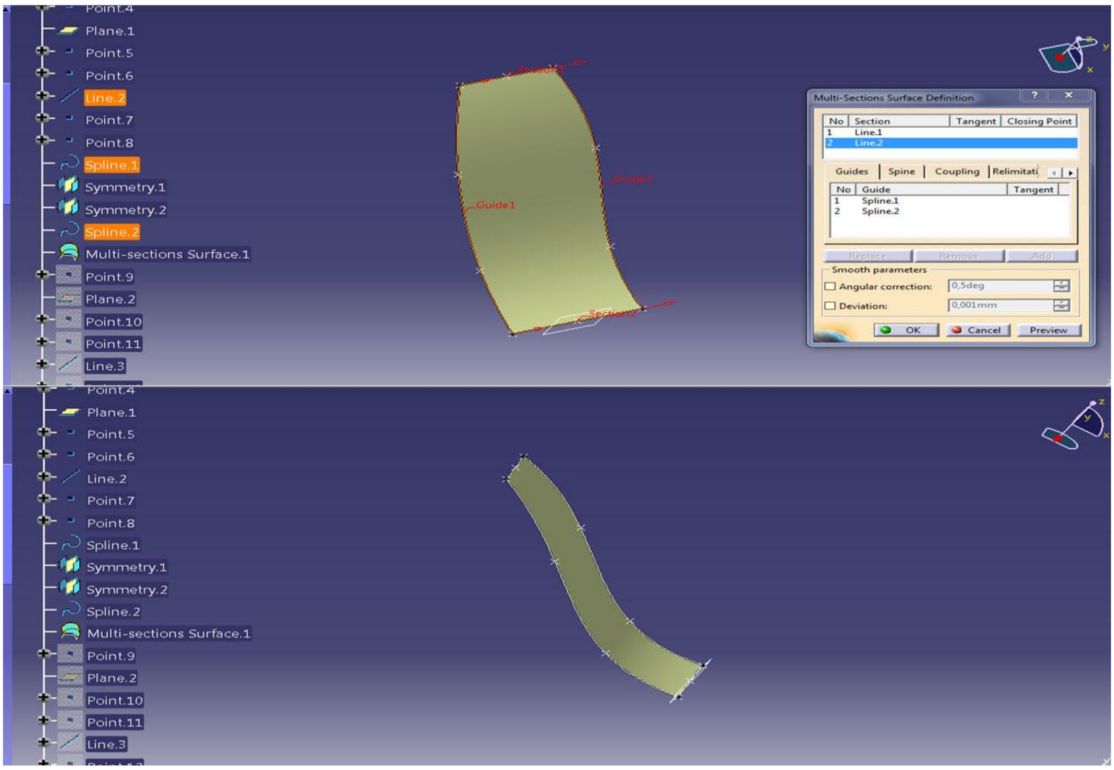
Picture 2.11.1.10 Execution of *"user pattern"* command.

2.11.2 CEFINITION OF THROTTLE'S, BRAKE'S AND CLUTCH'S PUSH ROD PEDAL BRACKET PARTS

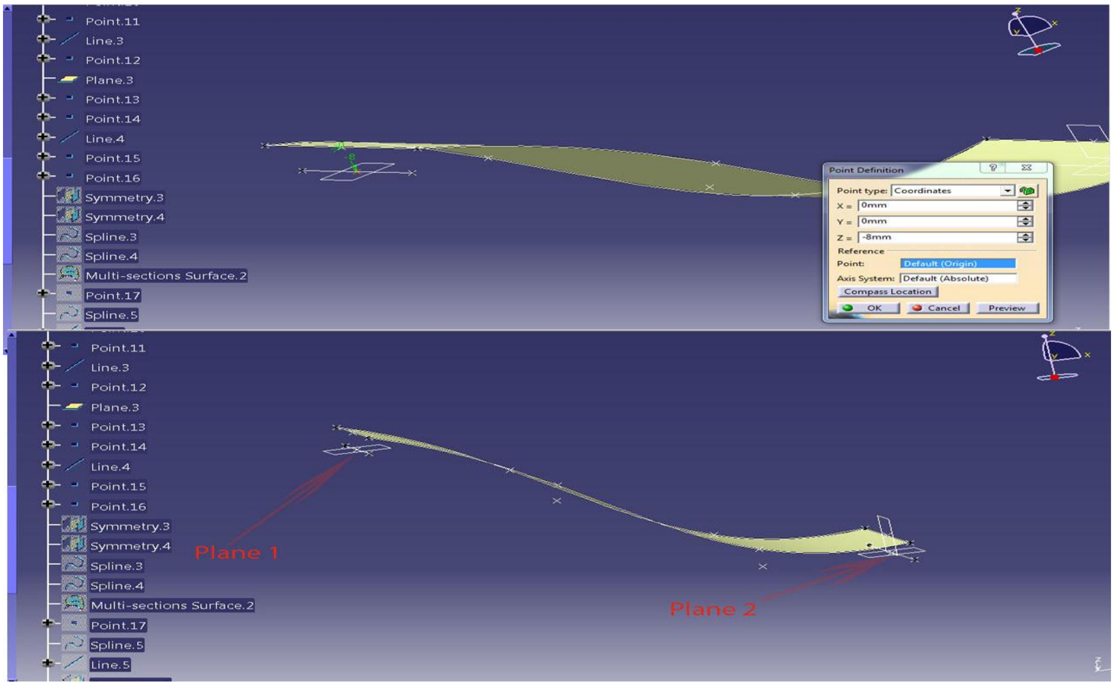
The push rod pedal bracket part was designed as the joining part that connects each of the pedal pad parts with the cabin's dashboard unit. As with all the other cabin parts, this certain part was designed as a parametric solid volume. For the shape's accurate design, an existing geometry of a real part was not used as reference. It was rather produced as a result of the user's personal experimentation. Firstly, the wireframe model was defined. This wireframe model formed a curvilinear section consisting of two B-Splines and two lines (Picture 2.11.2.1). Using the current section's contour as a reference, a multi section surface was created. For the execution of the *"multi section surface"* command, the two lines were imported as generative curves inside the command, while the remaining B-Spline curves were used as guides curves (Picture 2.11.2.2). Moreover, an additional wireframe section, similar to the first one, was created. For the representation of the current section, two ancillary planes were introduced inside the part's design. These two planes were used as references for the exact determination of the points which constituted the base of the B-Splines and the line creation. The distance of each plane from the existing multi section surface was measured equal to 8 mm (Picture 2.11.2.3). These planes were created for the purpose of being used as references in order for the second section to be formed. This section was designed as a resemblance of the first, with the major difference that it was narrower than its reference. Then, by using the same technique as before, the second multi section surface was constructed. The generative and guide curves that were defined for the execution of the current multi section surface creation command were respectively the same as before (Picture 2.11.2.4). Two additional lateral surfaces were created as multi section surfaces. As generative curves for the execution of the command the two planar edge lines that lied on the xz - plane were used, while as guide curves the corresponding B-Splines, which lied upon the same plane and were connected together with the two lines (Picture 2.11.2.5). The two remaining gaps between the two surfaces were covered with two additional surfaces. These two surfaces derived as filled surfaces. Naturally, as input at the boundary input selection inside the command definition each of the two sections' wireframe layout was inserted (Picture 2.11.2.6). Then, by joining the existing surface set with the "join" operation command, the resulting solid volume was formed with the execution of "*close surface*" command. As an input for the '*'object to close*" bracket inside the command window the set of surfaces, which was previously defined, was inserted. In order to complete the solid geometry, several fillets were inserted on the edges of the final solid geometry. These fillets were introduced into the final solid design solely for aesthetic purposes. As in the previously created solid parts, the shape of the fillets was completed with the use of *"edge fillet"* command. The radius of curvature for each edge filleting, was determined to be 1 mm (Picture 2.11.2.7).



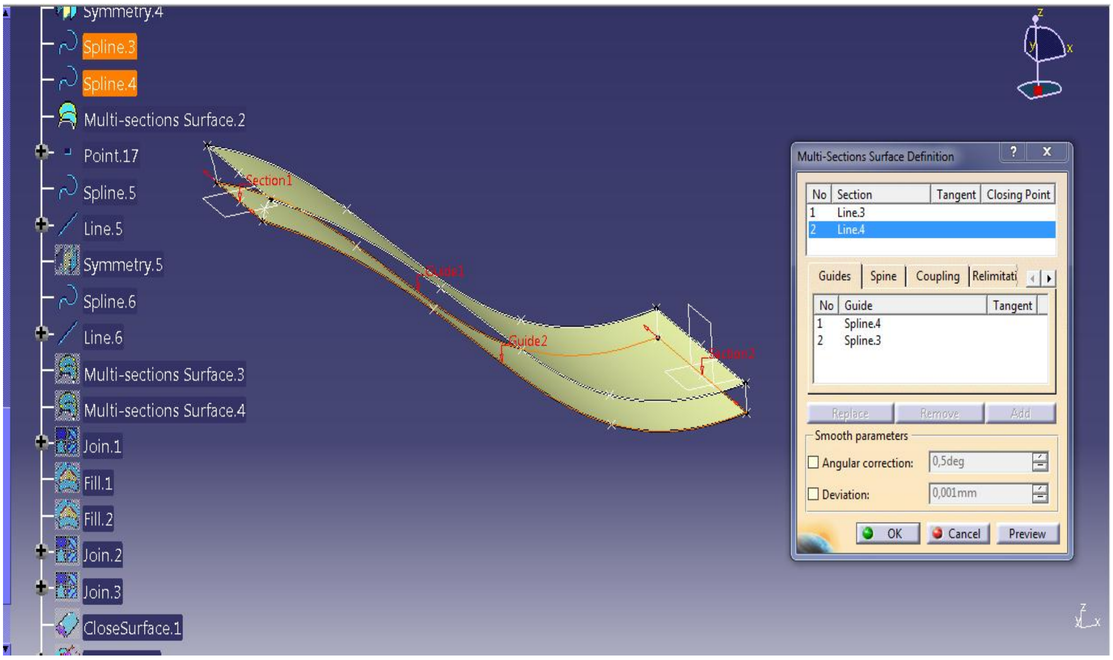
Picture 2.11.2.1: Push rod's pedal bracket wireframe model.



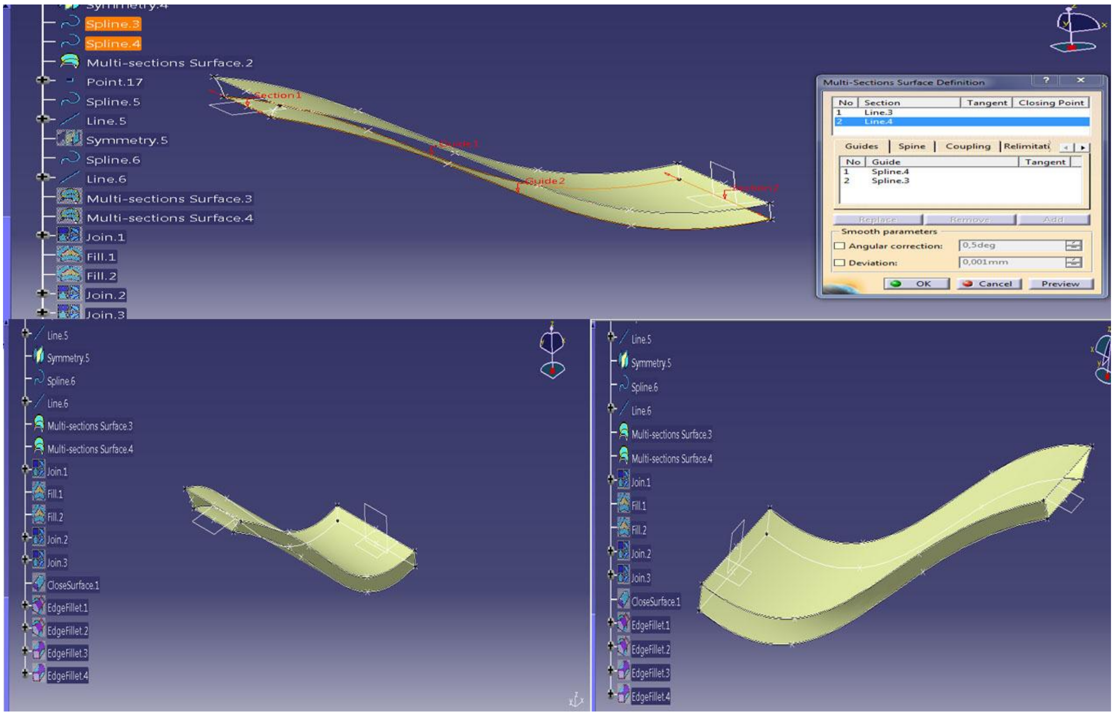
Picture 2.11.2.2: Creation of push rod's pedal bracket multi section surface.



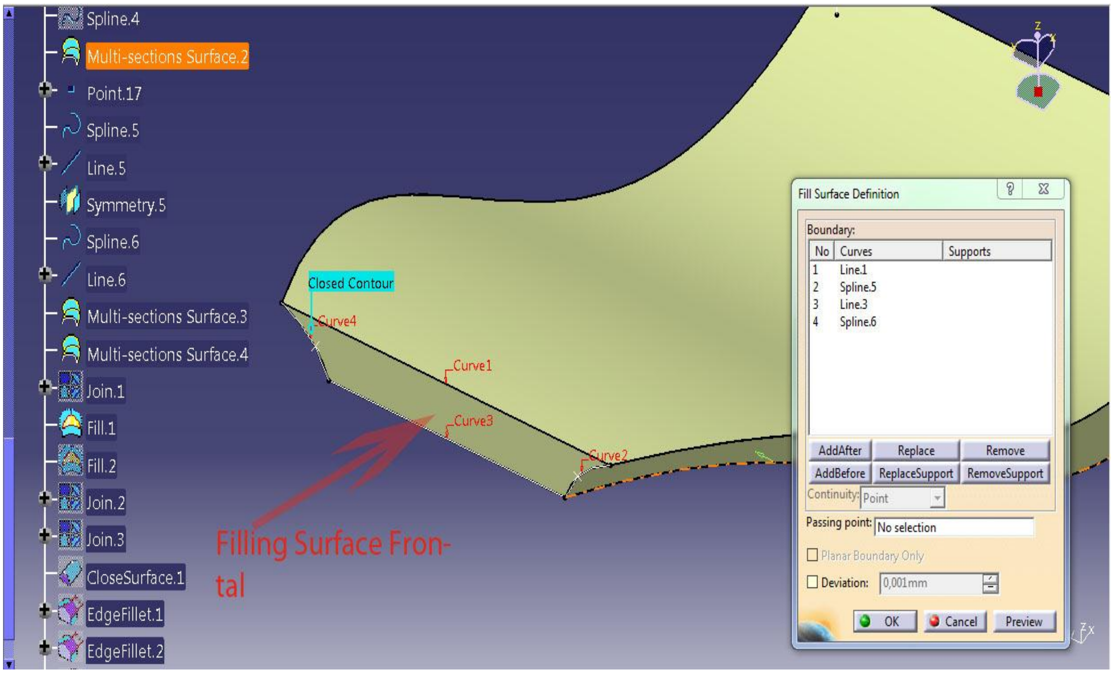
Picture 2.11.2.3: Definition of auxiliary planes.



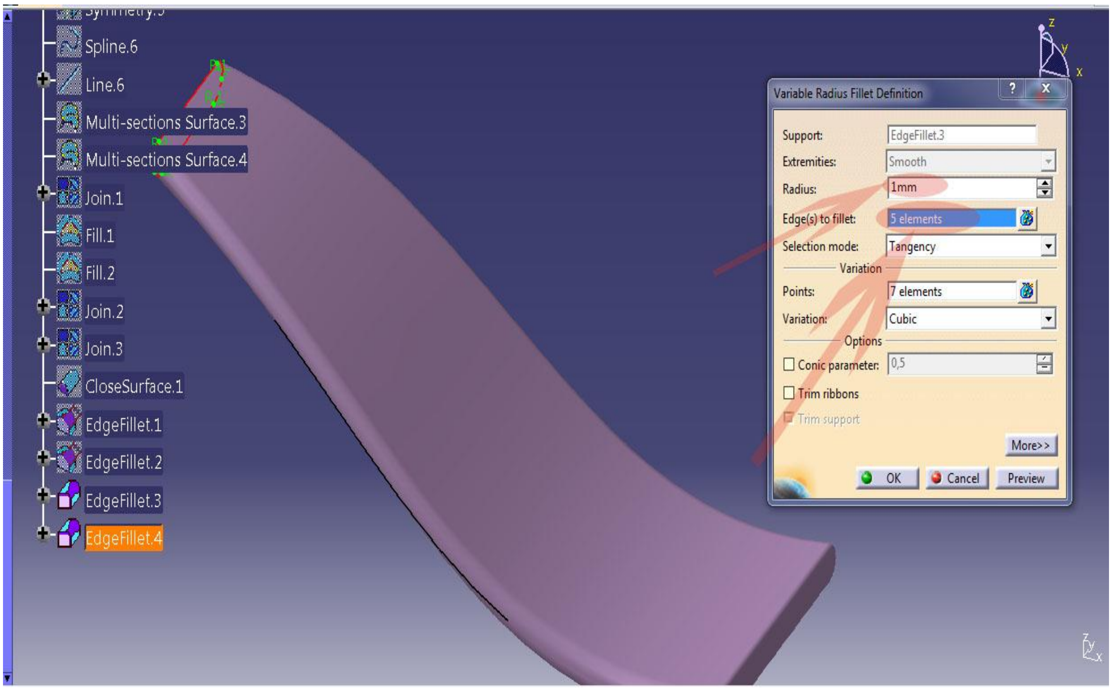
Picture 2.11.2.4: Creation of push rod's pedal bracket second section.



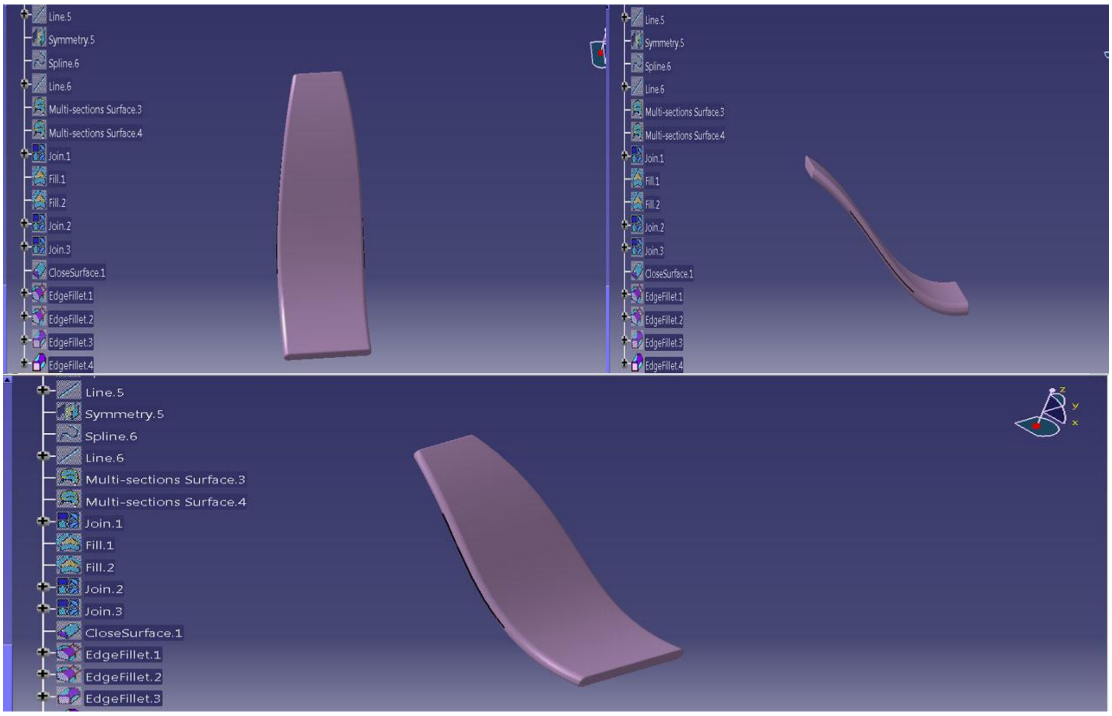
Picture 2.11.2.5: Creation of push rod’s pedal bracket peripheral surfaces.



Picture 2.11.2.6: Creation of push rod’s pedal bracket frontal surface.



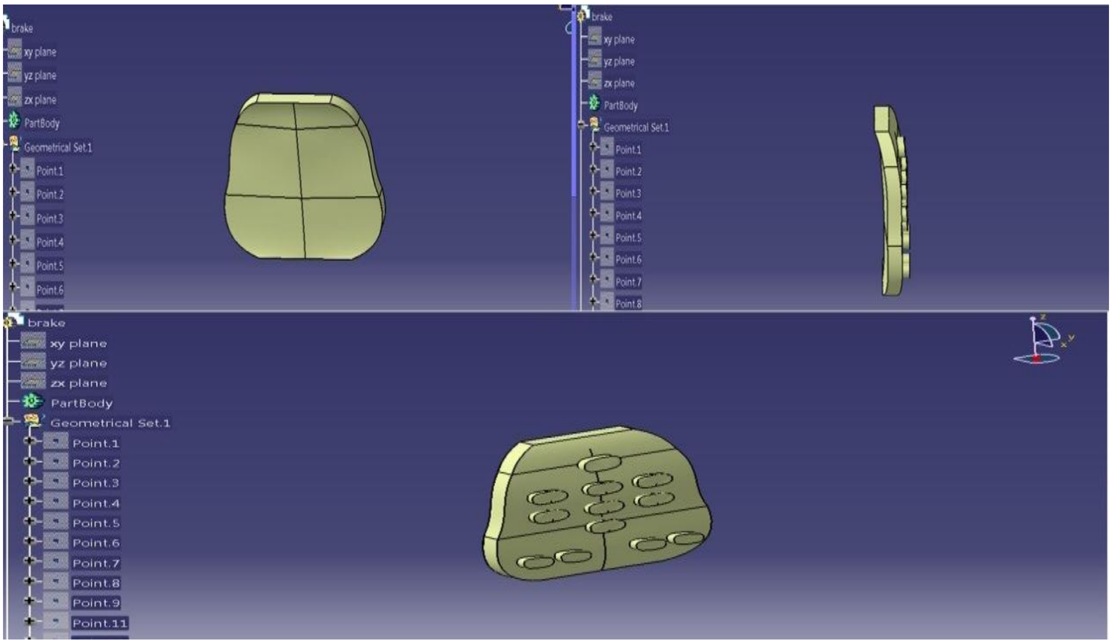
Picture 2.11.2.7: Definition of fillets on the rod's solid volume sharp edges.



Picture 2.11.2.8: Push rod's pedal bracket final solid volume.

2.11.3 DEFINITION OF BRAKE'S AND CLUTCH'S PEDAL PAD PART

The design process of the brake and clutch pedal part was exactly the same as with the throttle pedal part. The current geometry was designed as surface model for reasons of simplicity. An additional reason apart from that of simplicity was that the existence of this certain part for this thesis was not functional but auxiliary. As with all the other previous parts, this model was defined as a parametric one (Picture 2.11.3.1).



Picture 2.11.3.1: Surface model of brake and clutch pedal pad.

2.12 DEFINITION OF THE TIRE PART

The tire part was defined as a parametric solid volume. In order to produce a shape for the final solid geometry that would resemble a factual one, an existing tire was used as a reference for the creation of the solid geometry. The design of the current part was simpler than most of the previously created models. To this end, the fact that this part is a revolved geometry contributed greatly.

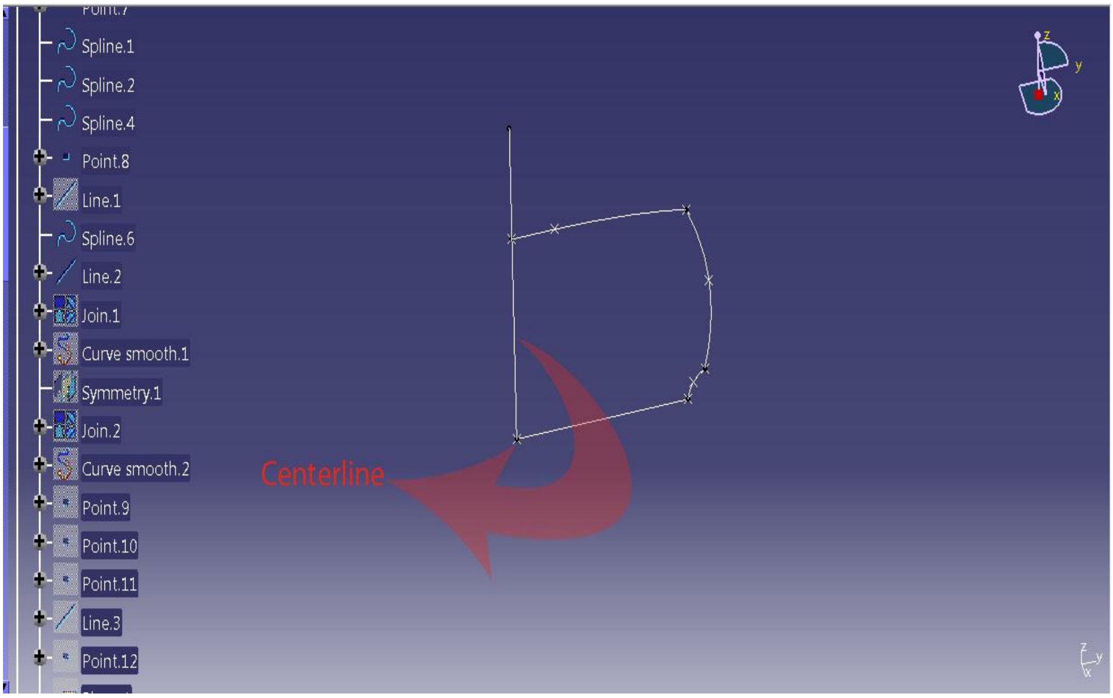
For the definition of the solid volume, the wireframe model was created first. The section's centre line is shown in Picture 2.12.1 below. The wireframe section consisted of four B-Spline curves. An important feature that is not obvious from the picture is that the section is not closed. In order to produce the second symmetrical half of the final wireframe section, a unification of the four B-Splines was needed to be performed. Consequently, a "*join*" operation command was opted for in order to produce a new unified curve that consisted of the four B-Spline curves. Then, a curve smooth command was also used to ensure second degree curvature to the connecting points between the two B-Splines. Then, the second symmetrical half of the final wireframe model was formed using "*symmetry*" operation command. As inputs inside the command's parameters window, the section's unified curve that was previously created was imported inside the ''element'' bracket, while as symmetry axis the section's centre line was selected (Picture 2.12.2). Then, the two symmetrical halves were unified together with a *"join"* operation command. This process had to be followed as it will become clearer at the stage of the surface model creation. Next in sequence followed the definition of the revolution axis of the current wireframe section. The axis of the current model was defined as a line parallel to a perpendicular unit vector of the xz - axis (Picture 2.12.3). The surface model derived by executing the *"revolve"* command. As profile inside the command's window the wireframe section was inserted and as axis of revolution the previously created line was selected. The profile was defined to be revolved around the axis for 360 degrees (Picture 2.12.4). The final solid geometry was formed by executing the "*close surface*" command. As input inside the command's execution window the existing revolved surface was used (Picture 2.12.5). Additionally, for aesthetic purposes, a number of corrugations was imported in the solid volume geometry. The corrugations were introduced on the external surface of the solid. In Catia software the corrugations could be inserted using the available Boolean operations of the software. Consequently, in order to remove material from the solid geometry it was needed for two volumes to exist. For the design of the auxiliary volume, from which the final geometry was going to be derived, a circle was first defined. The circle was introduced as being tangent to the tire's external surface by using the circle creation command and more specifically by exploiting the Center and Point option inside the ''Circle Type'' bracket (Picture 2.12.6). Then, by using the wireframe "*Translate*" command, an additional tangent circle was introduced on the existing solid. The new circle had a distance of 30 mm from the first one at a perpendicular direction towards xz - plane. The exact distance measurement between the two circles was defined inside the command window. Moreover, a point was inserted on the previously created circle. Then by using this point as a reference, an additional plane parallel to yz plane was imported. Additionally, a circle of 10mm radius having as centre the previously created point was designed on the plane (Picture 2.12.8). Next in sequence was the creation of a sweep surface. This surface was designed using the previously created circle as generative profile and the tangent to the solid volume circle as a guide curve. The result of this process was the creation of a closed surface, as it is shown in Picture 2.12.9 below. The solid volume derived by executing the "*close surface*" command (Picture 2.12.10). A second volume was constructed by executing the *"translation"* operation command. The distance between the two volumes was set as 30 mm and the translation's direction was created as perpendicular to the xz plane (Picture 2.12.11). Then, by performing two remove Boolean operations in sequence, two corrugations were created on the tire's solid geometry. As base object inside the command window the tire's solid volume was defined while as object to be removed the assistive circular volume for each command's execution was chosen respectively (Picture 2.12.12). Two more additive assisting volumes were constructed on the solid geometry. These volumes were defined as symmetries of the first two initial auxiliary volumes. As plane of symmetry for the implementation of each symmetry operation the xz plane was defined (Picture

2.12.13). By following the exact same technique as before, the tire's solid geometry with the additional corrugations was formed (Picture 2.12.14). Furthermore, for reasons of completeness an additional, and different in terms of shape, corrugation geometry was designed. Firstly, its wireframe model was defined. To this end, a tangent to the existing solid volume plane was defined. By using this plane as support, an elliptical wireframe closed section was created (Picture 2.12.15). Then, using the section's contour as input, a filled surface was created. Next, an extruded surface derived using the previously designed surface as an input. The extrusion's direction was perpendicular to the solid tangent plane in both sides (Picture 2.12.16). The solid volume was formed by executing the "*close surface*" command. As input inside the command's window the previous extruded surface was inserted (Picture 2.12.17). Moreover, 40 additional similar volumes were constructed by executing the *"Circular Pattern"* command. The angular space between two volumes was set to 15 degrees (Picture 2.12.18). In addition, forty more solid and symmetrical geometries were constructed by executing the ''symmetry'' operation command. As reference element the circular pattern was inserted, while the xz plane was chosen as plane of symmetry (Picture 2.12.19). Then, by using the same technique as with the creation of the first two corrugations, two sequential remove Boolean operations were executed. The definition for each pattern was the same as before (Picture 2.12.20). For ornamental purposes several fillets were inserted on several edges of the solid geometry (Picture 2.12.21). As for the dimensions measurement, they derived from the indicator information that was inscripted on the actual reference tire. The main dimension components were the tire's width, thickness and rim diameter.

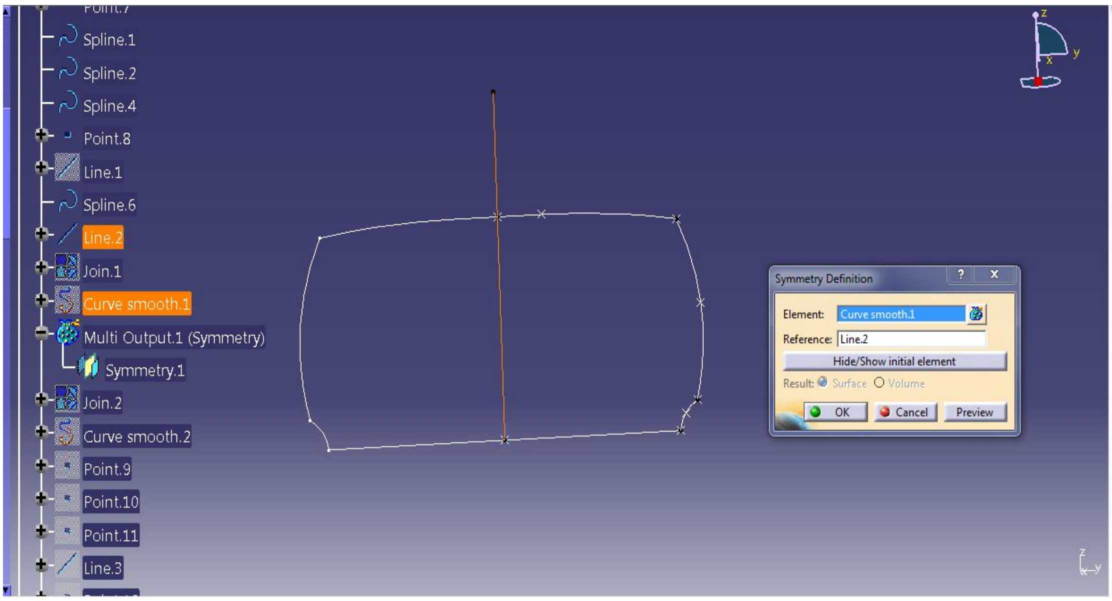
The exact numerical value for each of these dimensions are shown in Table 2.12.1 below.

Table 2.12.1

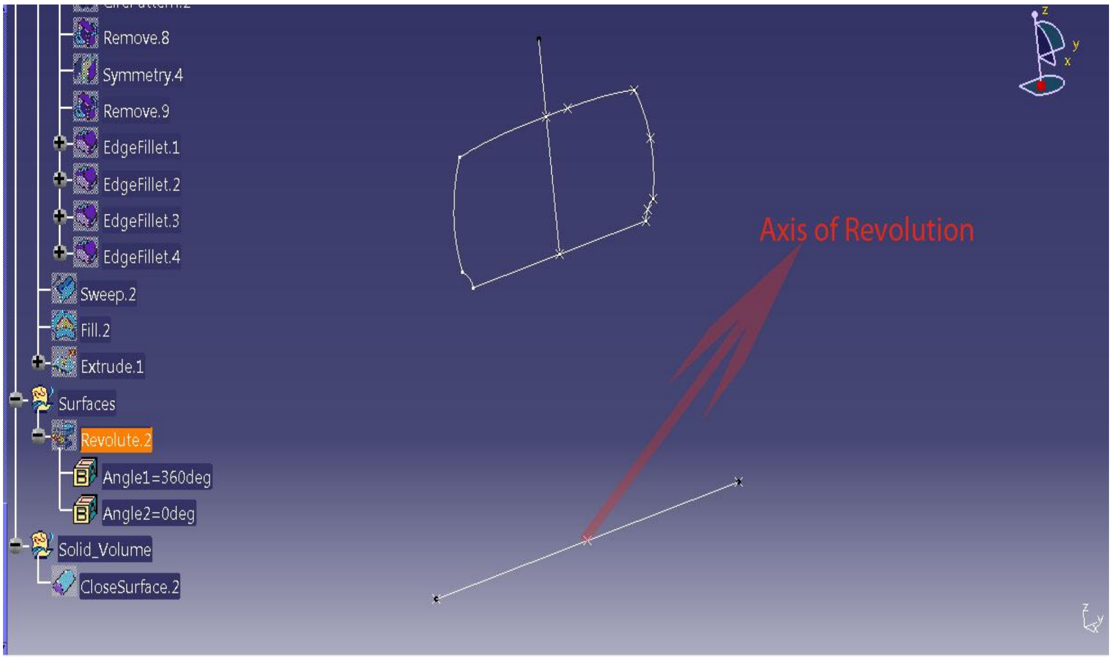
Dimensions / Numerical Values (mm)	
Width	235
Thickness	129,25
Rim Diameter	200



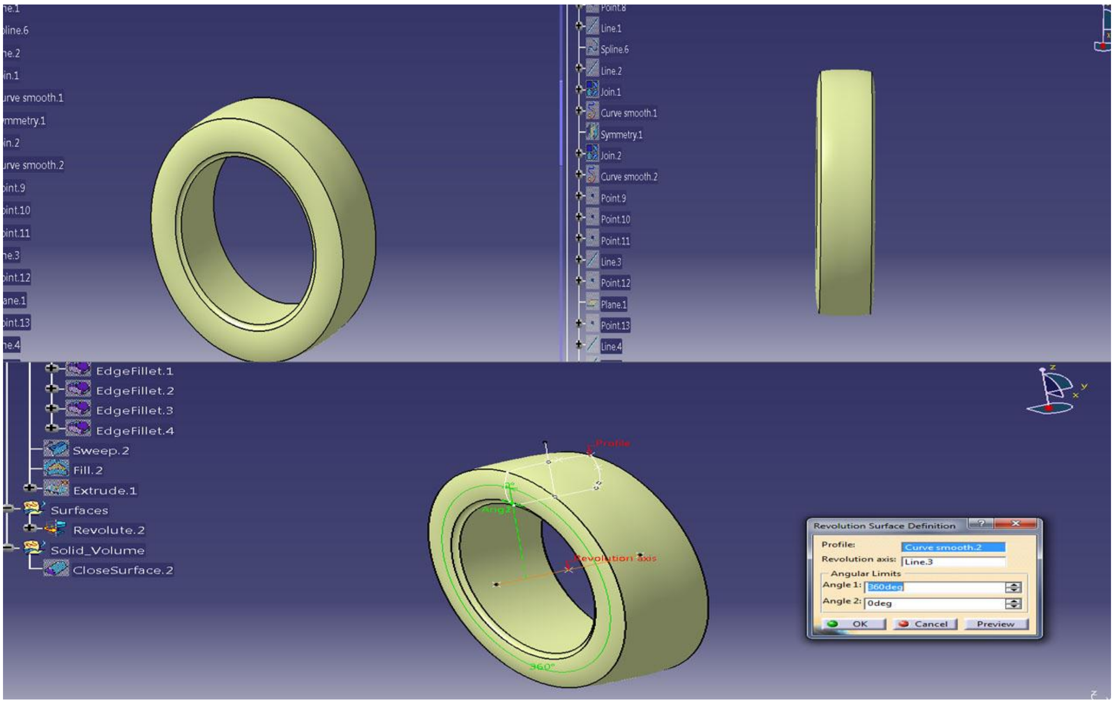
Picture 2.12.1: Definition of tire's centre line and wireframe section.



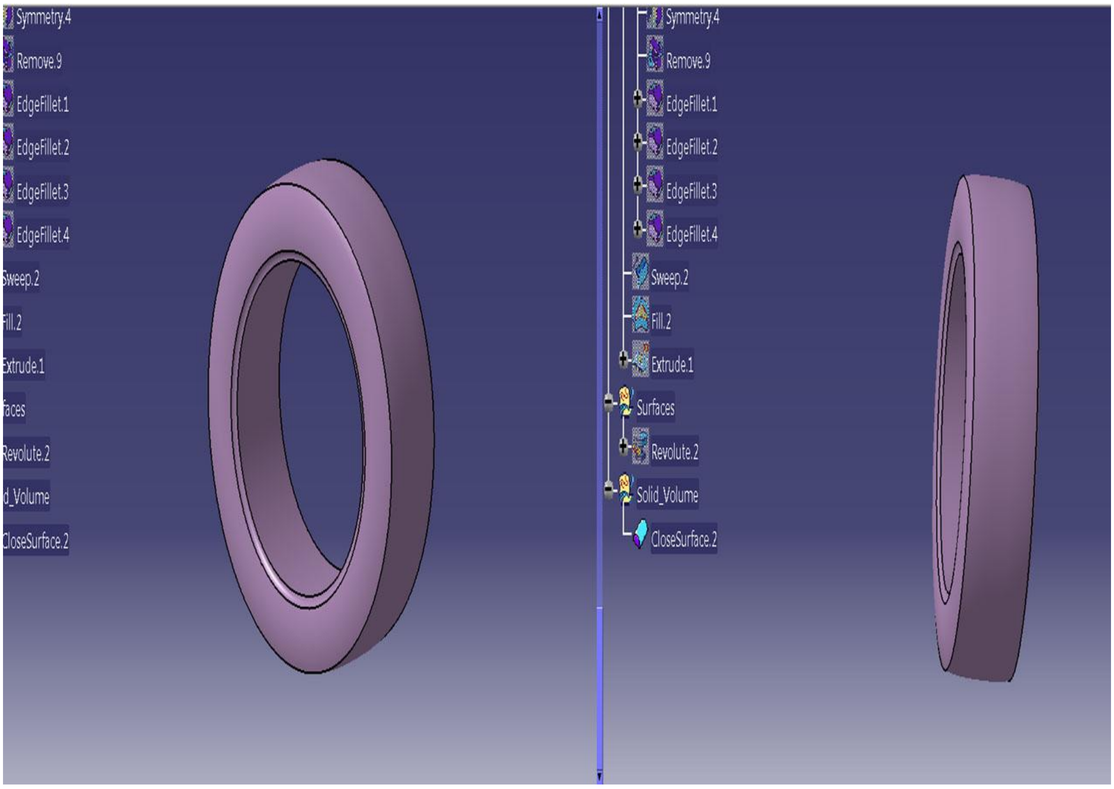
Picture 2.12.2: Designation of tire's final wireframe section.



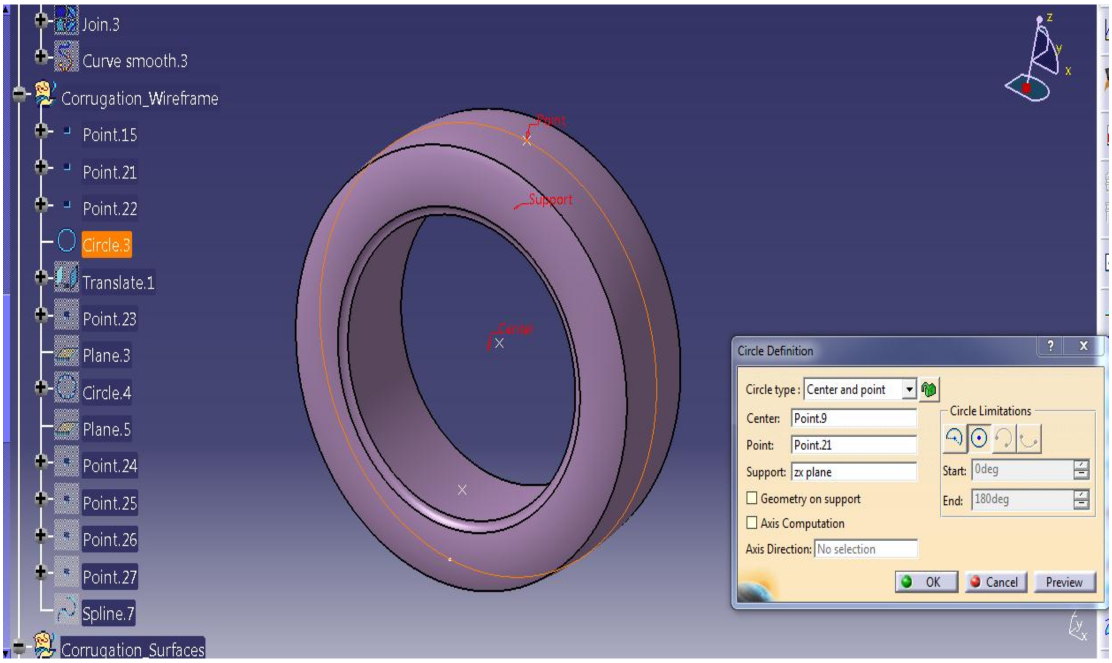
Picture 2.12.3: Designation of tire’s axis of revolution.



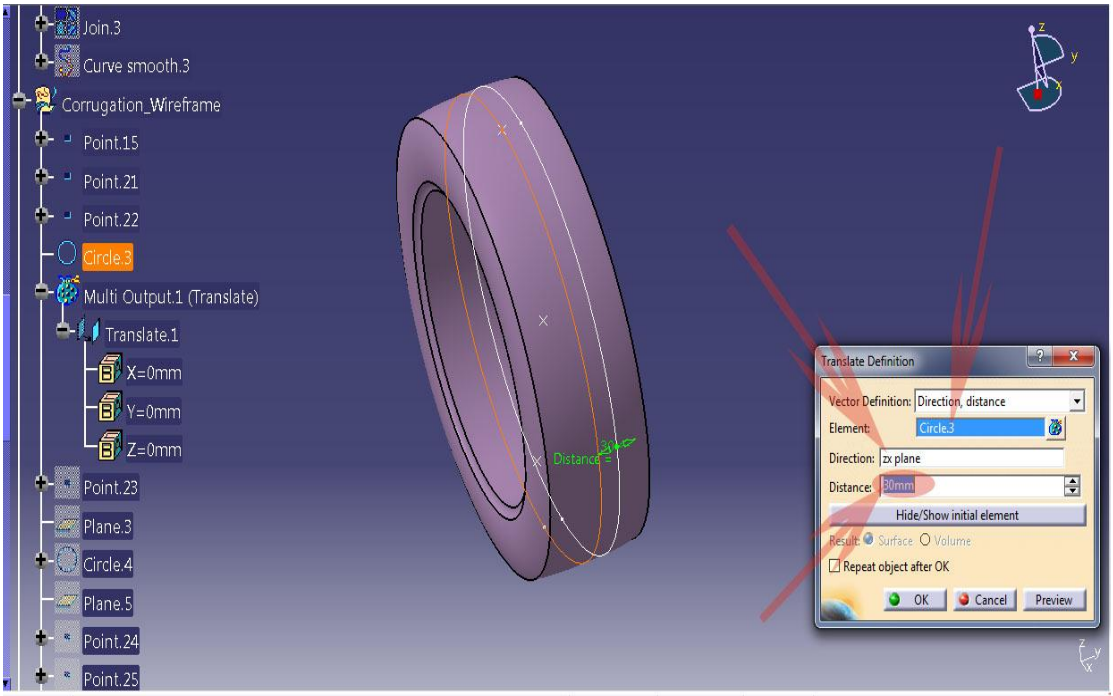
Picture 2.12.4: Creation of tire’s surface of revolution.



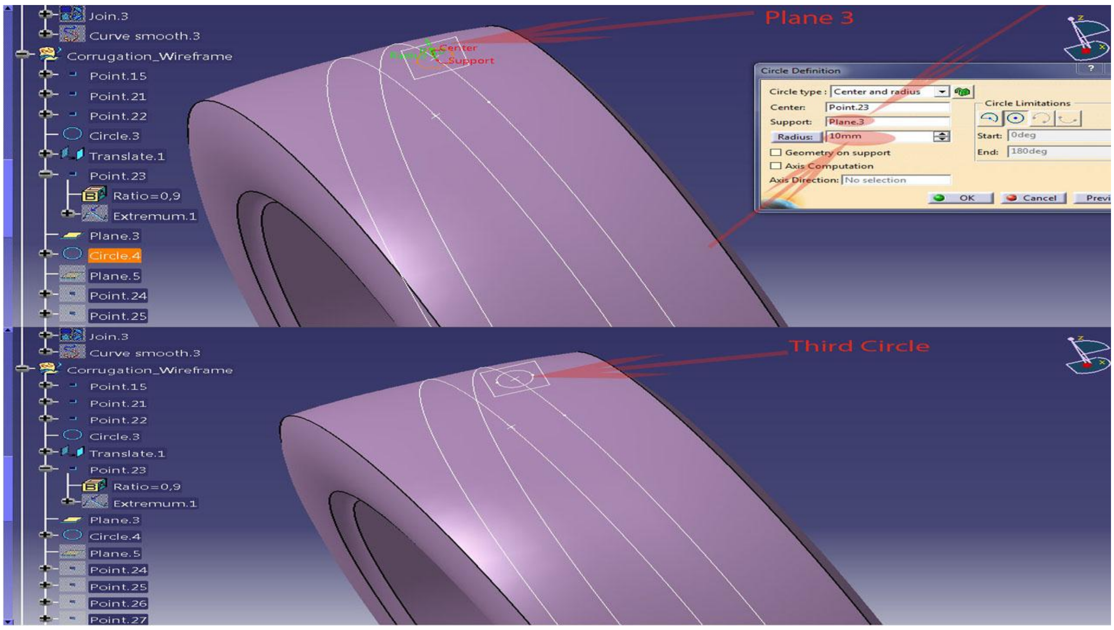
Picture 2.12.5: Creation of tire's initial volume.



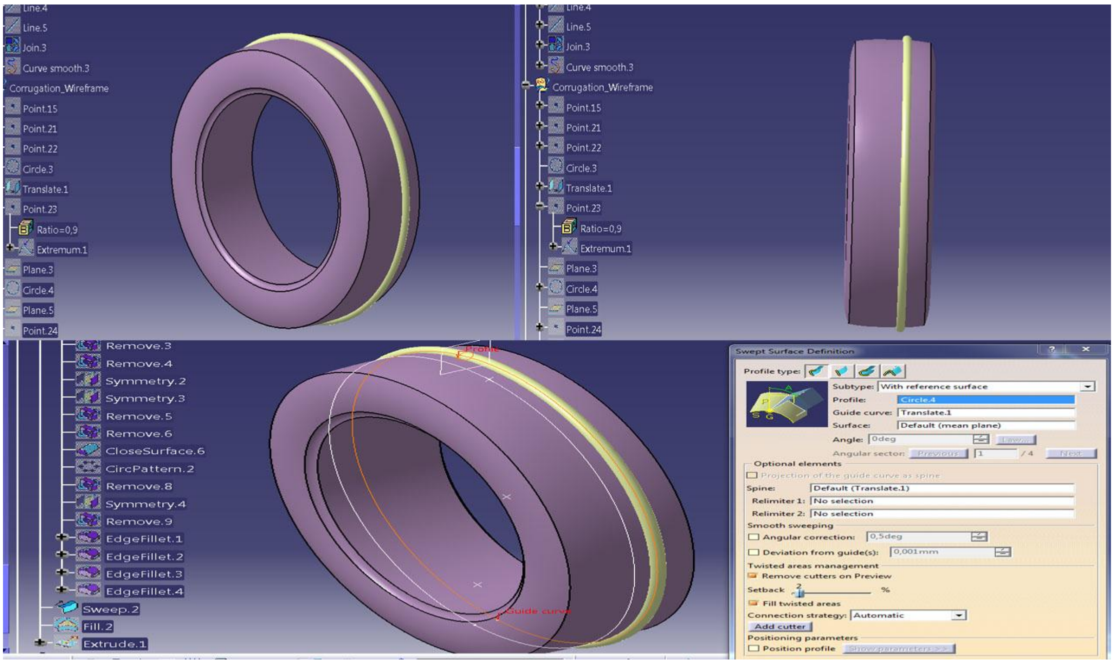
Picture 2.12.6: Definition of additional circle section.



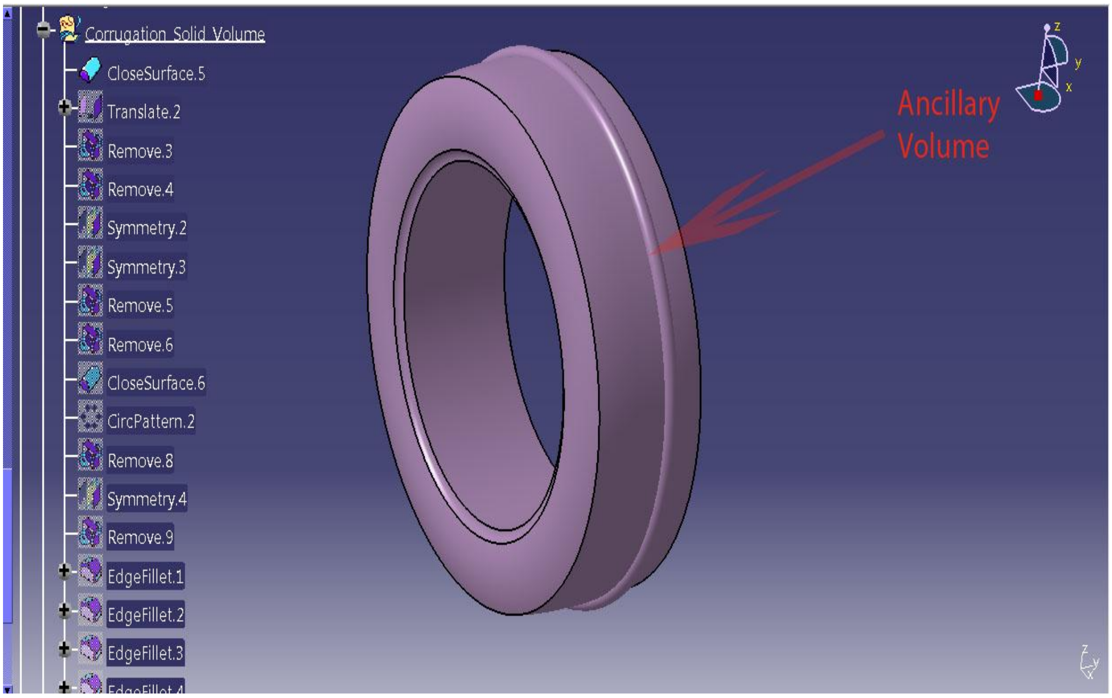
Picture 2.12.7: Definition of second circle section.



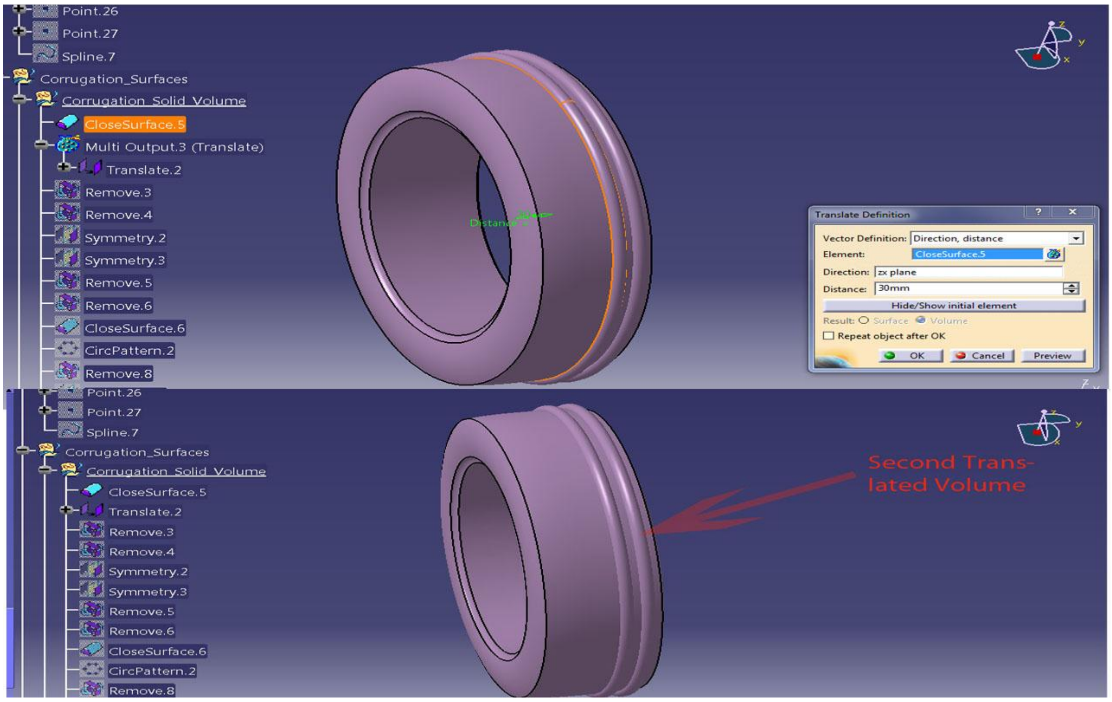
Picture 2.12.8: Definition of a circle section vertical to the first circle section.



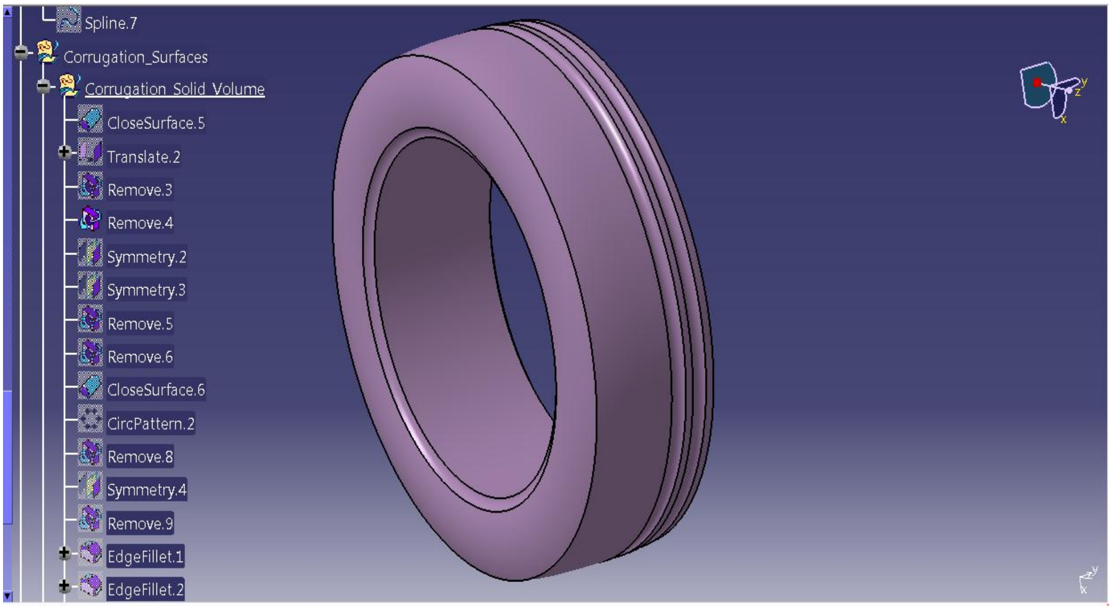
Picture 2.12.9: Creation of sweep surface.



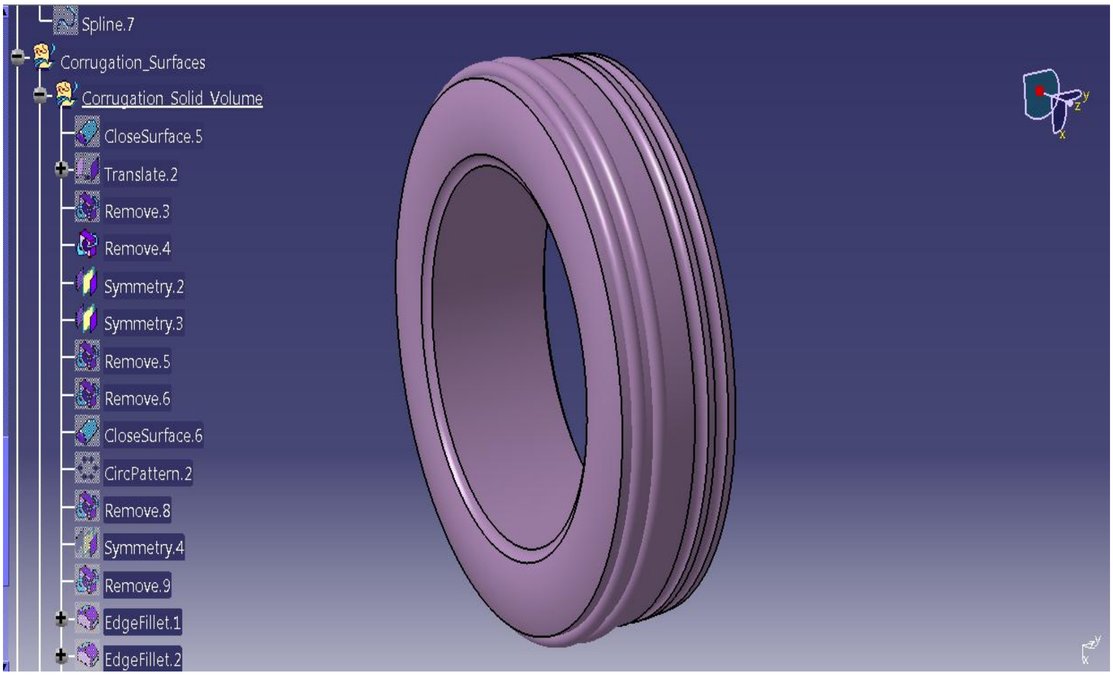
Picture 2.12.10: Creation of volume from the previous surface model.



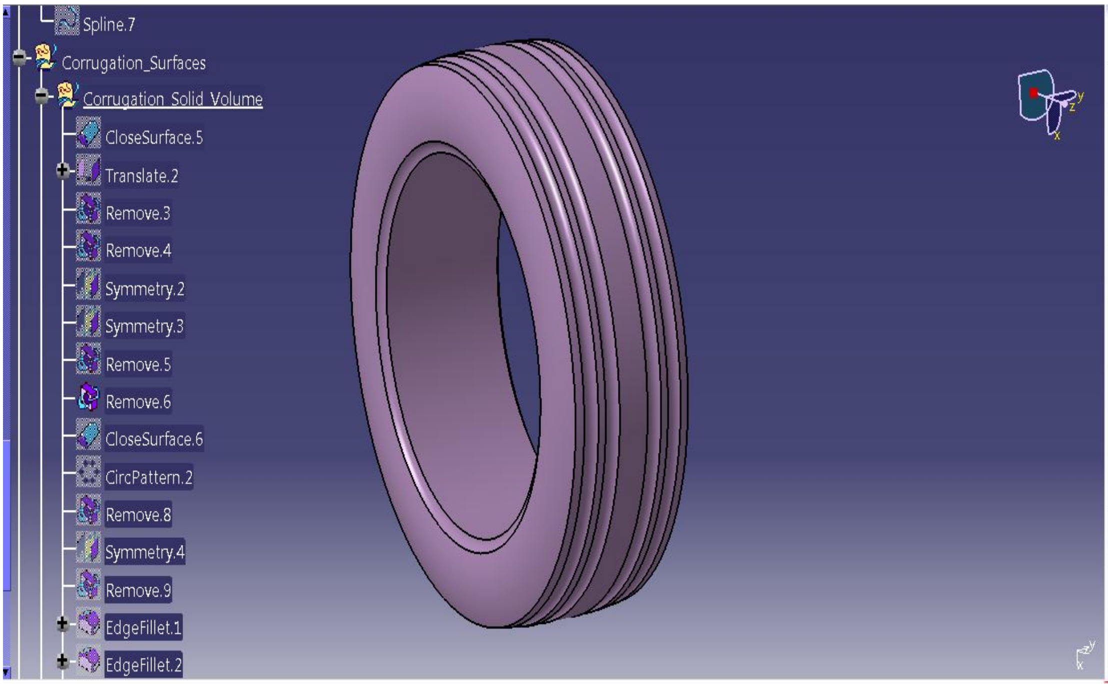
Picture 2.12.11: Creation of a second similar volume in sequence.



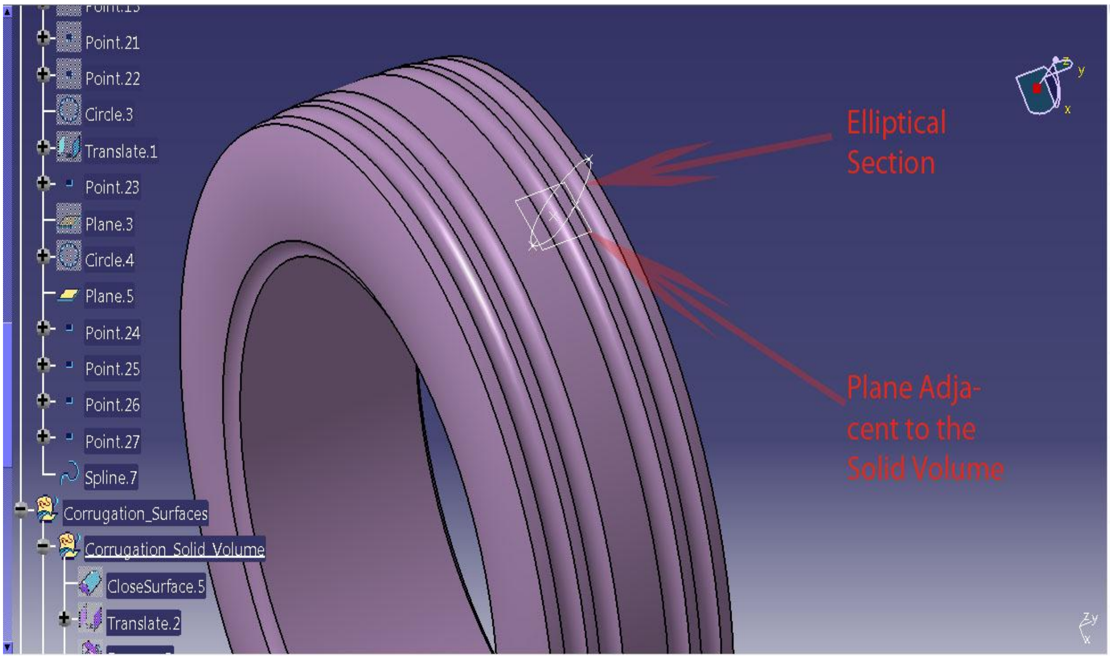
Picture 2.12.12: Definition of tire’s initial volume with the first two grooved channels.



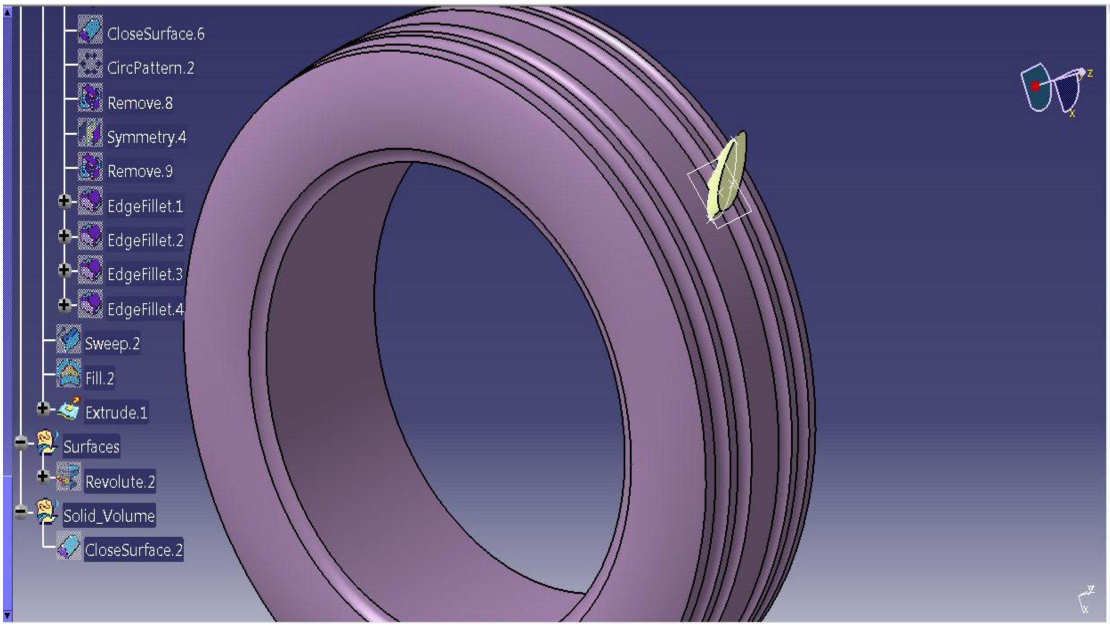
Picture 2.12.13: Creation of two additional circular volumes on the tire’s volume.



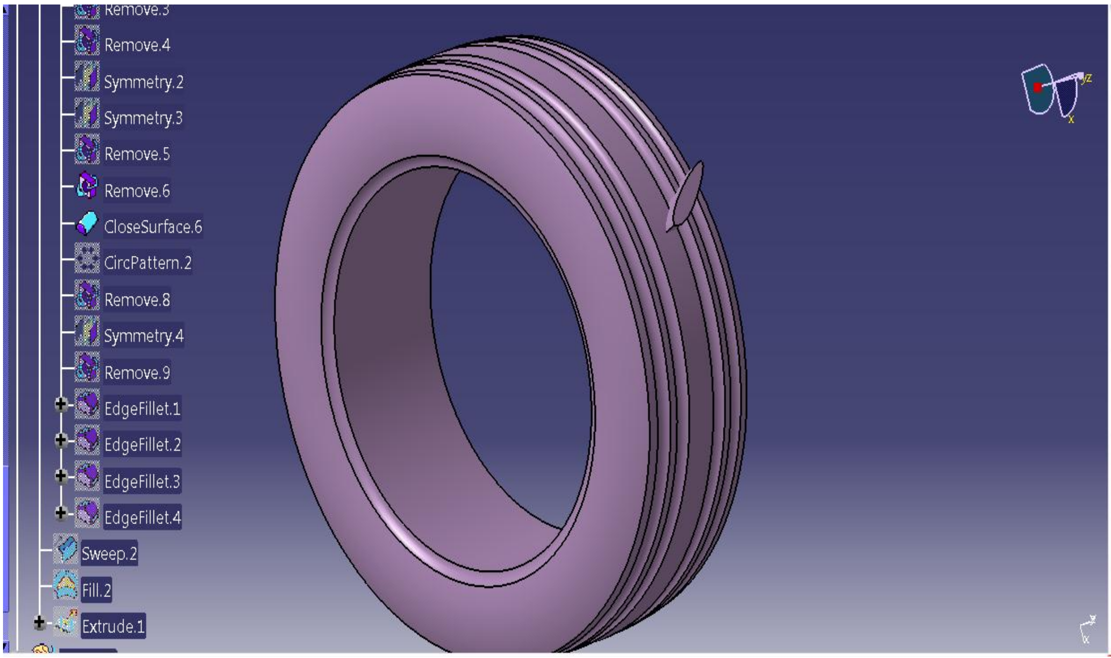
Picture 2.12.14: Creation of two more grooves on the tire’s volume.



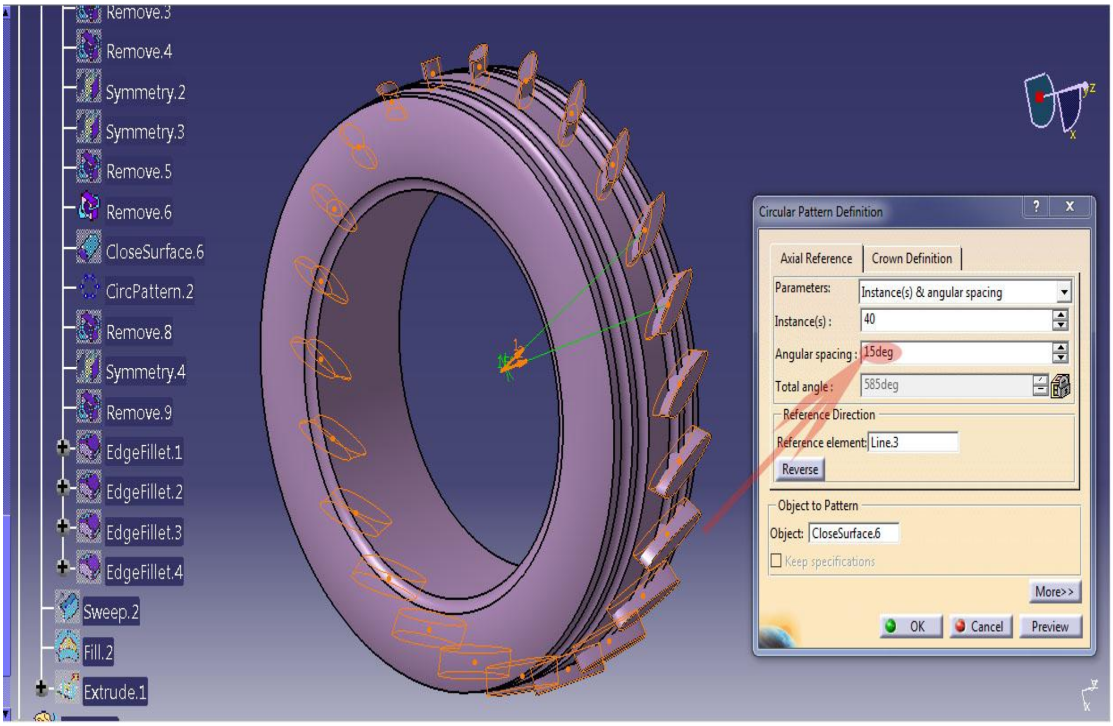
Picture 2.12.15: Wireframe model of tire's sipe.



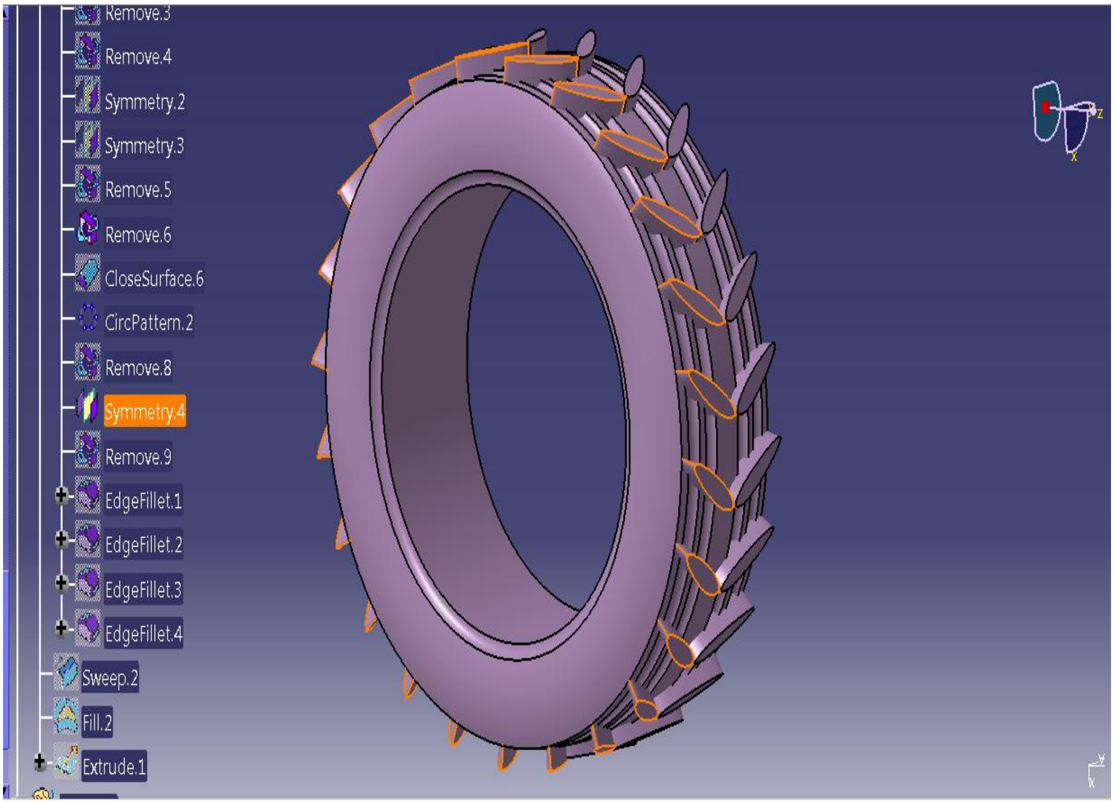
Picture 2.12.16: Initiation of sipe's surface model.



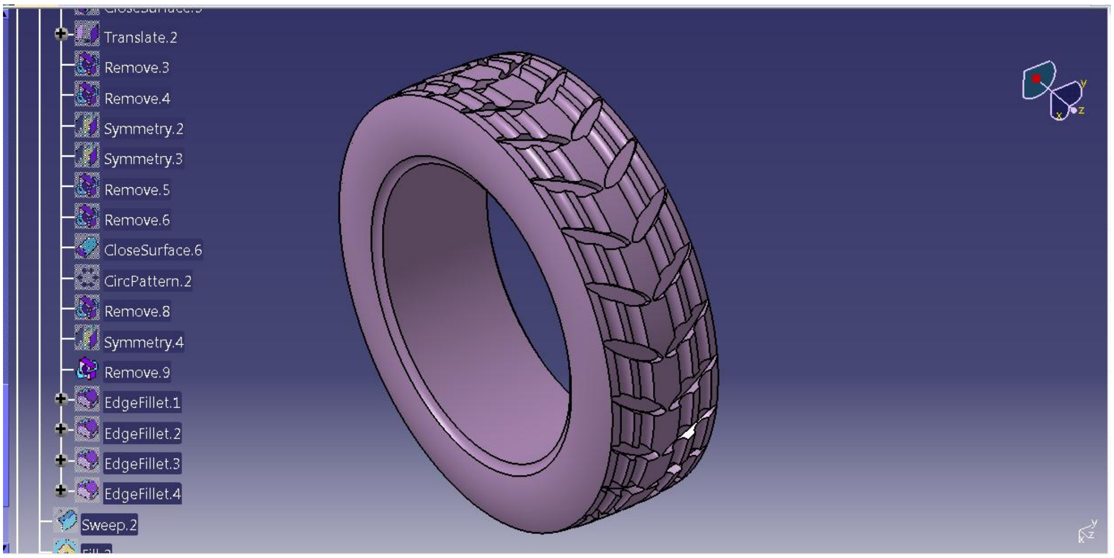
Picture 2.12.17: Creation of sipe volume.



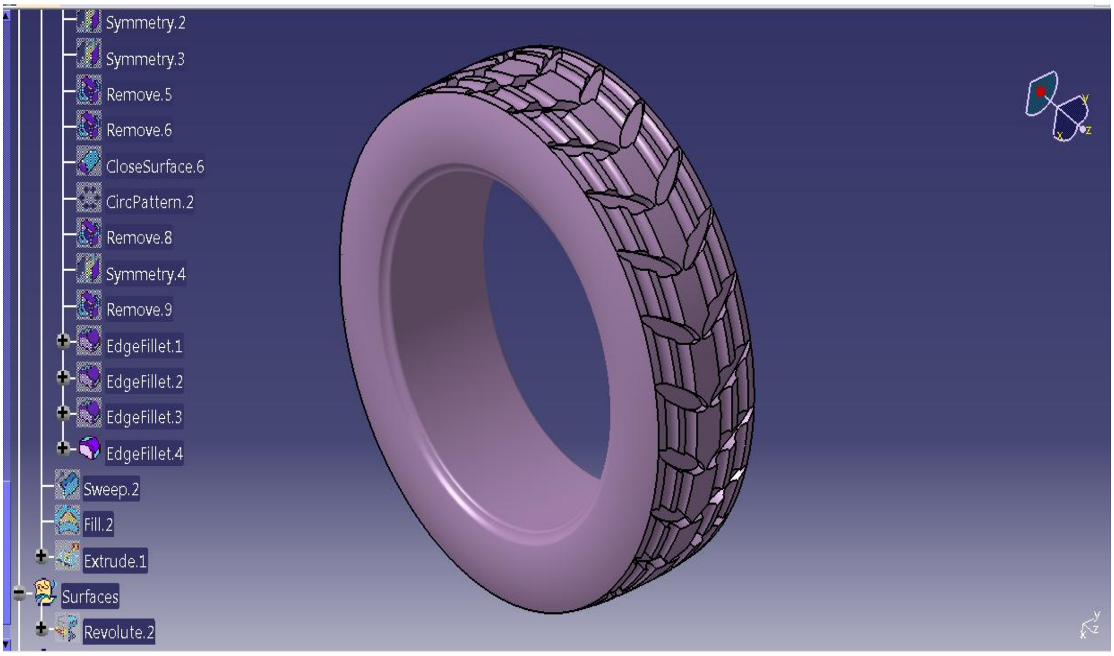
Picture 2.12.18: Execution of circular pattern on sipe volume.



Picture 2.12.19: Definition of symmetrical sipe volumes.



Picture 2.12.20: Tire's volume with grooves and sipes.



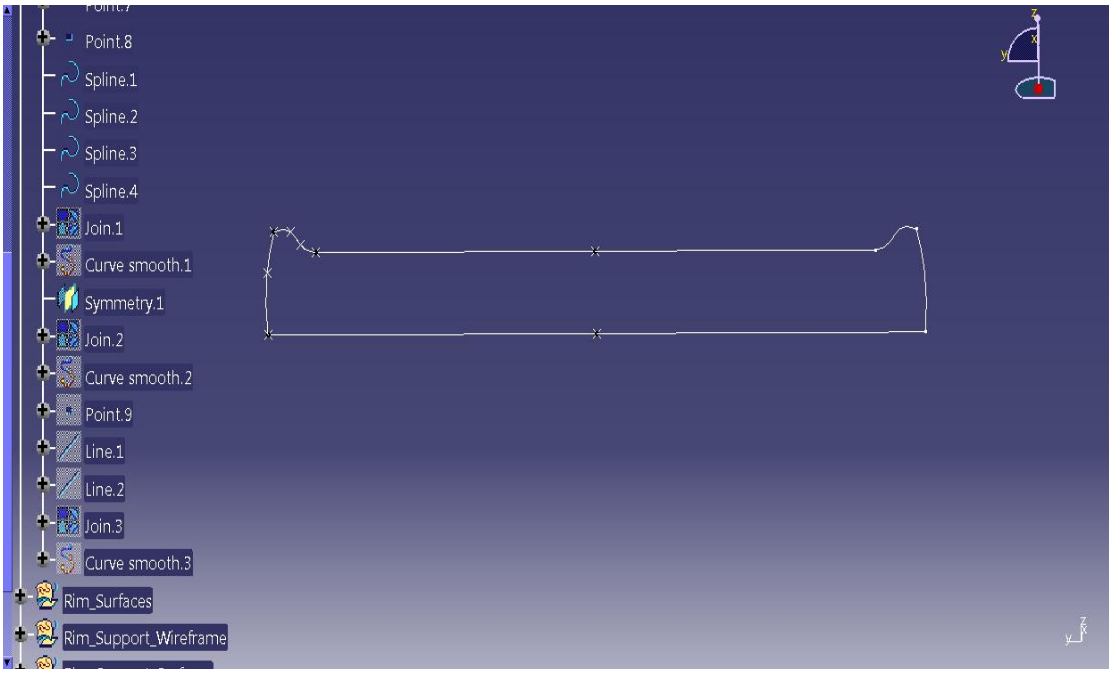
Picture 2.12.21: Tire's final volume.

2.13 DEFINITION OF THE RIM PART

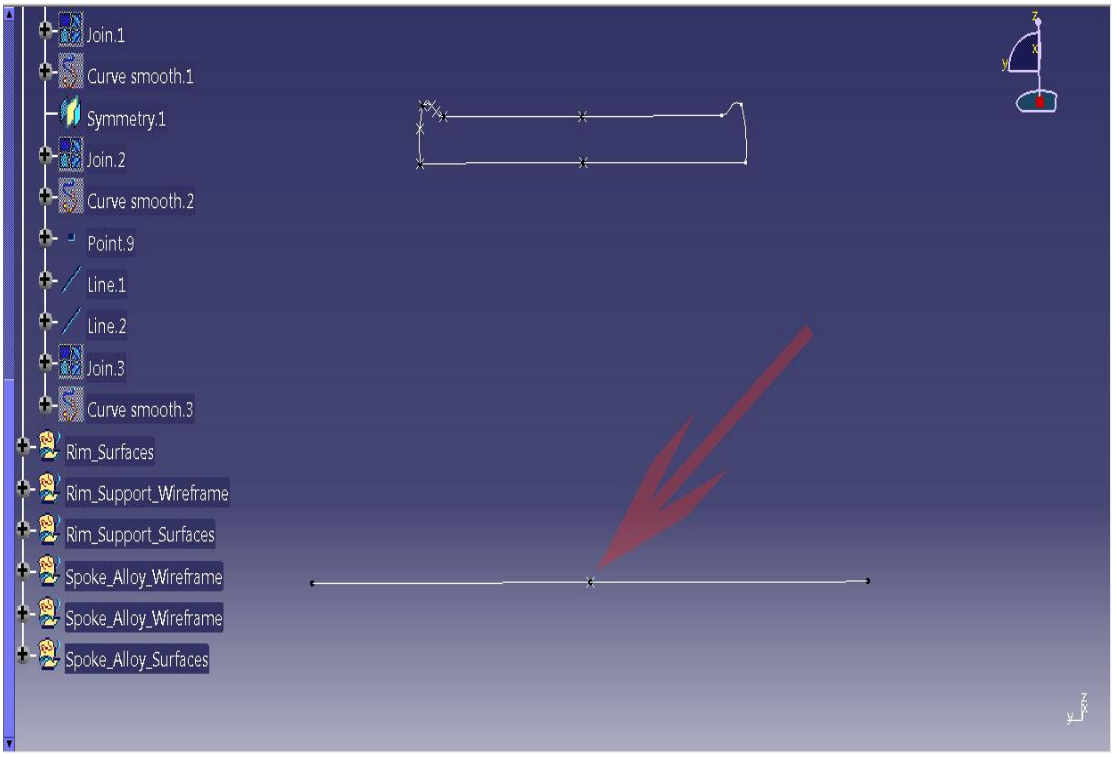
This certain part is the final independent part that was created. As with all the other parts above, its design was parametric. To approach an actual rim shape as much as possible, both images from an existing rim and images downloaded from the web were used. It is important to mention at this point that neither the actual rim's design nor any design depicted in the pictures was followed as such. The final part design was primarily a result of the designer's creativity. As for the dimensions measurement, it was made possible to secure accuracy, thanks to the fact the rim's diameter was designed using the tire's dimensions.

Firstly the wireframe model was designed (Picture 2.13.1). The model presents a symmetry with respect to the xz plane. As a result, only half of the wireframe model was defined. An additional feature was that the section of the wireframe model was not closed. Furthermore, the wireframe section consisted of four B-Splines. Then, the four splines of the wireframe were unified together with the use of "*join*" operation in order to form a single entity. Then, by executing a "*symmetry*" operation command the second symmetrical half was defined. As input for the element option inside the command window the previously joined entity was inserted, while as plane of symmetry the xz plane was imported. Additionally, a single line was defined (Picture 2.13.2). Its functional purpose is going to become clear later in the analysis. Next in sequence was the definition of the rim's metallic hoop surface. Its surface was designed as a revolved surface (Picture 2.13.3) As a generative profile for the command's execution the closed section wireframe was imported. As axis of revolution the previously defined line was selected. Then, the solid volume was formed by executing the "*close surface*" command. As input inside the command's window the existing solid volume was inserted (Picture 2.13.4).

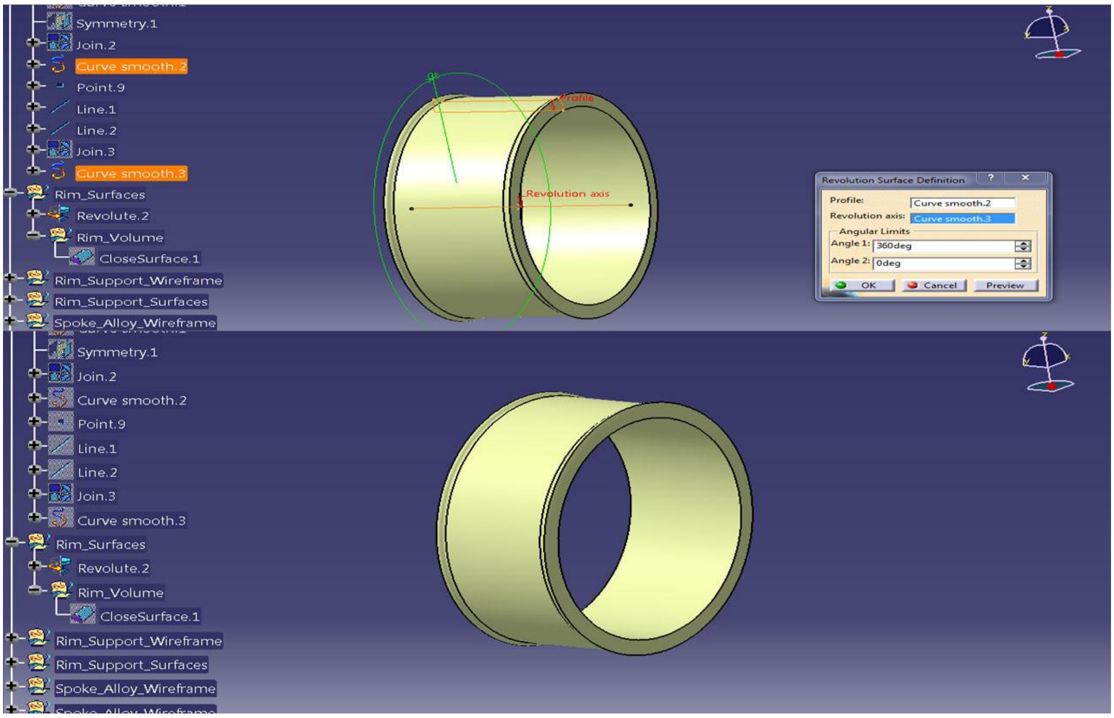
Next in sequence was the design of wheel central hub cap, inside the rim's metallic hoop. Its wireframe section consisted of two circles, which were defined on two sequential planes with an approximate distance of 50 mm from each other (Picture 2.13.5). Furthermore, by using the broader circle's contour as a reference, an extruded surface was constructed. The length of the extrusion was defined as 30 mm towards the positive direction of the y-axis (Picture 2.13.6). Moreover, an additional surface was created. This surface was created with the use of "*blend*" surface creation command. As profile inputs that were going to be blended the extruded surface's and the narrower circle's layout were inserted (Picture 2.13.7). The two gaps, which existed on the internal ring's front and rear sides, were covered with filled surfaces (Picture 2.13.8). Then, by joining the internal ring's set of surfaces together with sequential healing commands, a unified entity was formed. The final solid volume derived by executing the *"close surface*" command, using as input element the previously created unified set of surfaces (Picture 2.13.9). The next stage of this part's design was the design of several spokes. Firstly, the wireframe model was created. This certain wireframe model was different in comparison to others. Its primary difference was that it consisted of two separate closed sections, as shown in Picture 2.13.10 below. After the unification of each command's section, these new entities were blended together in order to form a single surface. The inputs for the resulting surface were the same as with the wheel central hub cap's blended surface (Picture 2.13.11). The solid geometry was formed by executing the "*closed surface*" command. For aesthetic purposes several edge fillet operations were opted for on the spokes design (Picture 2.13.12). The overall number of spokes on the rim's 3-D design derived with the execution of an additional *"circular pattern"* command. The total number of spokes was defined as 5 in total with the angular space between two of them being equal to 72 degrees (Picture 2.13.13). After inserting several additional edge fillets on the design's sharp edges, the final 3-D solid volume was completed (Picture 2.13.14).



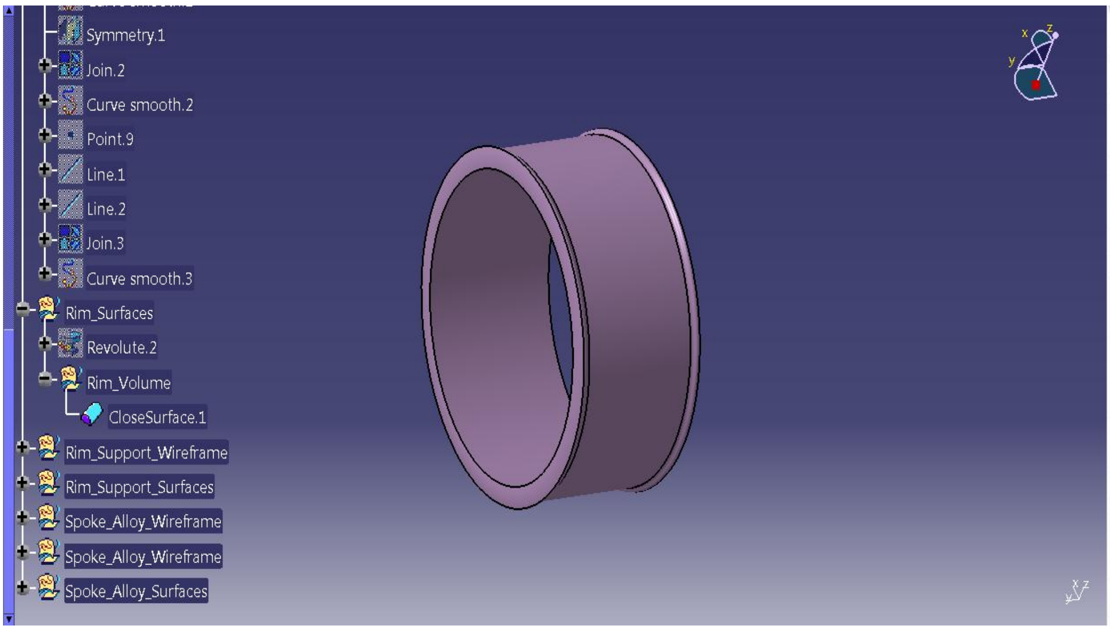
Picture 2.13.1: Rim's wireframe model.



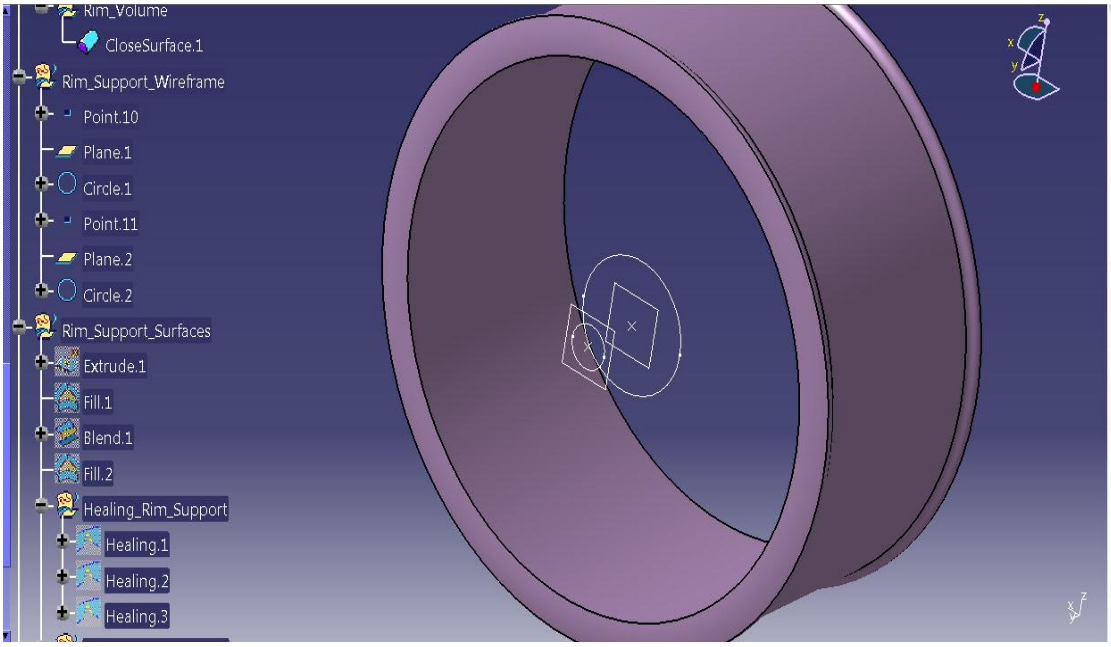
Picture 2.13.2: Creation of rim's centre line.



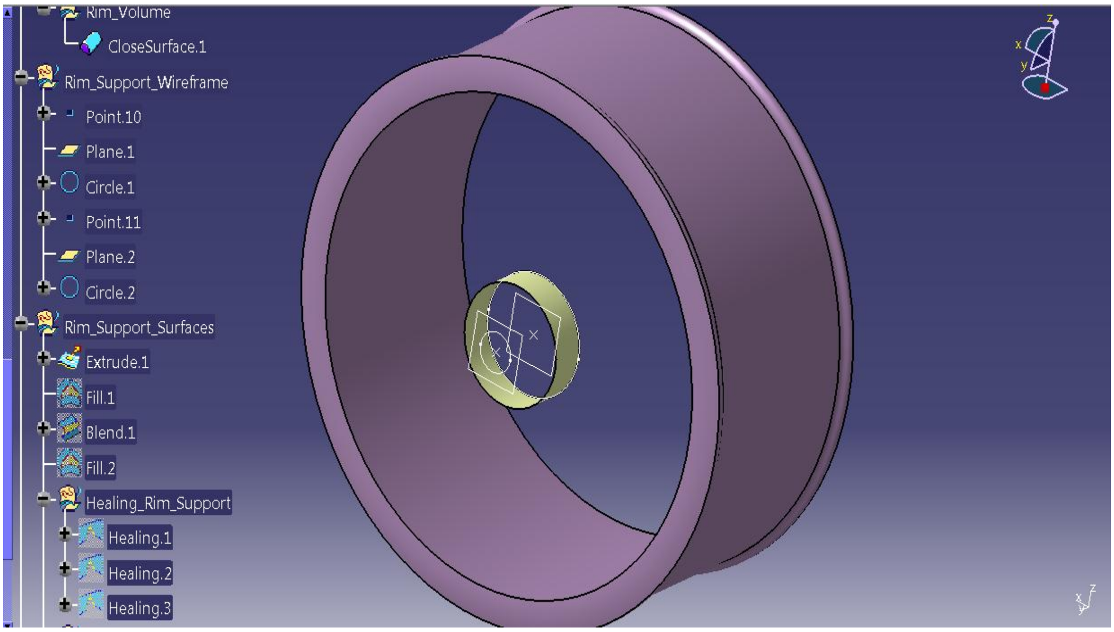
Picture 2.13.3: Creation of rim’s peripheral surface.



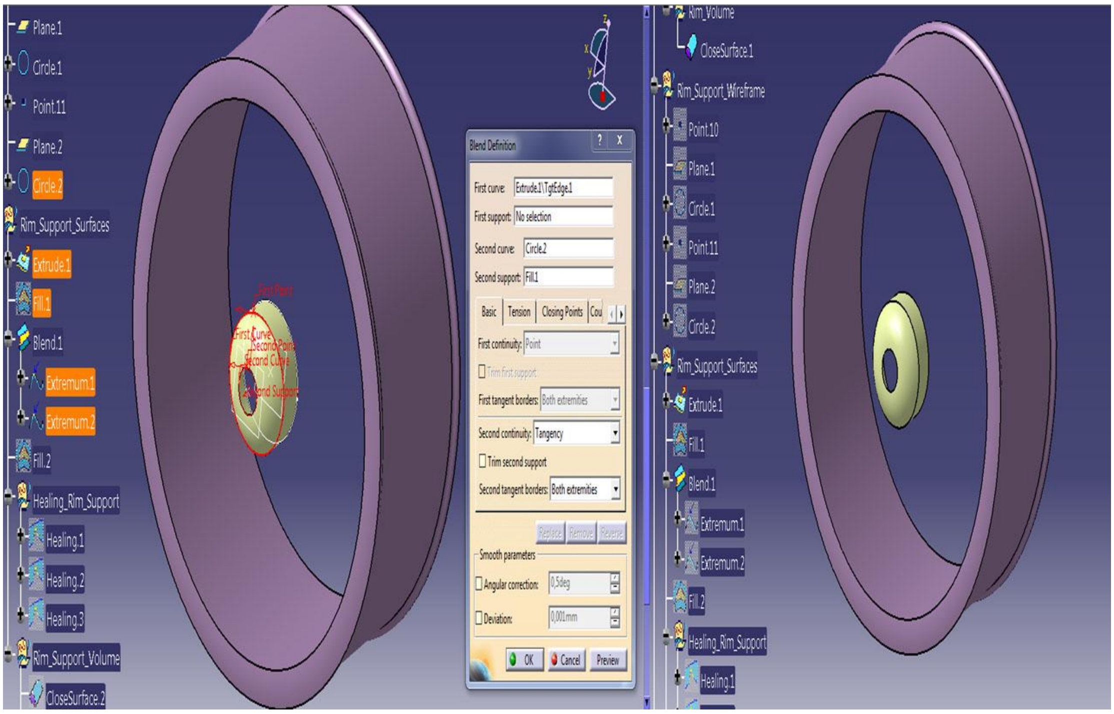
Picture 2.13.4: Creation of rim’s solid volume.



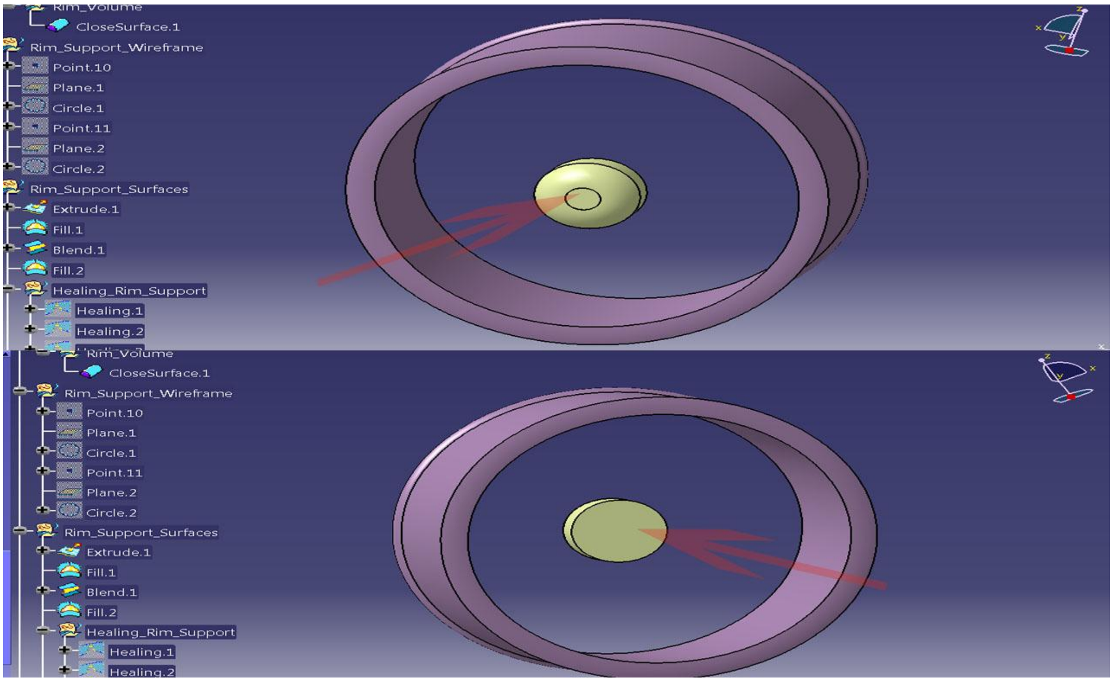
Picture 2.13.5: Wheel's central hub cap’s wireframe model.



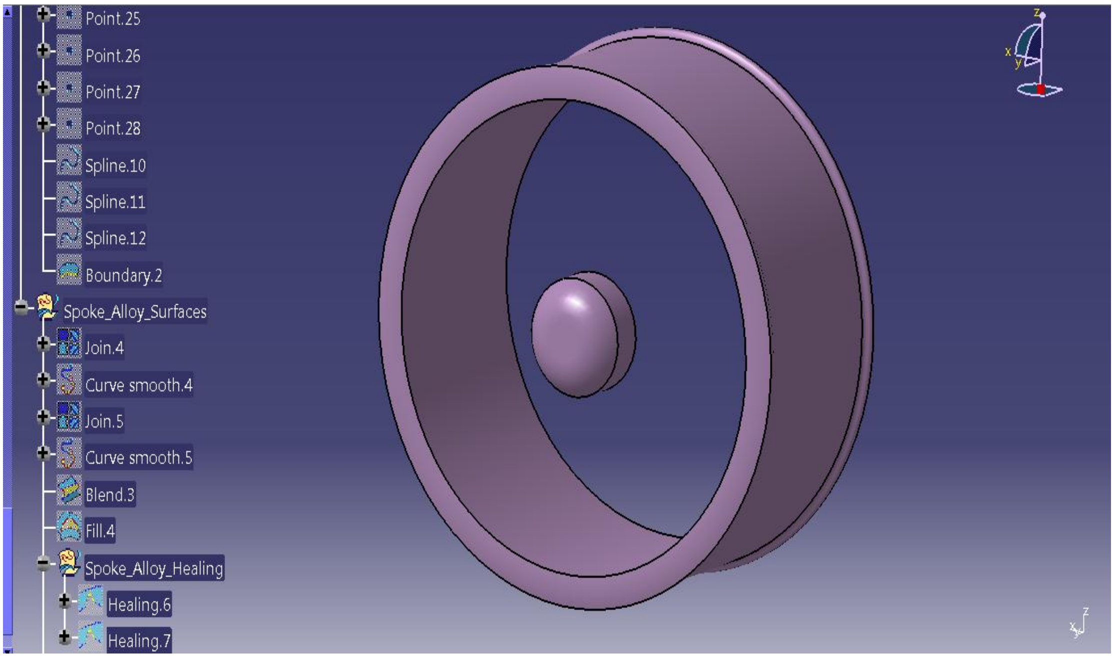
Picture 2.13.6: Creation of wheel's central hub cap’s peripheral surfaces.



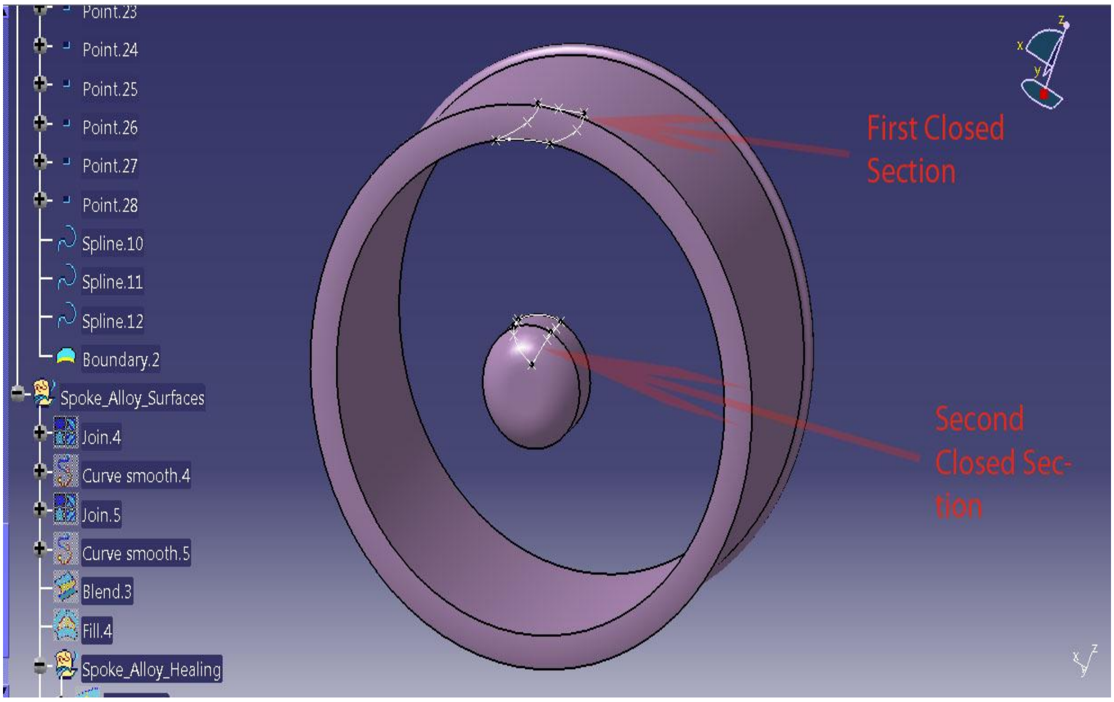
Picture 2.13.7: Creation of wheel central hub cap's peripheral surfaces.



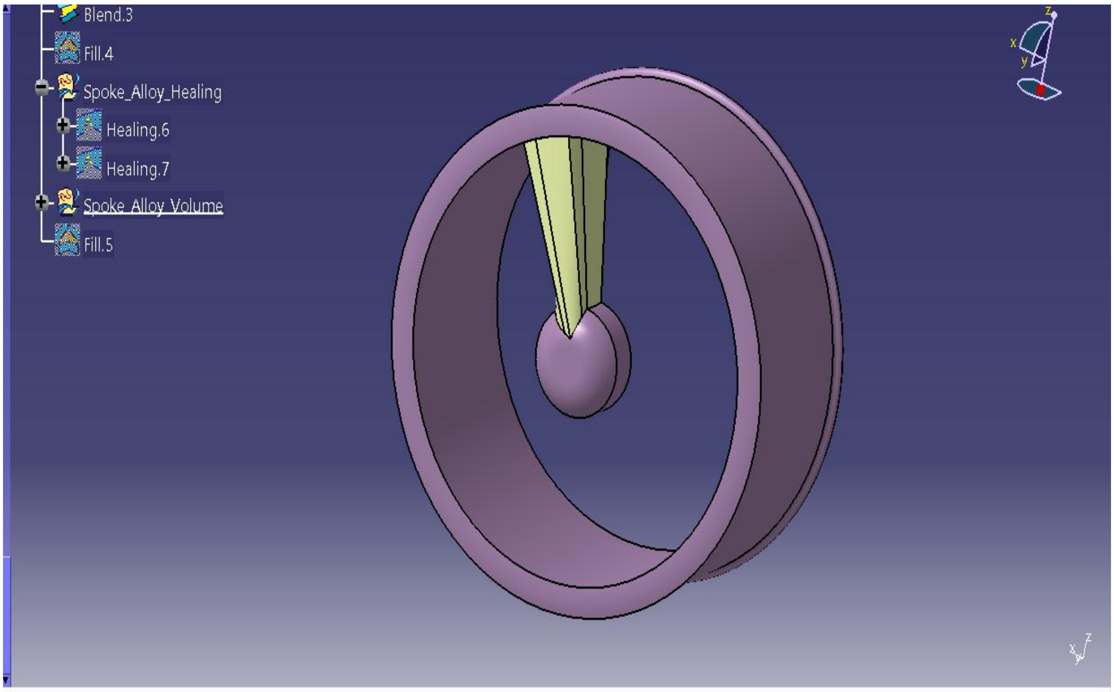
Picture 2.13.8: Creation of wheel central hub cap's peripheral surfaces.



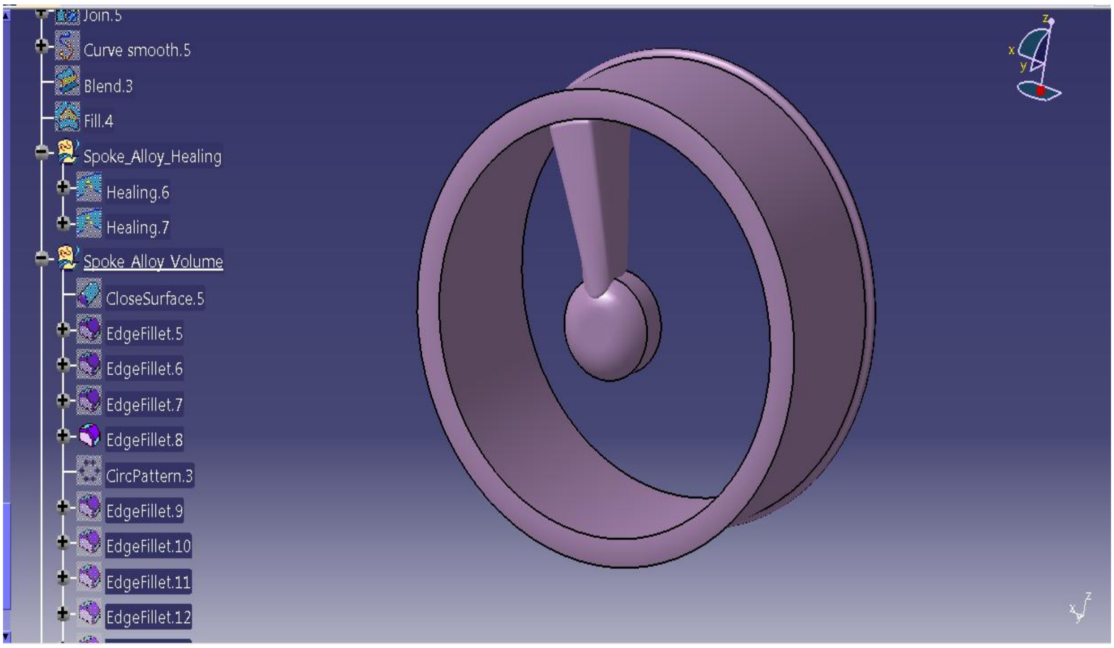
Picture 2.13.9: Creation of wheel central hub cap volume.



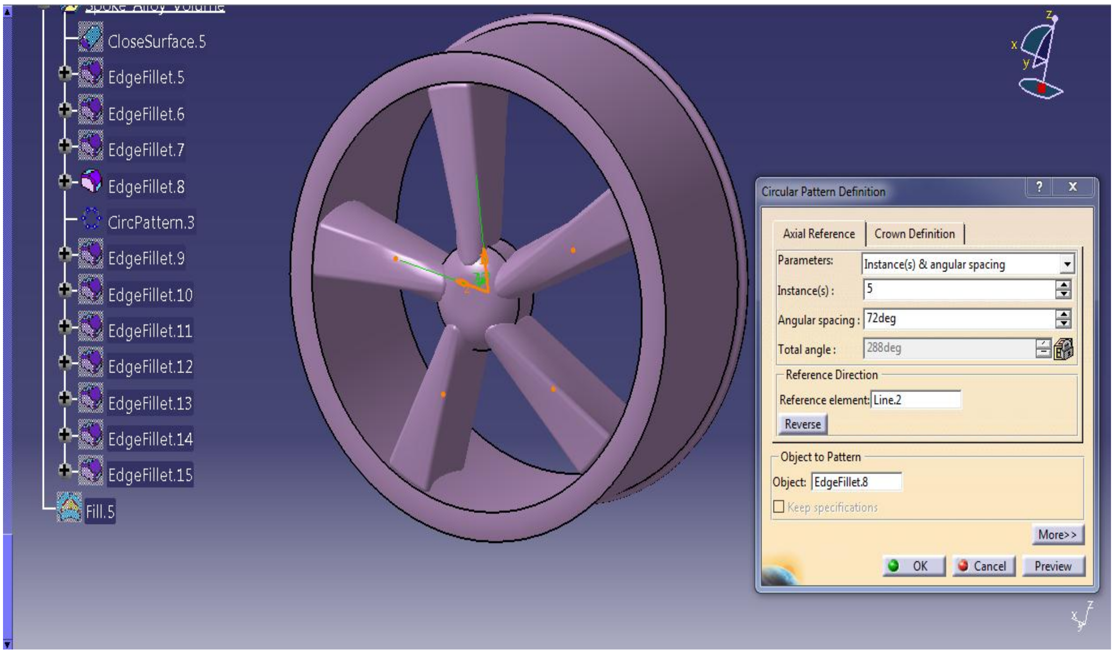
Picture 2.13.10: Spoke's wireframe model.



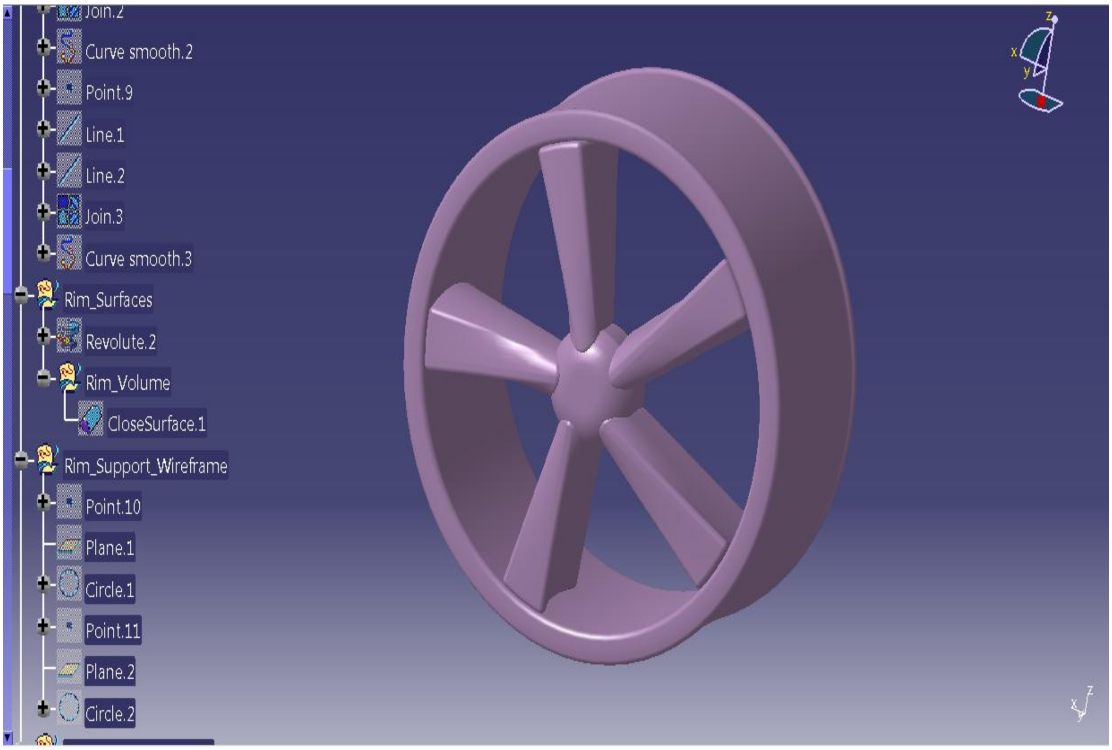
Picture 2.13.11: Creation of spoke's blended surface.



Picture 2.13.12: Creation of spoke's final volume.



Picture 2.13.13: Creation of rim's total number of spokes.



Picture 2.13.14: Rim's final volume.

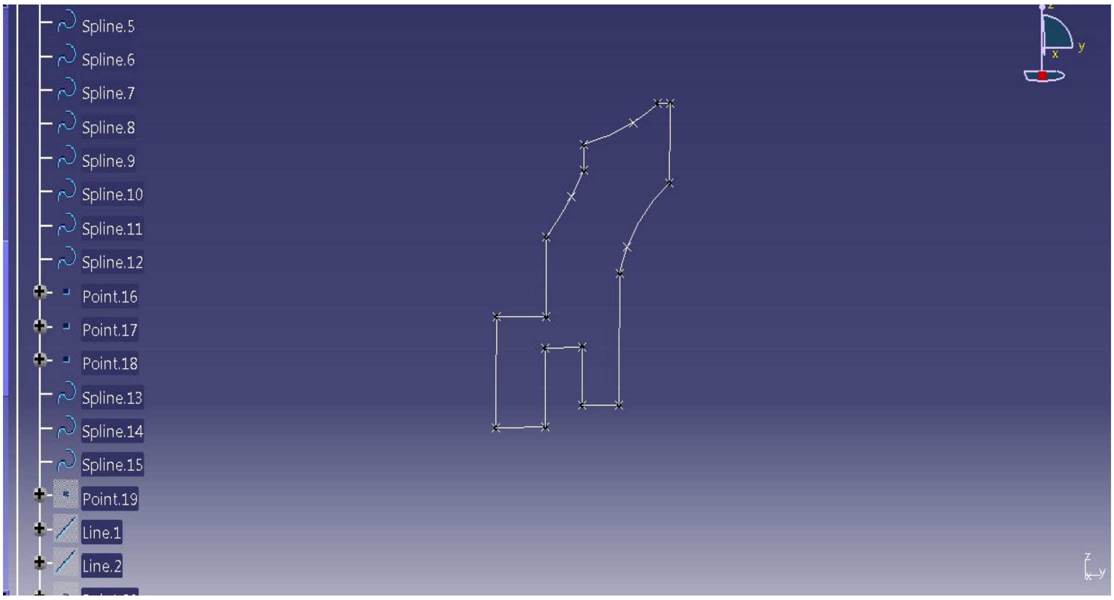
2.14 DEFINITION OF THE SEAT'S UPPER SLIDE PART

The support part was one of the two module parts of the seat's upper slide unit. This part was not going to be included inside the cabin's aerodynamical analysis. Its definition was implemented primarily for completion purposes and its existence did not have any functional utility. As every independent part design, this module was designed as a parametric solid volume.

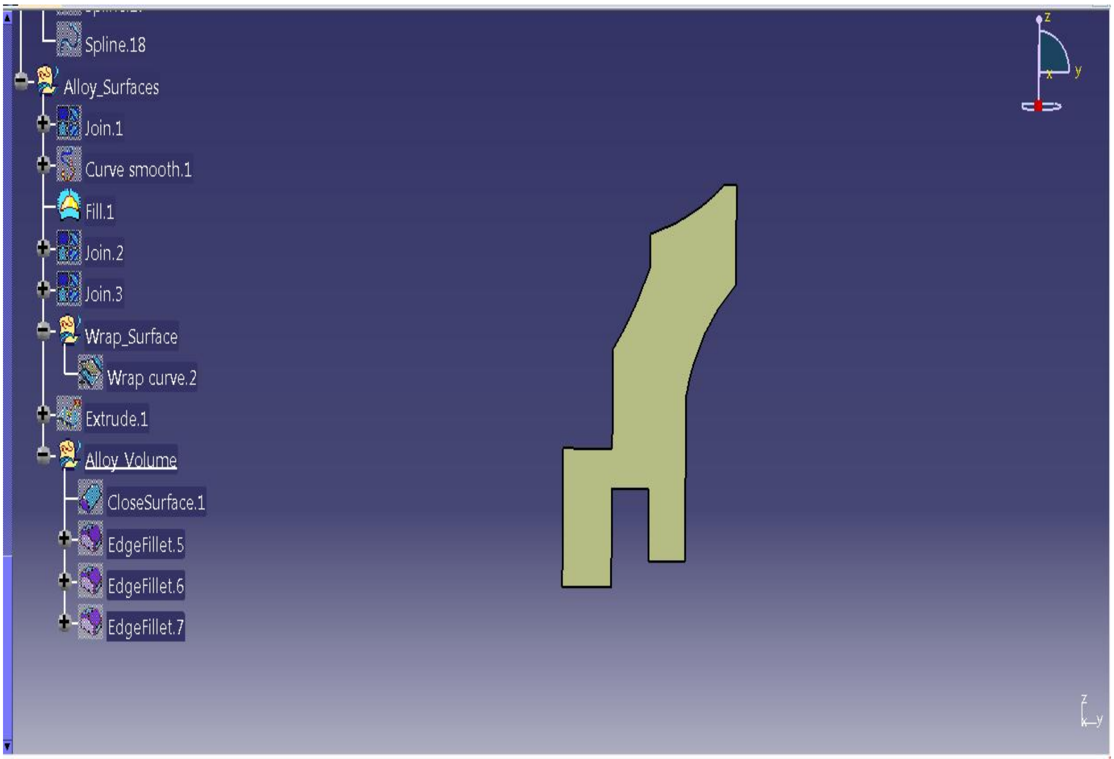
Firstly, the module's wireframe model was defined (Picture 2.14.1). This model presented a closed and planar section on the yz - plane. Its design was sophisticated and consisted of fifteen different B-Spline curves. Then, the fifteen separate B-Spline curves were unified using the joining operation command. Moreover, a *"curve-smooth"* operation was executed in order to ensure curvature of second degree on each section's point and better quality of the ensuing constructing surfaces. The wireframe's internal gap was filled by constructing a fill surface. As input inside the boundary option of the command window the section's contour was imported (Picture 2.14.2).

The next task was the deformation of the planar surface in order to form a solid volume that resembled an actual one as much as possible. The planar surface deformation was accomplished by using the "*wrap curve*" operation. As mentioned at several parts before, in order for this operation to be executed it is needed for at least two wireframe sections to be designed. The wireframe models that were designed for the command's implementation were a single linear B-Spline and an additional curvilinear B-Spline. These two Spline curves were planar and were on a plane parallel to the xz plane (Picture 2.14.3). The deformed surface was constructed with the execution of the "*wrap surface*" command. As inputs in the command's window for the ''surface to be deformed'' option the filled surface was inserted, while as reference and target surface for the deform specification the linear and curvilinear B-Splines were imported respectively (Picture 2.14.4). By using the deformed surface as reference, an extruded surface was formed. The direction of the extrusion was perpendicular to the yz – plane towards the negative direction of the x axis. The total length of the extrusion was 80 mm, as shown in Picture 2.14.5 below. The final solid volume arose with the use of "*close surface*" volume creation command. As input object the previously designed surface was selected (Picture 2.14.6). Finally, for aesthetic purposes an *"edge fillet"* command was executed on every edge of the solid volume. The radius of curvature for each edge filleting was determined as 5 mm (Picture 2.14.7).

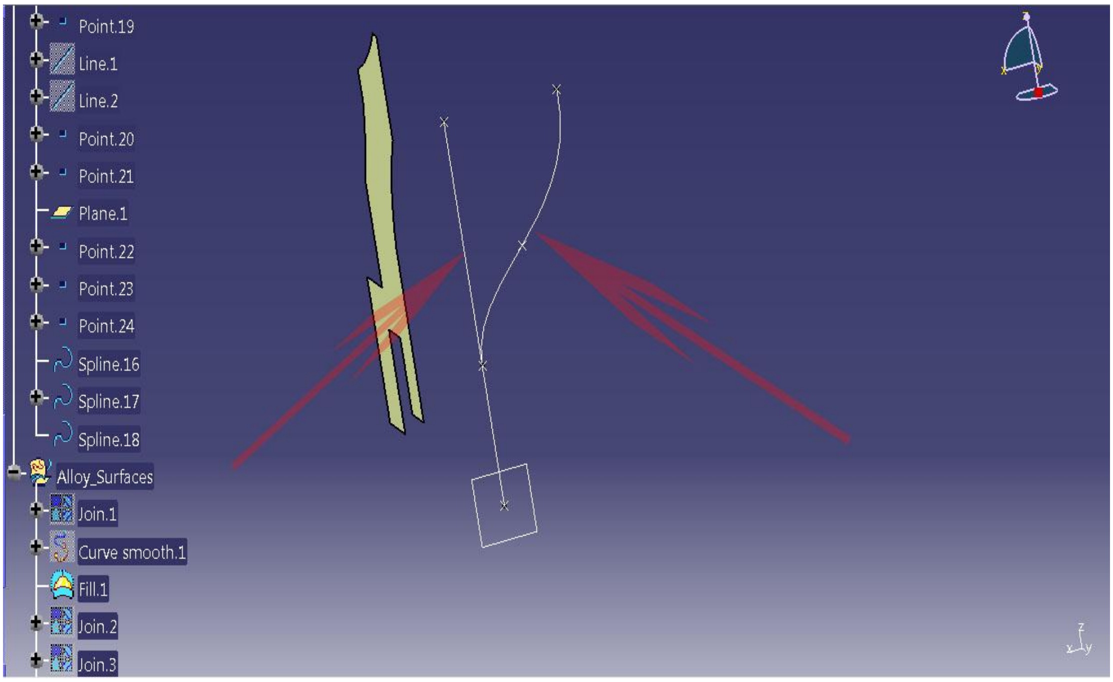
The proper dimension assessment was a trial and error procedure during the seat's assembly. Its methodological application was executed in exactly the same way as with the steering wheel's part.



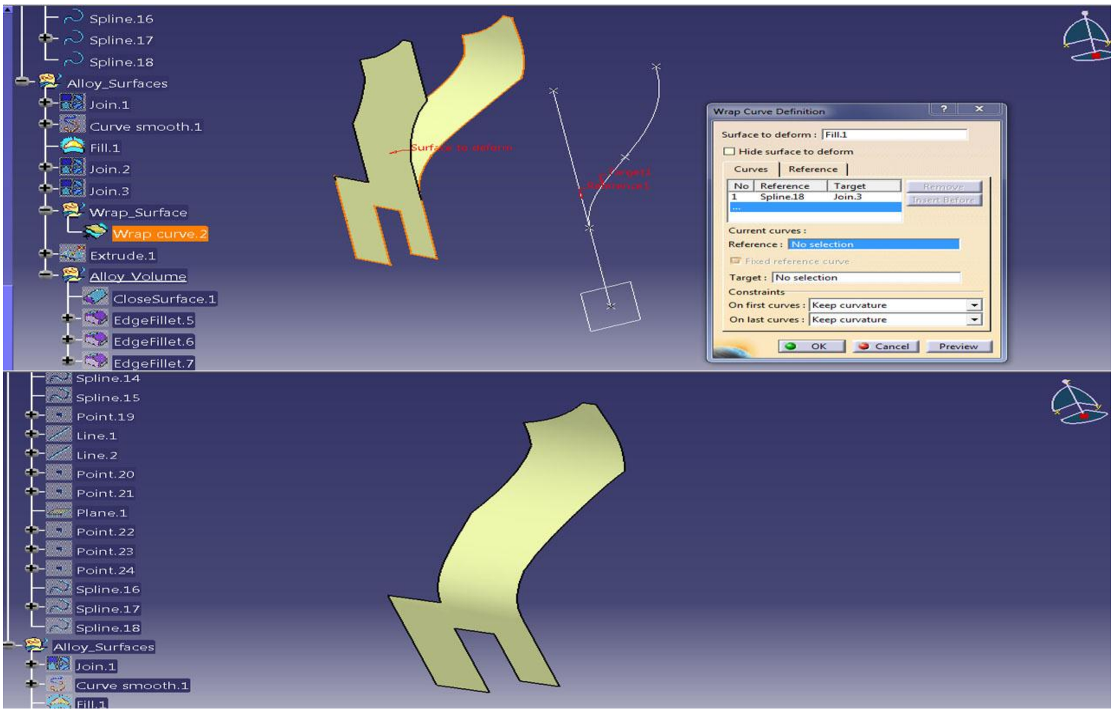
Picture 2.14.1: Wireframe model of slide's upper part.



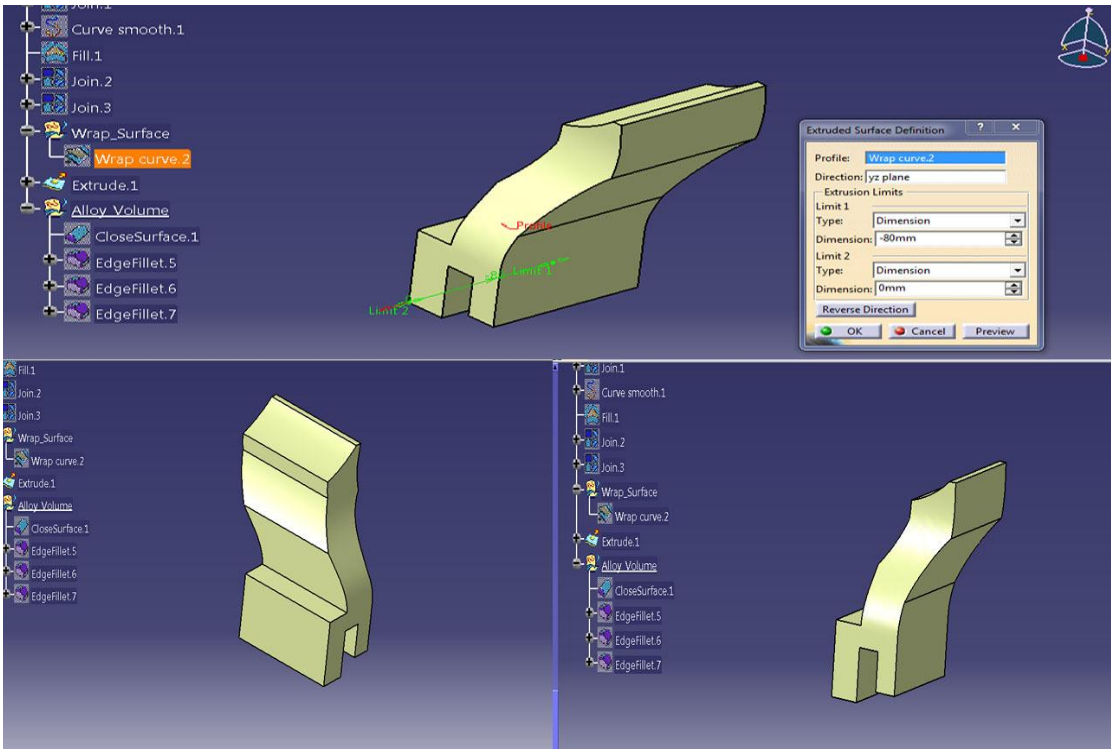
Picture 2.14.2: Creation of upper slide's part first fill surface.



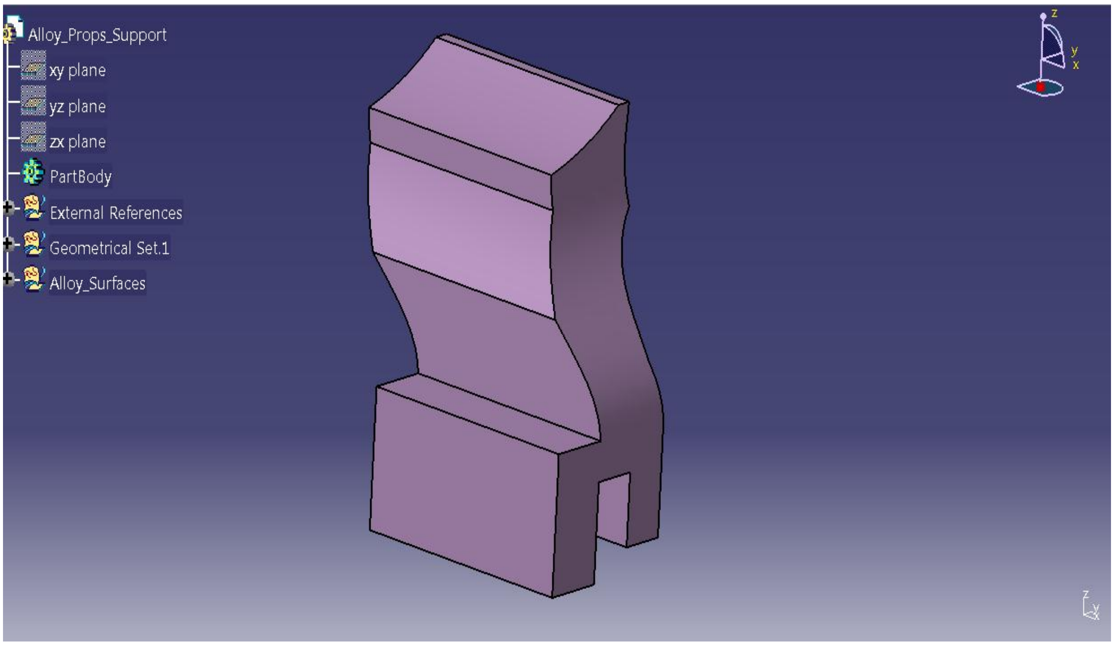
Picture 2.14.3: Representation of upper slide part initial and final shape.



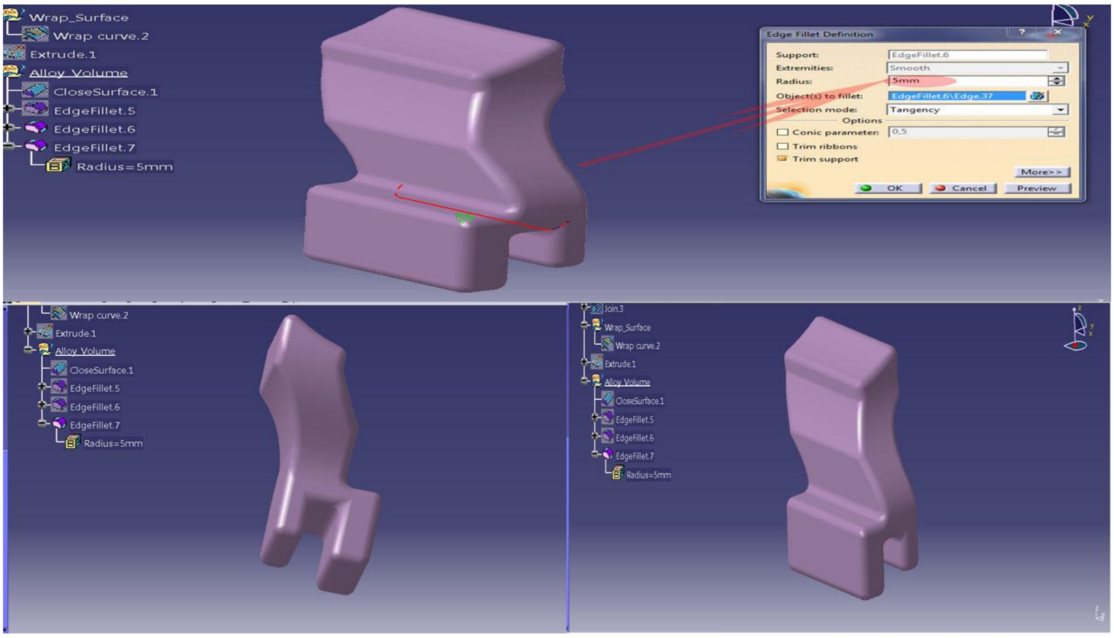
Picture 2.14.4: Deformation of upper slide's initial surface with the execution of "*wrap curve*" command.



Picture 2.14.5: Definition of upper slide's surface of extrusion.



Picture 2.14.6: Upper slide's initial volume.

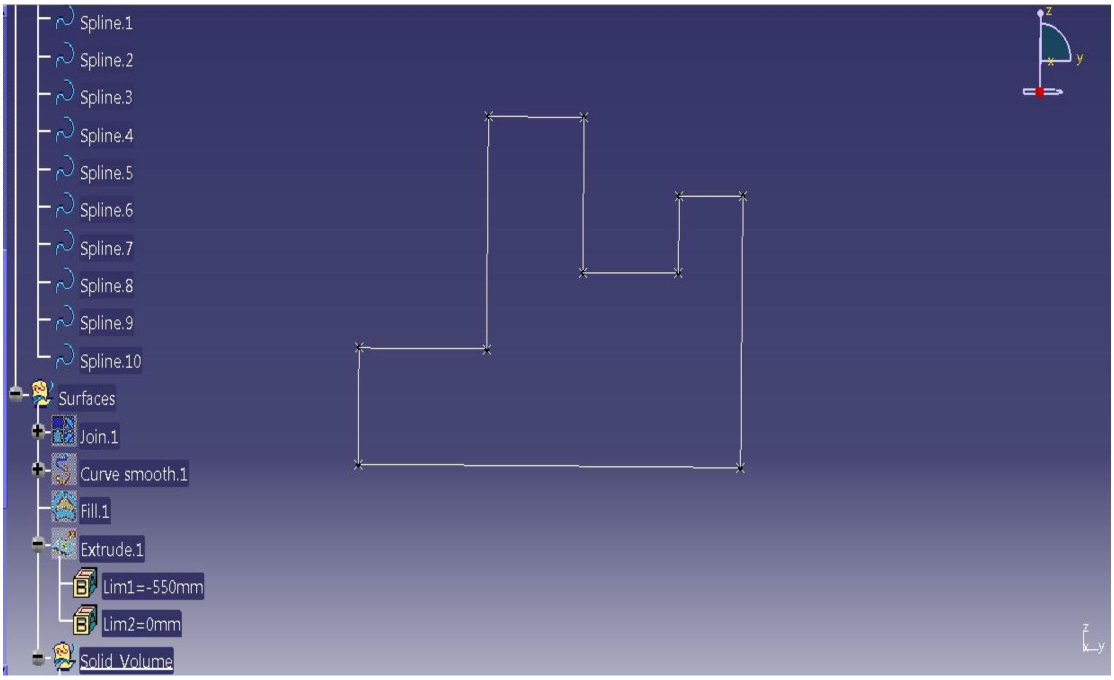


Picture 2.14.7: Upper slide's final volume.

2.15 DESIGN OF SEAT'S LOWER SLIDE PART

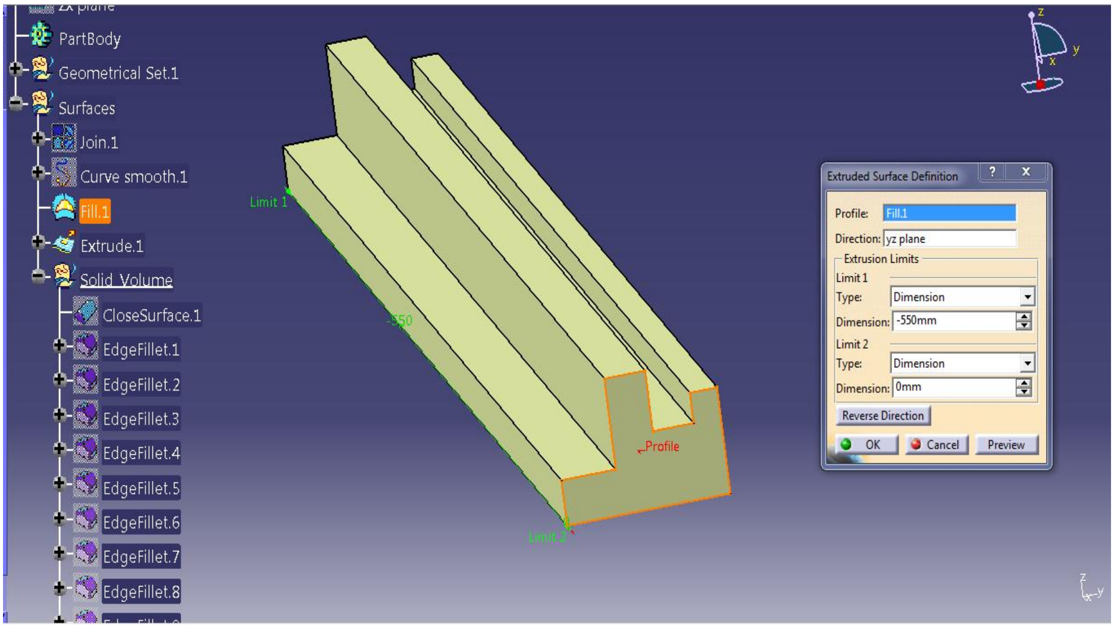
This part constitutes the second module of the lower slide's geometry. As with the upper slide part, this module is not part of the aerodynamical analysis of the vehicle's cabin. Furthermore, the actual dimensions numerical values were assessed with the same trial and error technique that was followed in the previous part.

For the construction of the 3-D solid part its wireframe model was first defined. This part's wireframe was planar and less sophisticated than the previous one. The model also formed a closed section that lied on the yz – plane and consisted of 10 linear Splines (Picture 2.15.1). The set of Splines included inside the section's wireframe, had to be turned into a single entity. The unification was realized with the use of "*join*" operation command. In order to improve the quality of the following surface model, a "*curve smooth*" operation was opted for on the previously united section. Then, the wireframe's interior section was covered with a fill surface. As input for the command's execution the existing close section was inserted (Picture 2.15.2). By using the previously produced fill surface as reference, an extruded surface model was created. The extrusion's direction was defined to be perpendicular towards the yz - plane. Its length was determined as 550 mm towards the negative direction of the x-axis (Picture 2.15.3). The part's initial solid volume was produced using the "*close surface*" solid volume command. As reference for the production of the solid volume, the previously created extruded surface was used (Picture 2.15.4). To complete the 3-D part's design, edge filleting operations were executed on the volume's edges. The curvature radius of each filleting on each edge was defined as 3 mm (Picture 2.15.5).

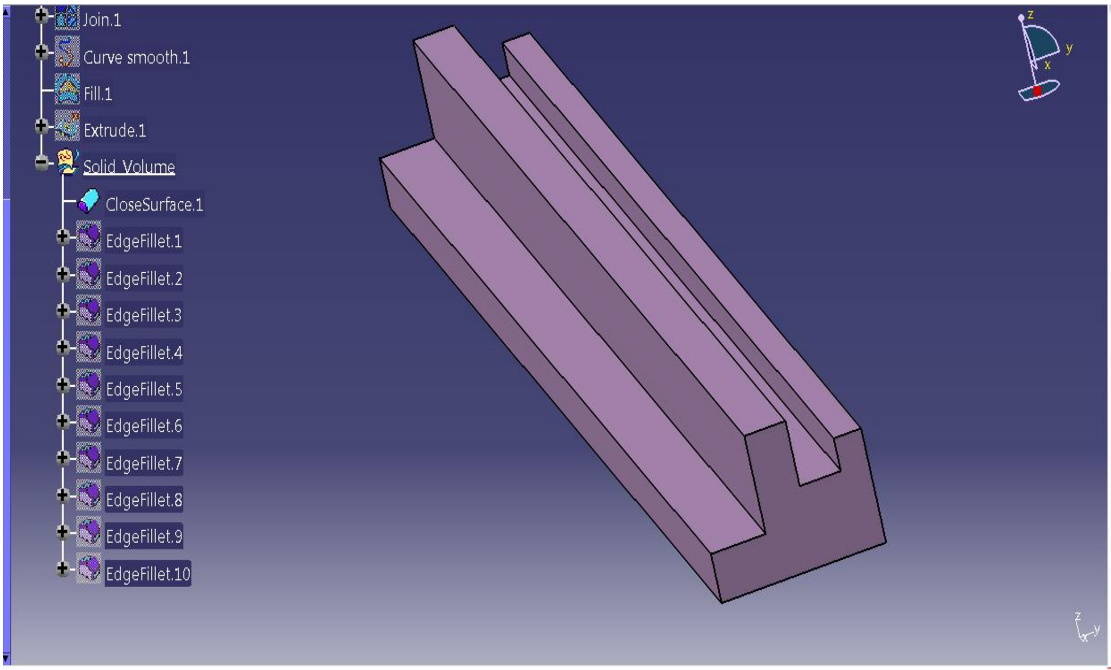


Picture 2.15.1: Definition of lower slide's wireframe model.

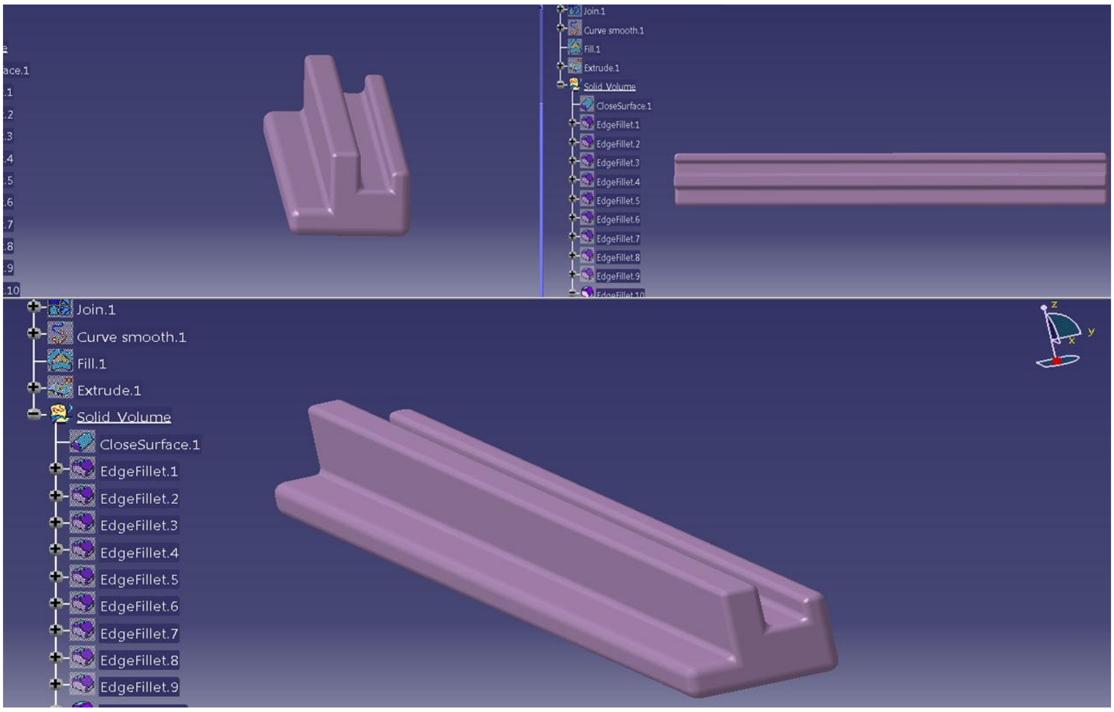
Picture 2.15.2: Creation of lower slide's fill surface.



Picture 2.15.3: Creation of lower slide's surface of extrusion.



Picture 2.15.4: Definition of lower slide’s initial volume.



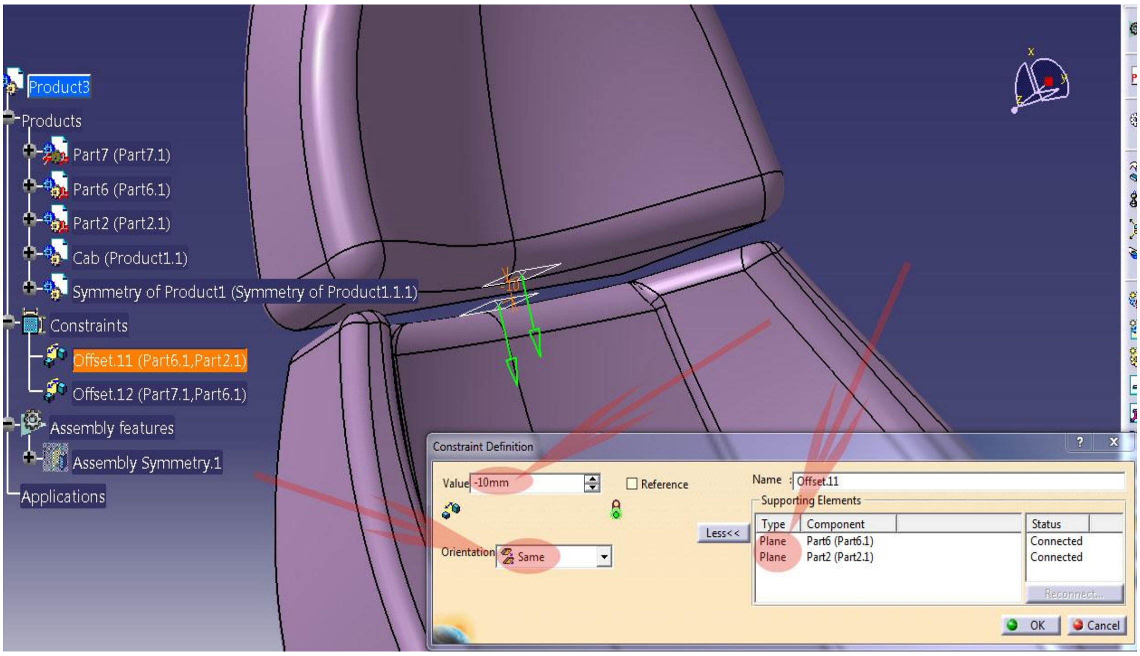
Picture 2.15.5: Definition of lower slide’s final solid volume.

3 FORMATION OF THE ASSEMBLIES AND SUBASSEMBLIES

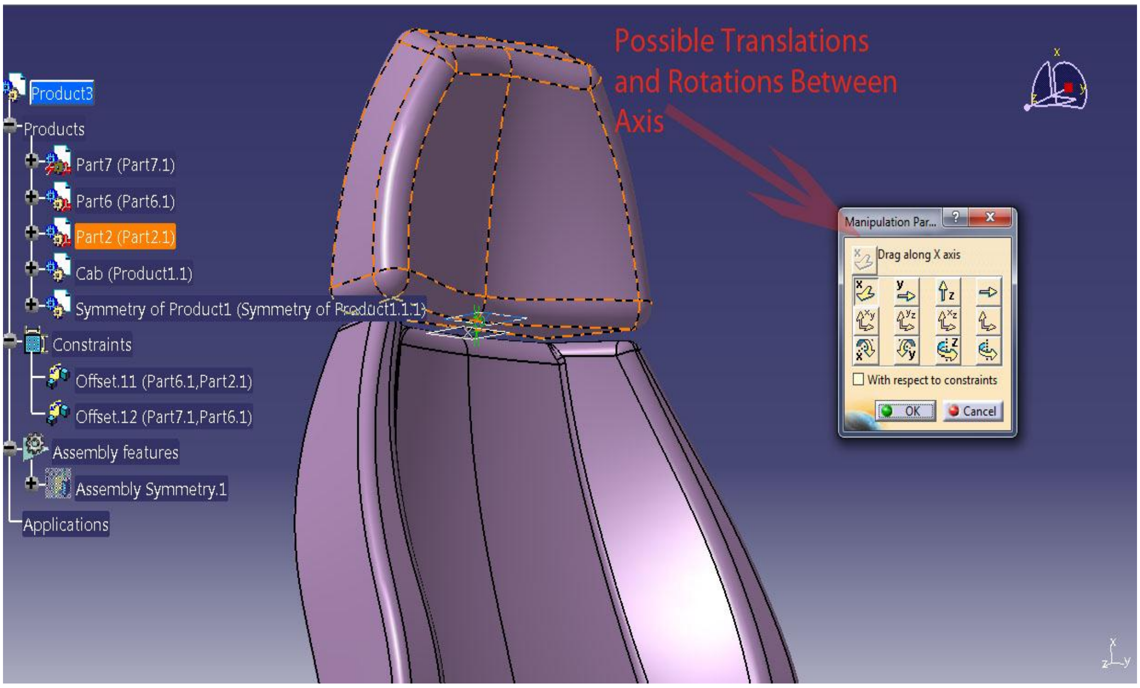
3.1 SEAT ASSEMBLY

The final 3-D solid of this part was the result of five parts in total. Three of these parts were independent while two of them were subassemblies. The independent parts were the backrest, headrest and underest seat components. The subassembly imported to this assembly was the seat's slides assembly.

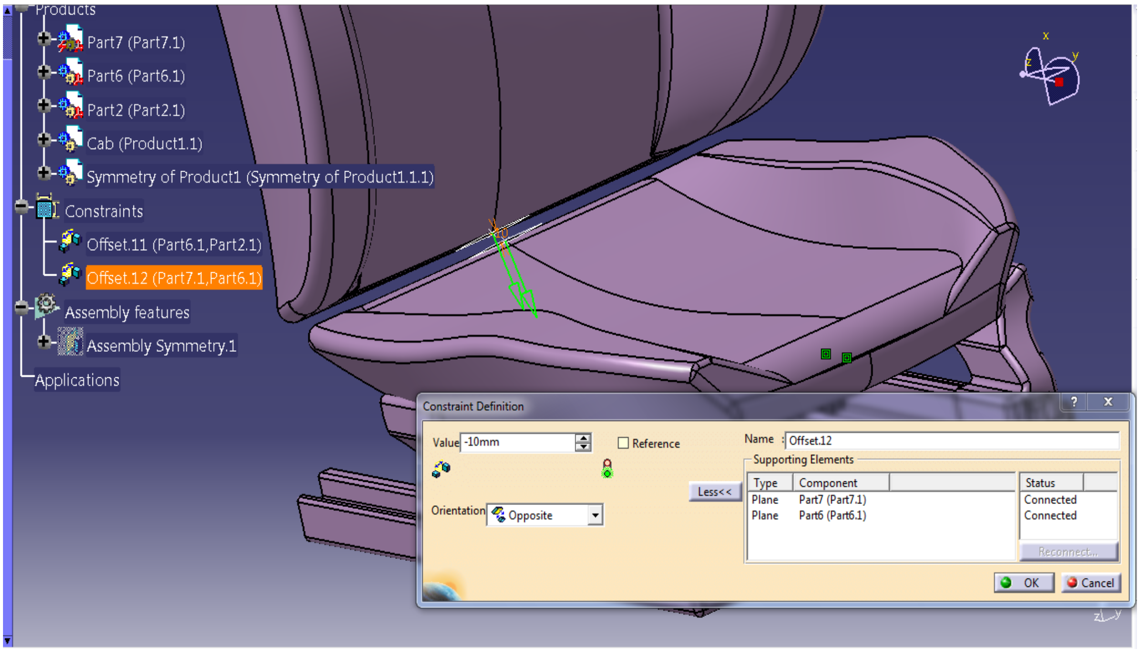
Firstly, the headrest and backrest solid parts were assembled together by imposing an offset constraint between two of each part's border planes respectively. The distance between the two planes was defined as 10 mm. The translation's orientation between the two planes was set as ''same'' inside the command window as shown in picture 3.1.1 below. This distance between the two solids was defined to receive a certain numerical value, because it was essential for the aerodynamics analysis. An additional important characteristic that had to be clarified at this point was that the exact position and orientation of the headrest part, with respect to the backrest part, was determined with the translation tools of the mechanical assembly platform. This certain tool that was used is called ''manipulation'' (Picture 3.1.2). With this technique, the backrest and underest solid parts were united. The two slide assemblies were connected with the rest assembly simply by using the manipulation tool. As shown at the model tree (Picture 3.1.4), two slide structures were used. The second slide geometry was derived by executing the ''*symmetry*'' operation after the first structure was set at its required position inside the final assembly (Picture 3.1.4). The symmetry operation was available inside the ''insert'' option which existed at the menu bar of the Catia software.



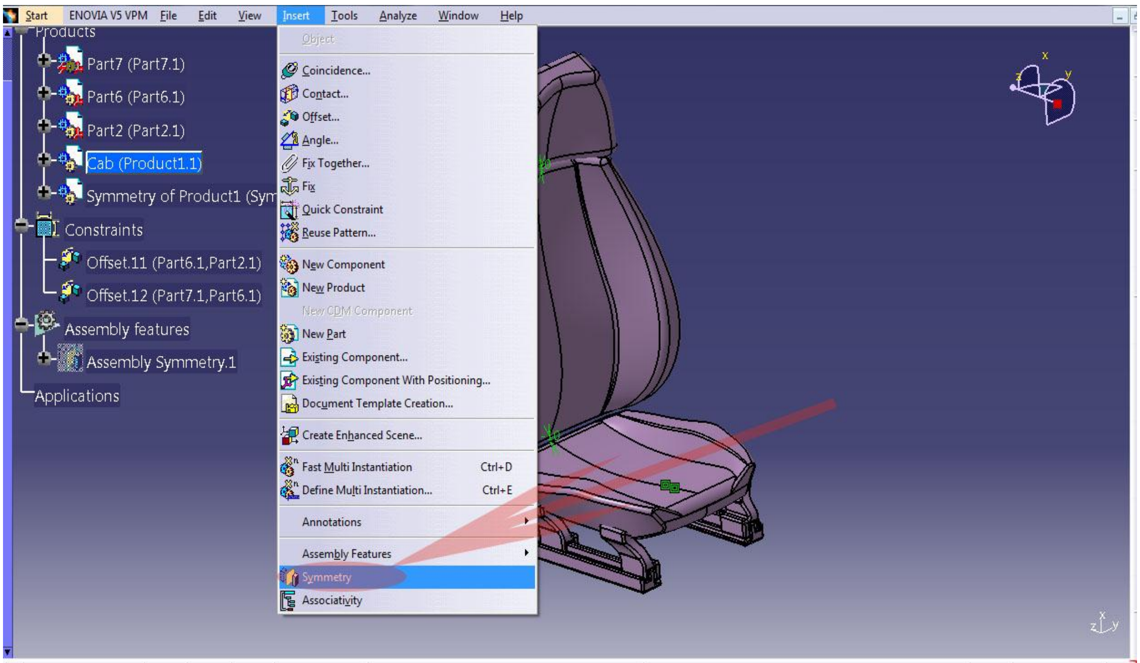
Picture 3.1.1: Creation of headrest and backrest offset constraint.



Picture 3.1.2: Use of manipulation tool.



Picture 3.1.3: Creation of the junction between the backrest and underest parts, by using an offset constraint.

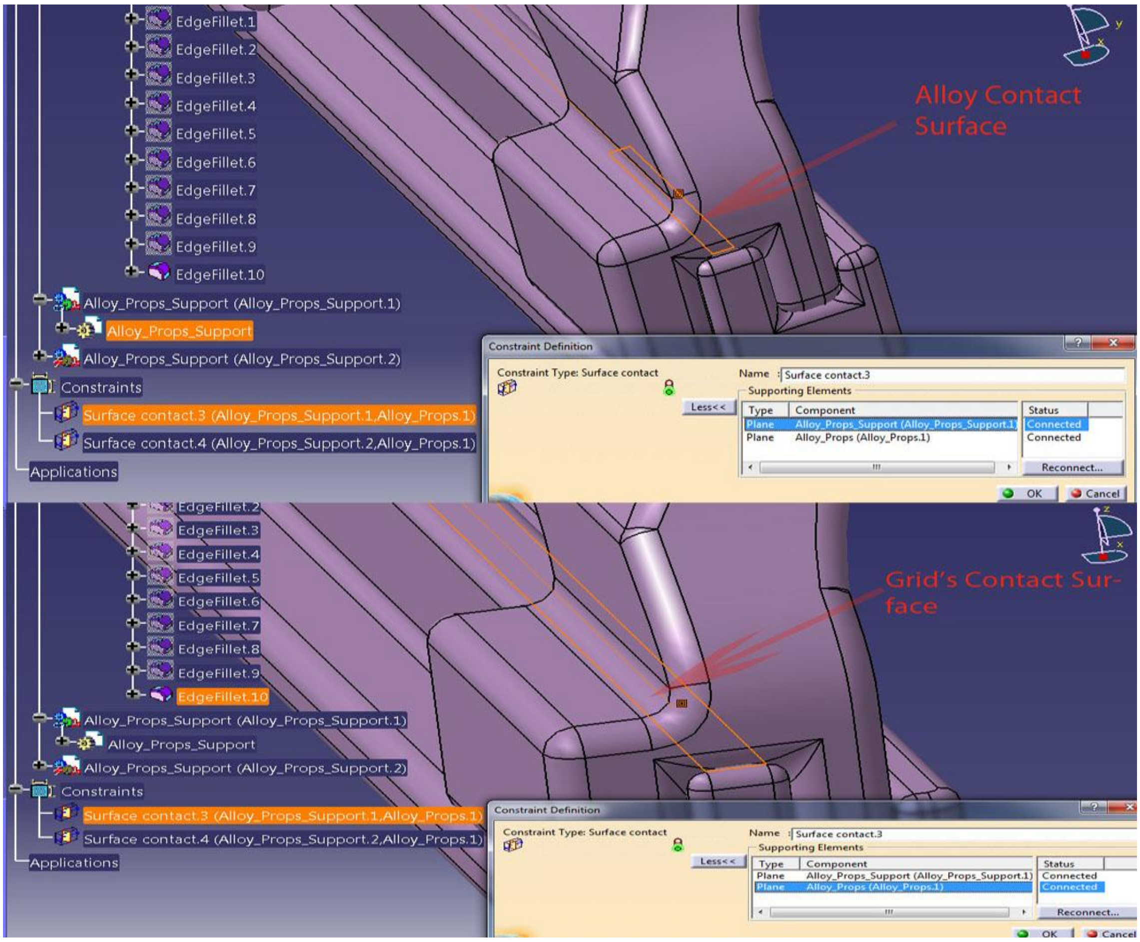


Picture 3.1.4: Use of the symmetry operation inside the assembly platform.

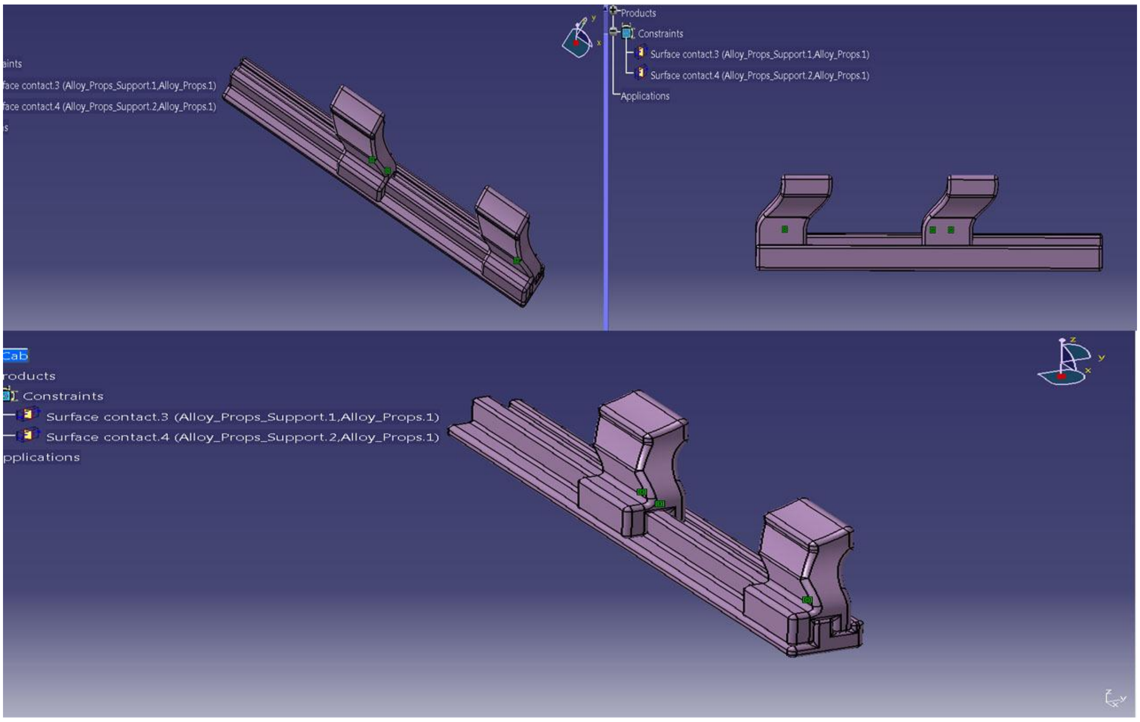
Picture 3.1.5: Definition of seat’s final assembly.

3.2 SEAT SLIDES ASSEMBLY

The complete slide structure, was the result of two independent solid parts. As it is shown in Picture 3.2.1 below, inside the platform two slide parts were introduced. The second upper slide solid was introduced into the design simply by being inserted, following the same methodology as with the other parts. The component parts were linked together by imposing two sequential contact constraints. The contact constraints were imposed between the upper and the lower slide solid respectively. The surfaces, which were defined to be tangent with the existence of the constraint, were the internal surface of lower slide and the internal surface of upper slide solid respectively.



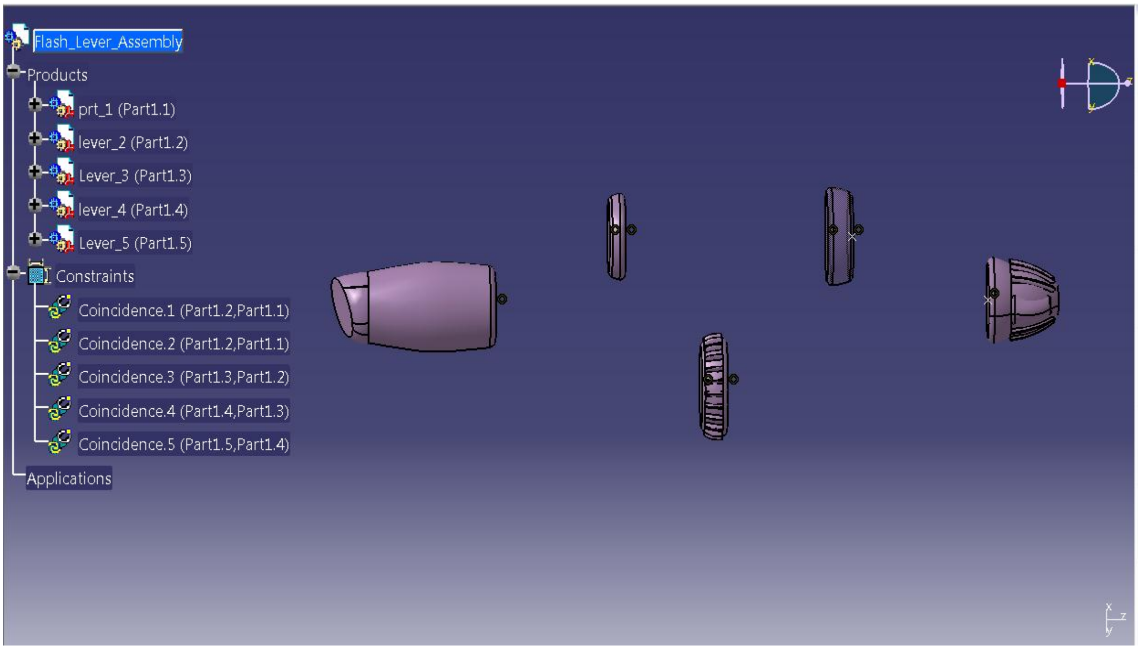
Picture 3.2.1: Definition of contact constraint between the slide parts.



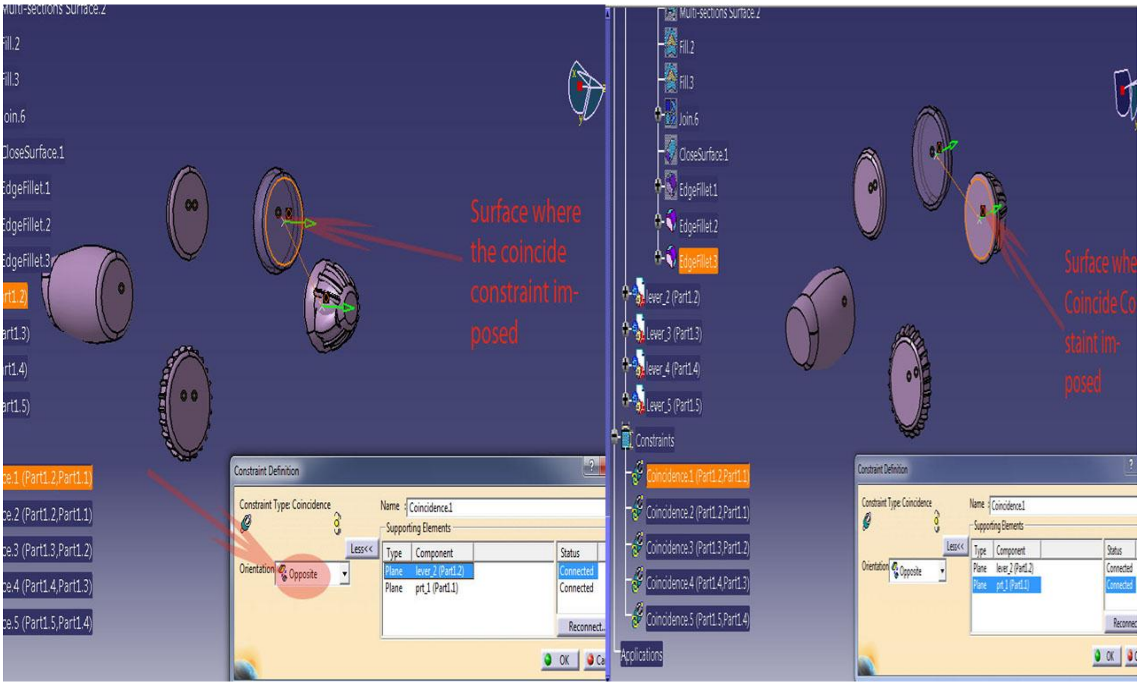
Picture 3.2.2: Creation of parts' complete assembly.

3.3 INDICATOR LIGHT LEVER ASSEMBLY

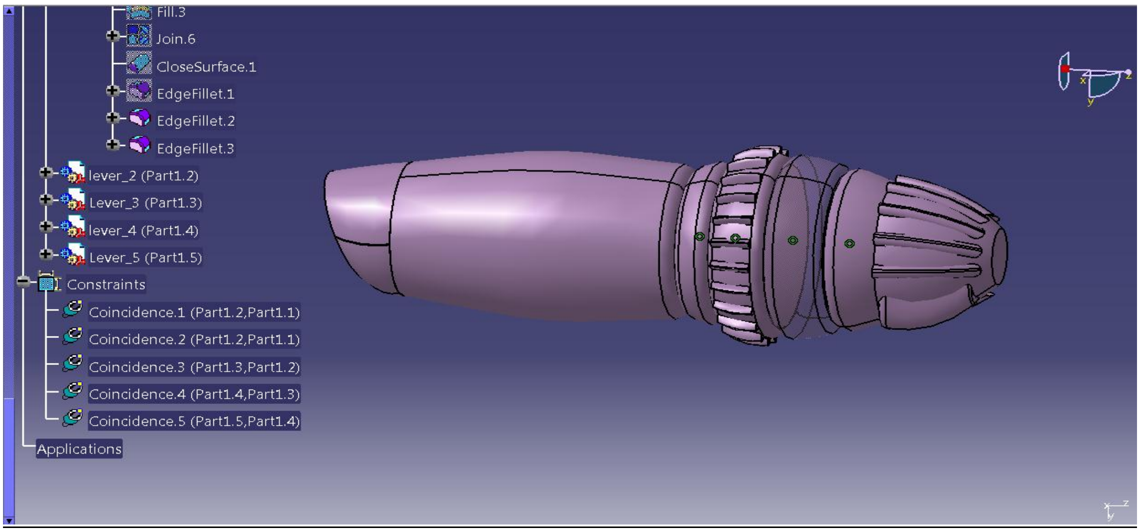
The complete indicator light lever structure was defined as an assembly of five independent solid parts. The five parts are depicted on the exploded view in Picture 3.3.1 below. All the parts were conjugated along with the same coincide constraint. This constraint, as every constraint contained in the platform, is designed to be imposed in a single pair of parts each time. The coincide constraint was imposed between the two subterminal surfaces of each solid part respectively. Inside the command window at the orientation option the opposite orientation was selected. The surfaces, where the coincide constraint was imposed, are depicted in Picture 3.3.2 below. By using the same technique, the remaining parts were linked together in order to form the final indicator light lever assembly part (Picture 3.3.3).



Picture 3.3.1: Indicator light lever exploded view.



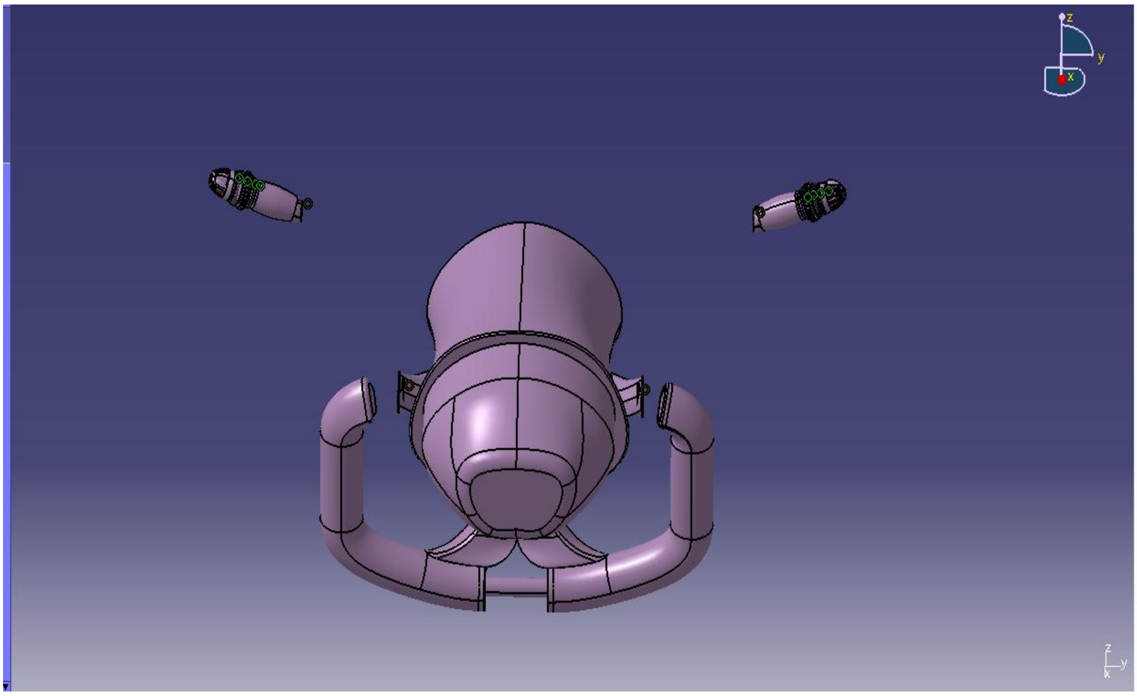
Picture 3.3.2: Definition of indicator light lever surfaces where the coincide constraint is imposed.



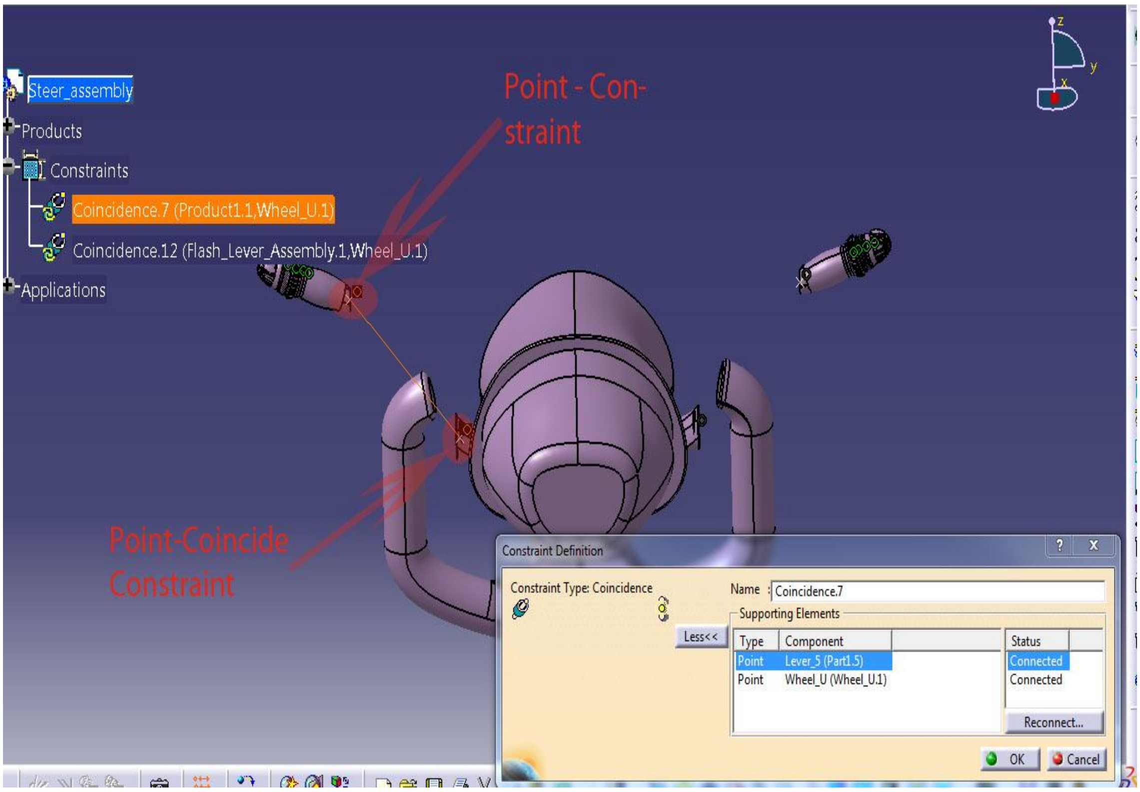
Picture 3.3.3: Indicator light lever final assembly.

3.4 STEERING WHEEL ASSEMBLY

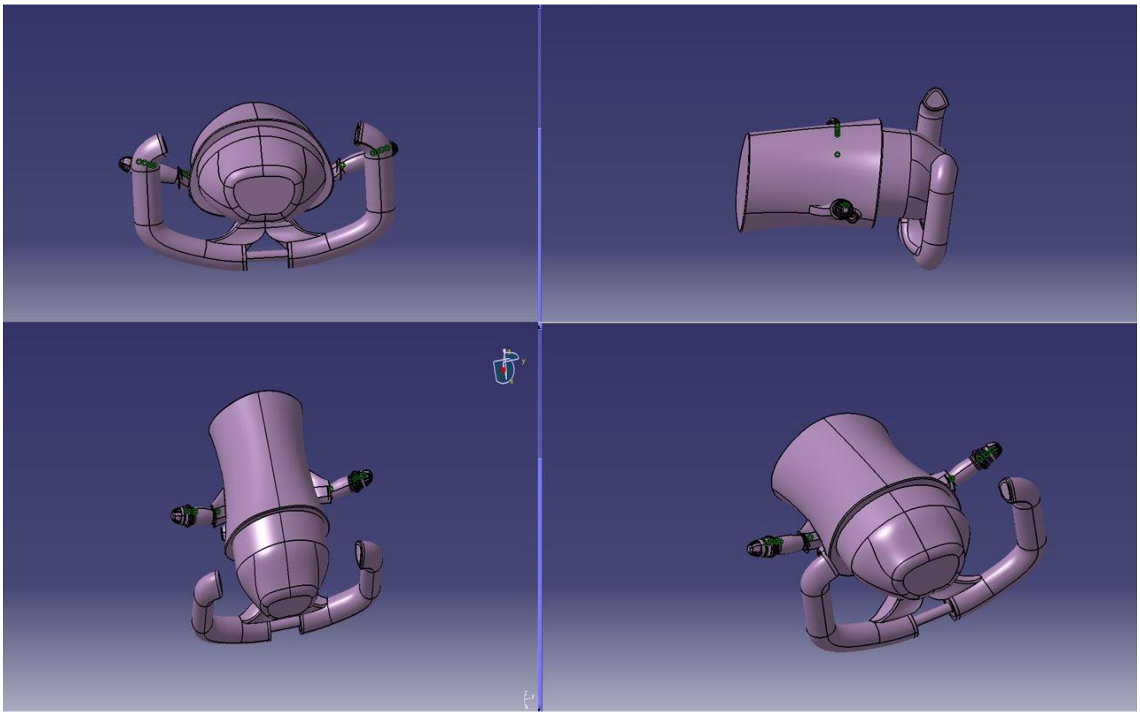
This assembled part consisted of two different independent parts. The first part was the main solid volume of the steering wheel while the second was the assembled part of the indicator light lever. As it is shown in Picture 3.4.1 below, the complete solid geometry of the steering wheel consisted of three independent entities, one wheel part and two indicator light levers. The second indicator light lever part was defined inside the assembly by using the *"symmetry"* operation. The two levers were linked to the steering wheel's primary solid volume by imposing two sequential coincide constraints (Picture 3.4.2). The use of this constraint was exactly the same as in the previous part. The main difference was that in this certain case the constraint was not imposed on surfaces between the two volumes but on extremum points belonging to each solid respectively. The final solid volume of the steering wheel assembly is depicted in Picture 3.4.3 below.



Picture 3.4.1: Exploded view of the steering wheel assembly.



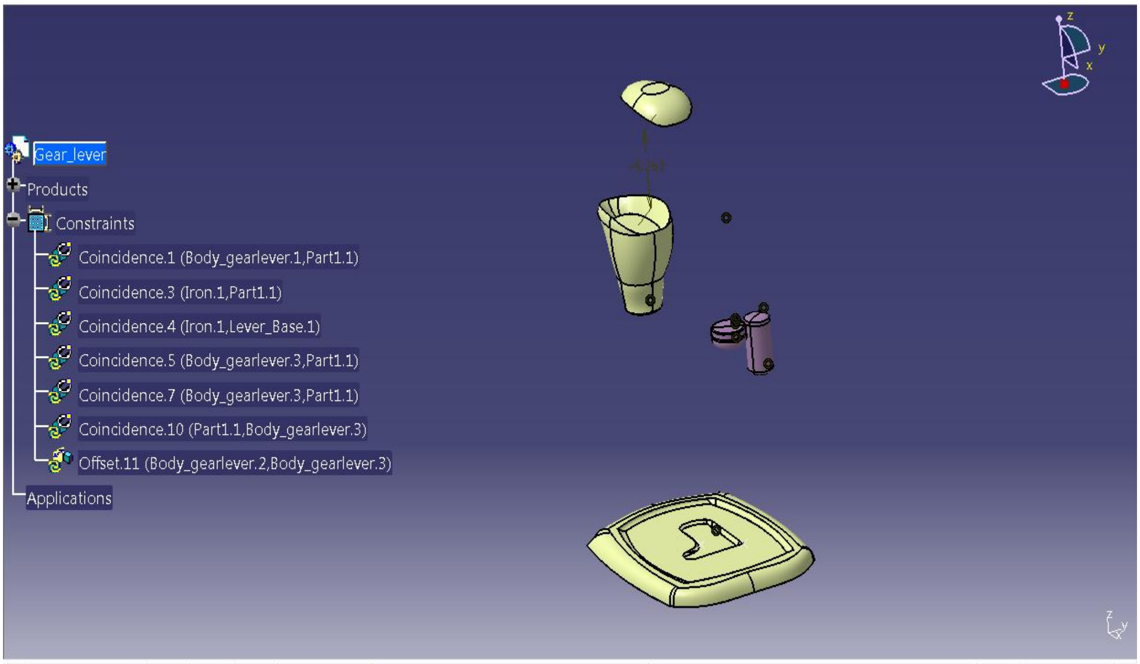
Picture 3.4.2: Definition of the coincide constraint between the steering wheel and indicator light lever parts.



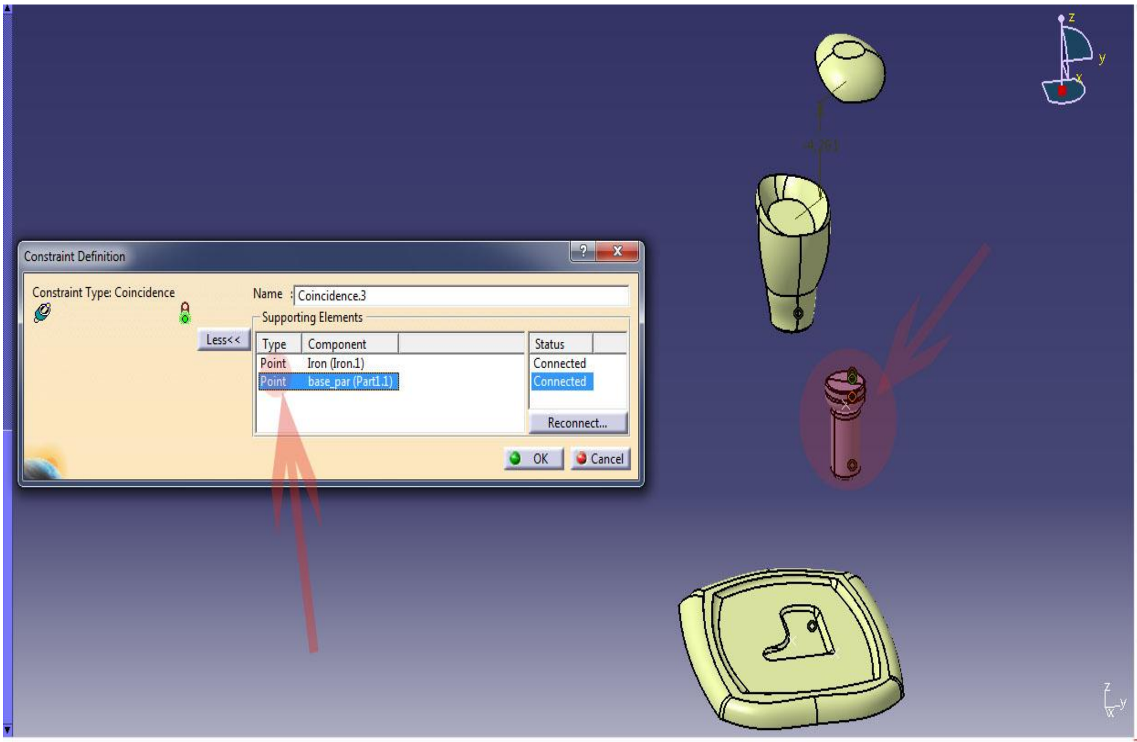
Picture 3.4.3: Creation of the final steering wheel assembly.

3.5 GEARSHIFT ASSEMBLY

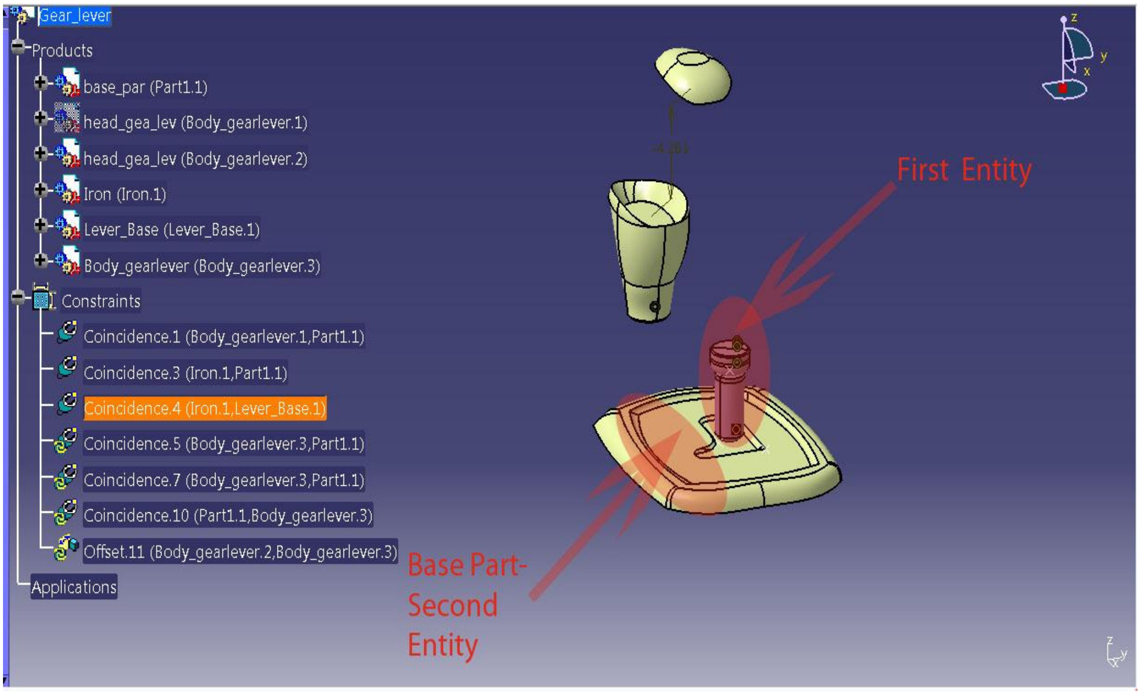
This assembly was more sophisticated than the assemblies of the previous parts. The gear lever's geometry was derived as the junction of five different and independent entities. The constitution parts are depicted in the assembly's exploded view in Picture 3.5.1 below. Moreover, inside the gear's assembly were imposed two different forms of constraints. The most frequent one was the coincide constraint, while an additional offset constraint was imposed to the remaining two entities. For specific entities, in order to be instated into their proper position in platform's space, it was needed to be further shifted. This additional translation of specific entities was implemented by using the ''*manipulation*'' operation command. The first two solid volumes of the assembly were joined together with the implementation of a coincide constraint (Picture 3.5.2). This constraint was imposed between two adjacent points that belonged to each volume respectively. Next in sequence, an additional coincide constraint was imposed. This second constraint joined the boot lifter's geometry with the previously connected solid geometries. Moreover, the constraint was implemented in the same way as the previous one. With the same methodology, the handle object was joined with the previously joined parts. The last object of the lever's part was positioned by using a combination of a coincide constraint and an additional translation of the part. The translation was implemented by using the manipulation tool.



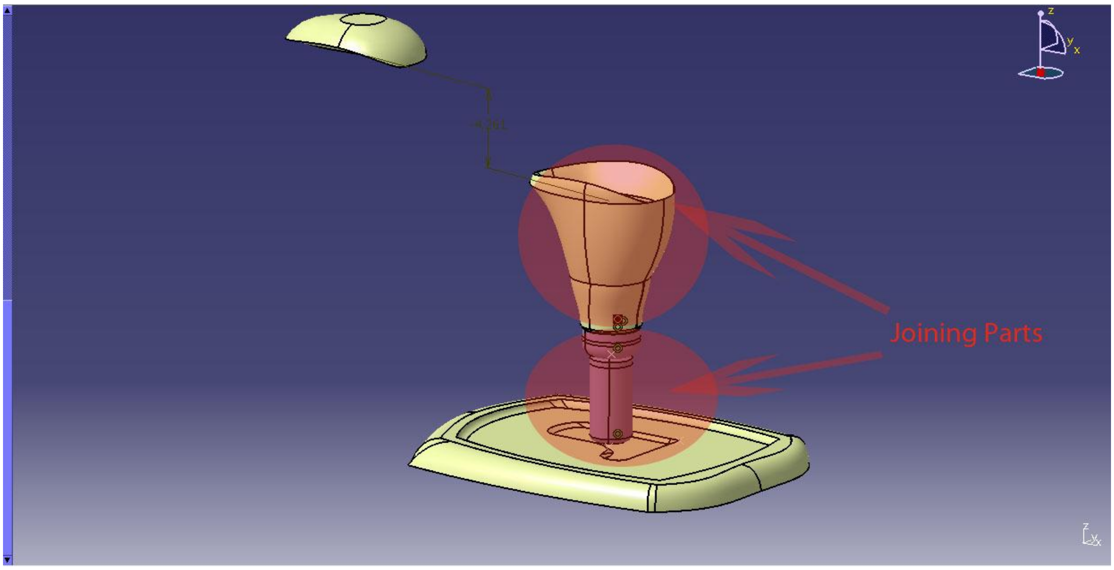
Picture 3.5.1: Exploded view of gearshift's assembly.



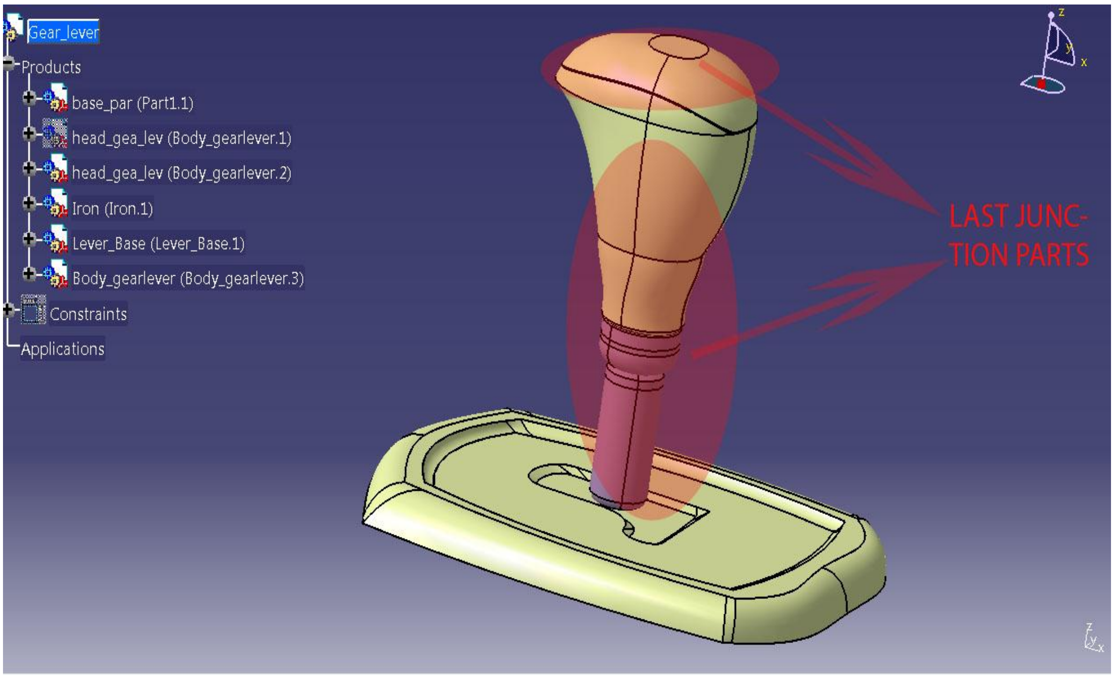
Picture 3.5.2: Combination of the gearshift's first two entities.



Picture 3.5.3: Entities of the gearshift's assembly participating in the second coincide constraint.



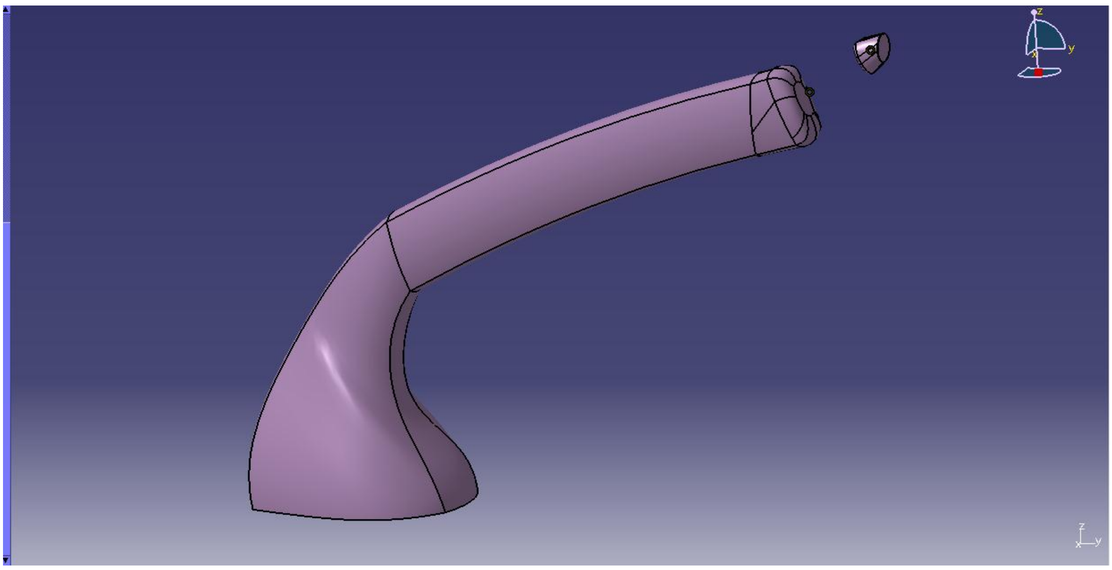
Picture 3.5.4: Entities of gearshift's assembly participating in the third coincide constraint.



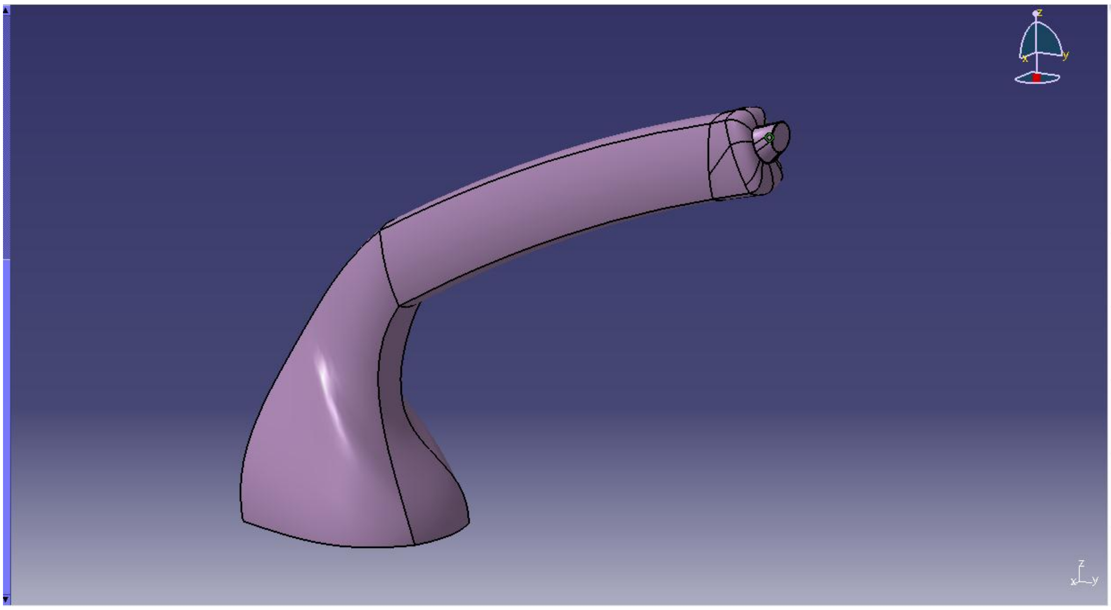
Picture 3.5.5: Creation of gearshift's final constraint.

3.6 HANDBRAKE LEVER ASSEMBLY

The handbrake's assembly was consisted of two independent parts. The lever's handle part and the knob. The two parts were joined together by imposing a coincide constraint (Pictures 3.6.1, 3.6.2).



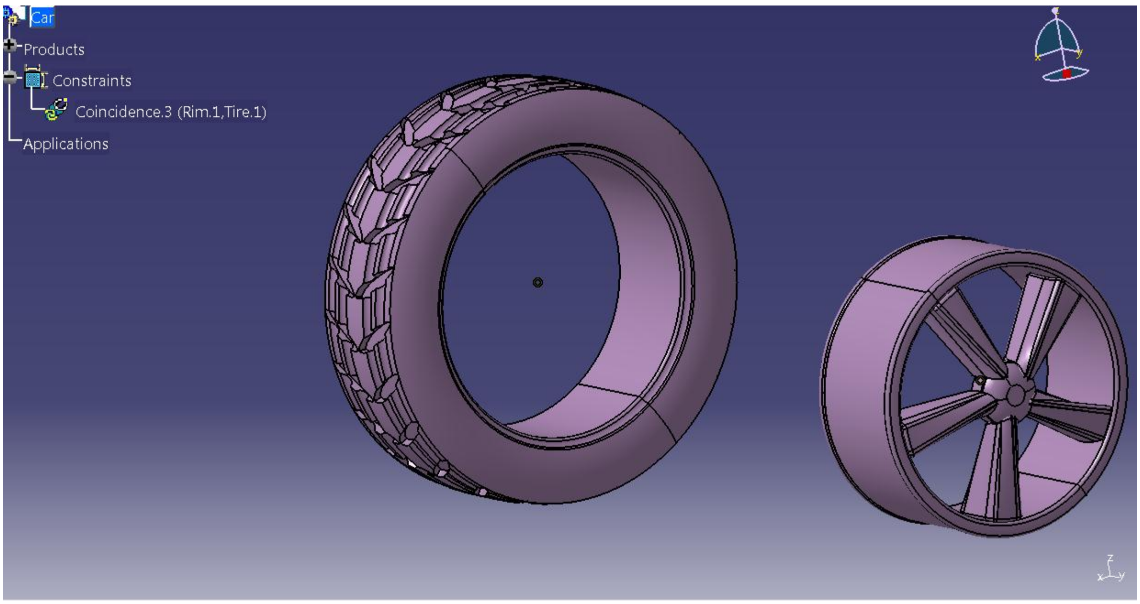
Picture 3.6.1: Exploded view of handbrake's lever.



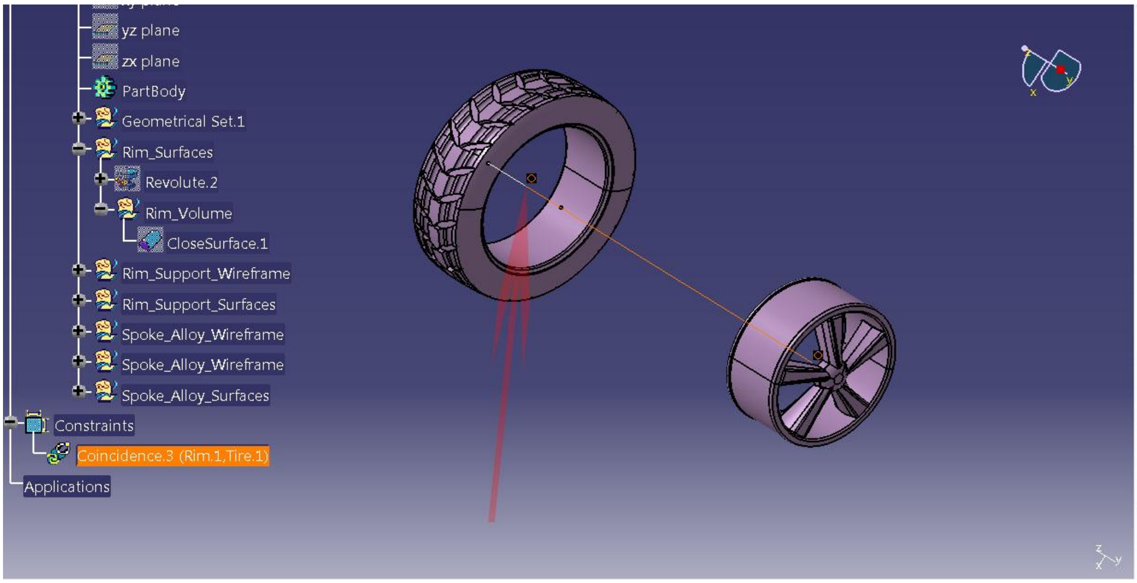
Picture 3.6.2: Handbrake's lever final assembled part.

3.7 CAR TIRE ASSEMBLY

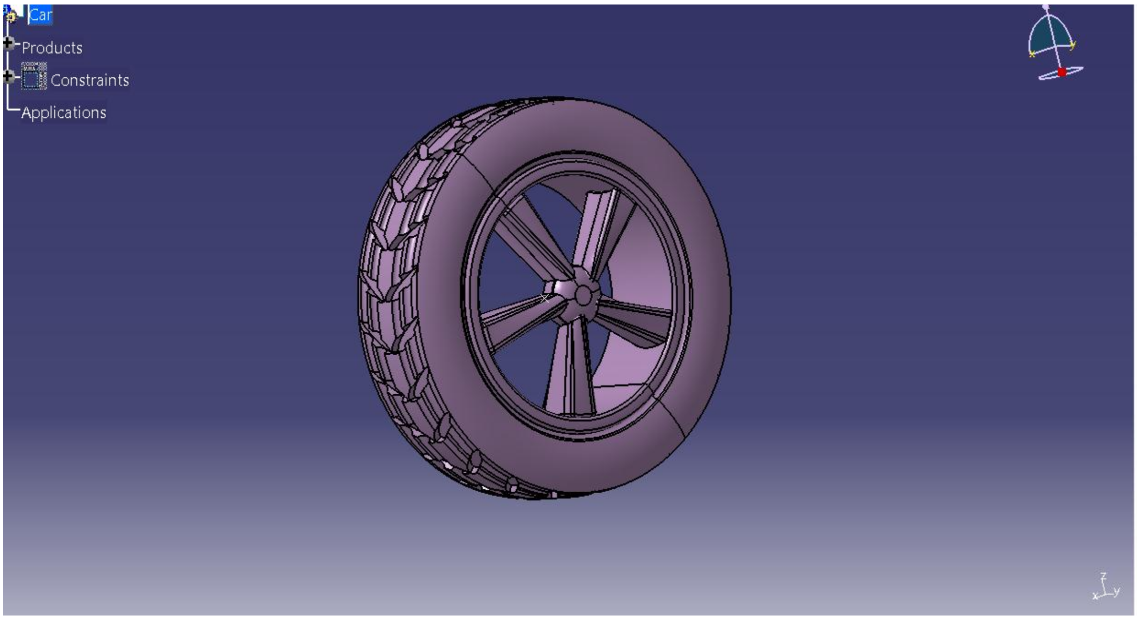
This assembly was also quite simple, just like the previous one. It was consisted of two different solid volumes, the tire and the rim, as it is shown at the exploded view in Picture 3.7.1 below. The two parts were united together with a coincide constraint (Picture 3.7.2). The main difference of this certain constraint with the previously implemented, was that it was imposed upon centerlines instead of surfaces and points (Picture 3.7.3).



Picture 3.7.1: Exploded view of tire's assembly.



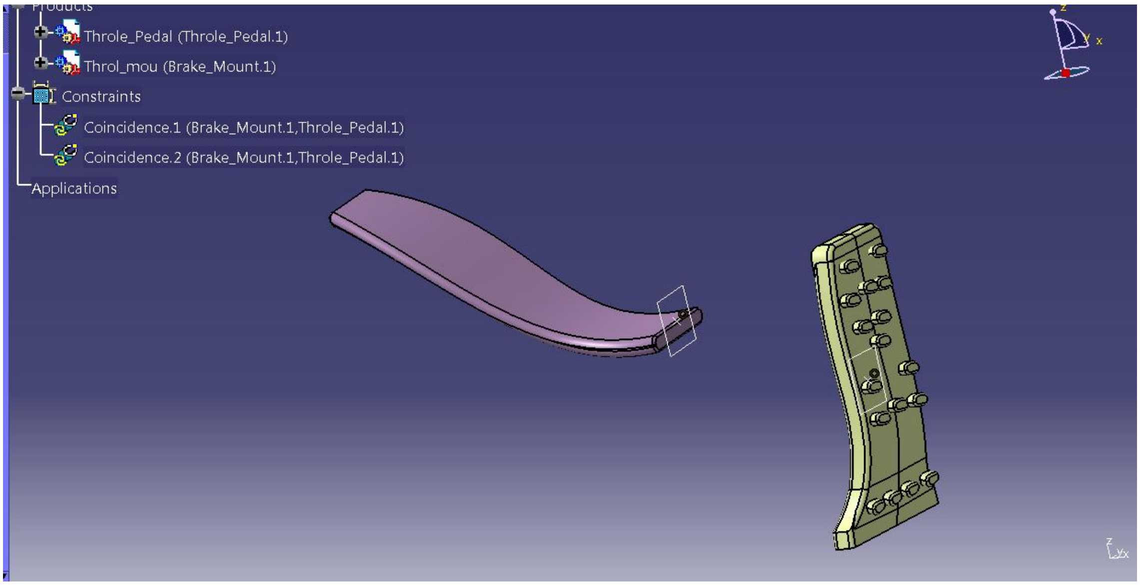
Picture 3.7.2: Coincide constraint of tire's centerline.



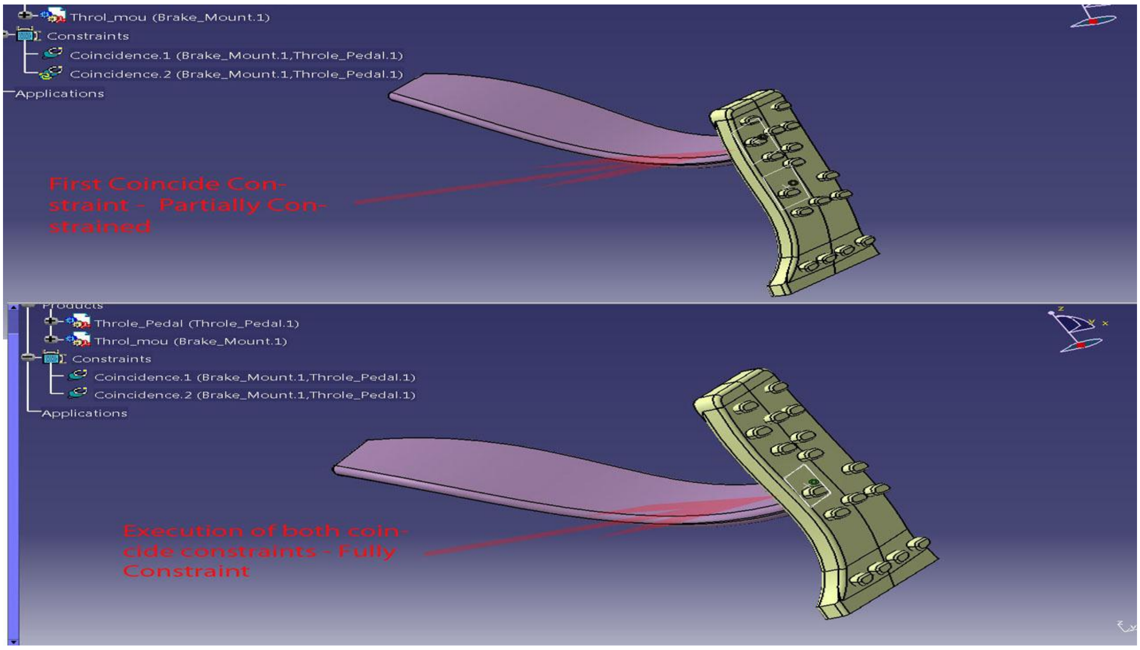
Picture 3.7.3: Tire's complete part.

3.8 THROTTLE PEDAL ASSEMBLY

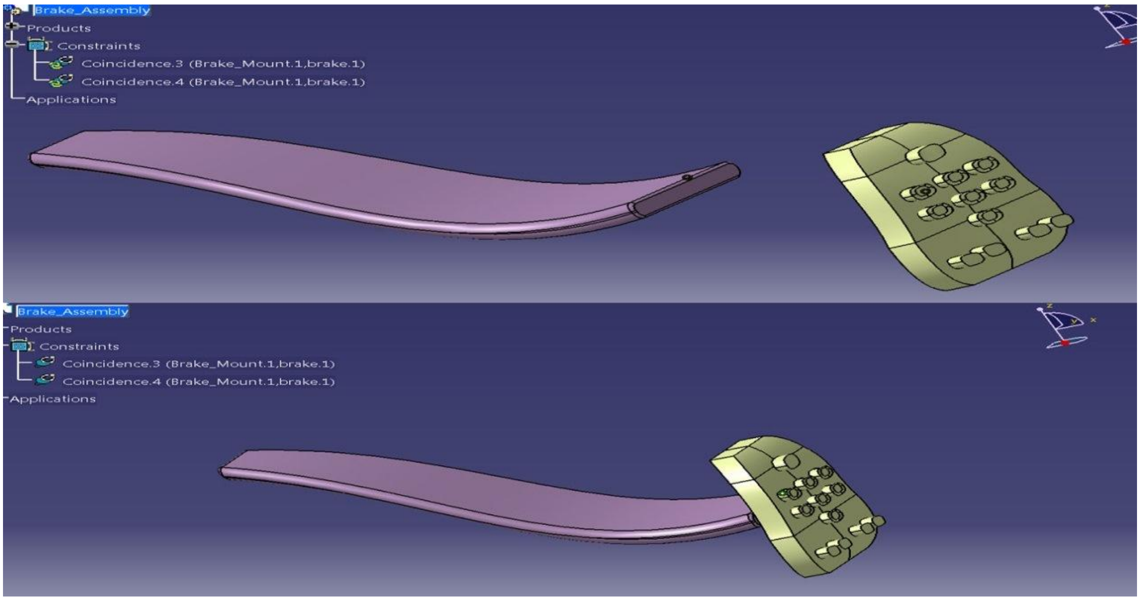
This object was the outcome of two separate parts. The parts that constituted the assembly were the brake pad part and the pedal part (Picture 3.8.1). For the current assembly to be fully constrained, it was mandatory for two coincide constraints to be imposed. The first coincide constraint was imposed between the marginal planes that existed on each part respectively (Picture 3.8.2). The second coincide constraint was set at the two parts' adjoining points. With the imposition of these constraints the exact position of each part in space was determined with absolute accuracy, as shown in Picture 3.8.3 below. Following the exact same methodology, the brake's and clutch's pedal assembly were realized.



Picture 3.8.1: Exploded view of throttle assembly.



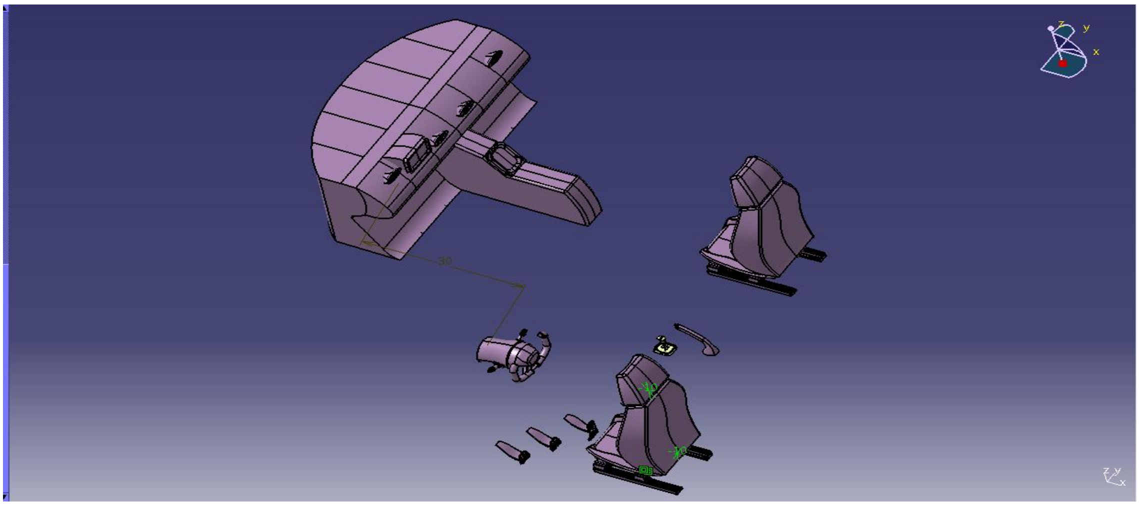
Picture 3.8.2: Definition of stages of throttle’s pedal assembly.



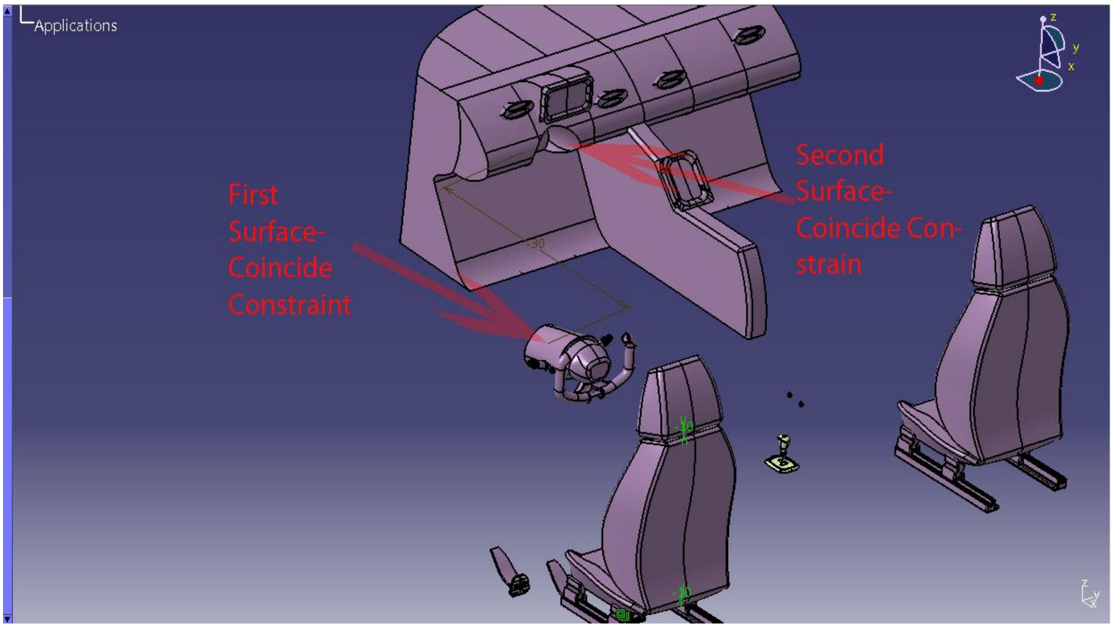
Picture 3.8.3: Definition of brake’s and clutch’s pedal assembly.

3.9 VEHICLE’S CABIN ASSEMBLY

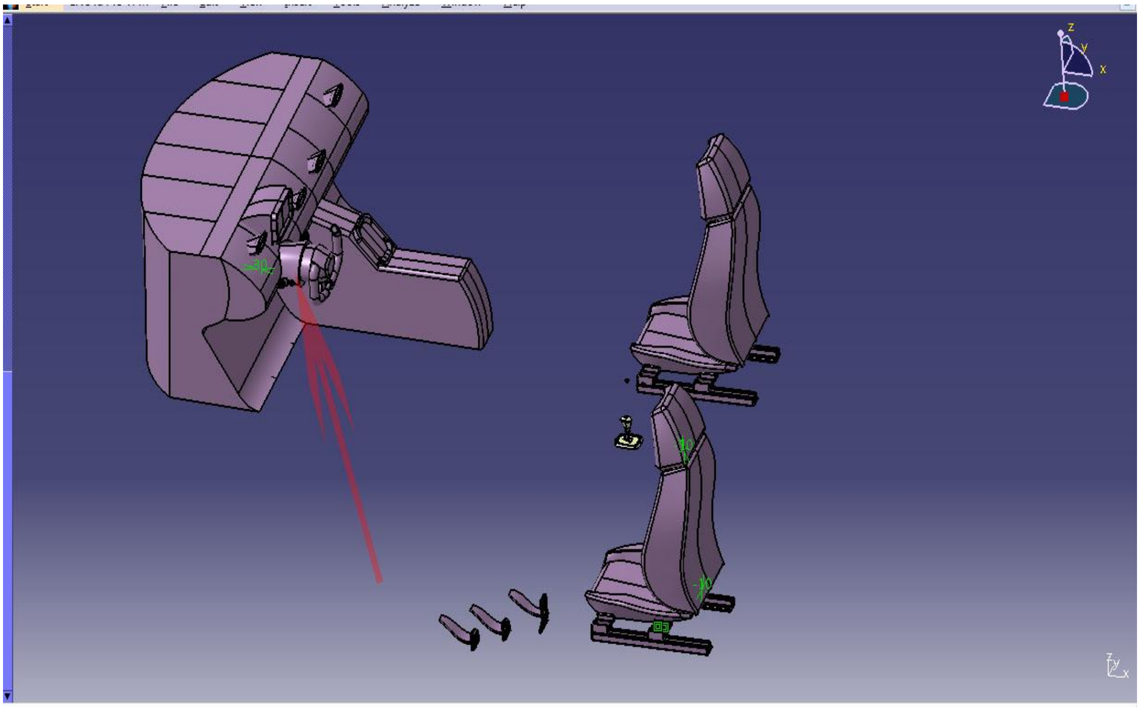
The complete cabin’s assembly constituted the final integration of almost all the independent part designs. In fact it derived as the result of the unification of six independent subassemblies and the dashboard’s solid volume. As it is shown in the exploded view (Picture 3.9.1) below, the cabin’s assembly integrated 8 different entities. For the creation of the final assembly, only two different constraints were imposed. The placement of the remaining modules was done using the ‘’manipulation’’ tool. The two constraints were used to place the steering wheel on the dashboard volume. The first constraint was an offset constraint imposed between two frontier planes. The second coincide constraint was imposed between two surfaces, one peripheral belonging to the steering wheel and another belonging to the dashboard. The remaining modules were placed on their proper position manually, using the manipulation tool. The cabin’s final assembly is shown in Picture 3.9.4 below.



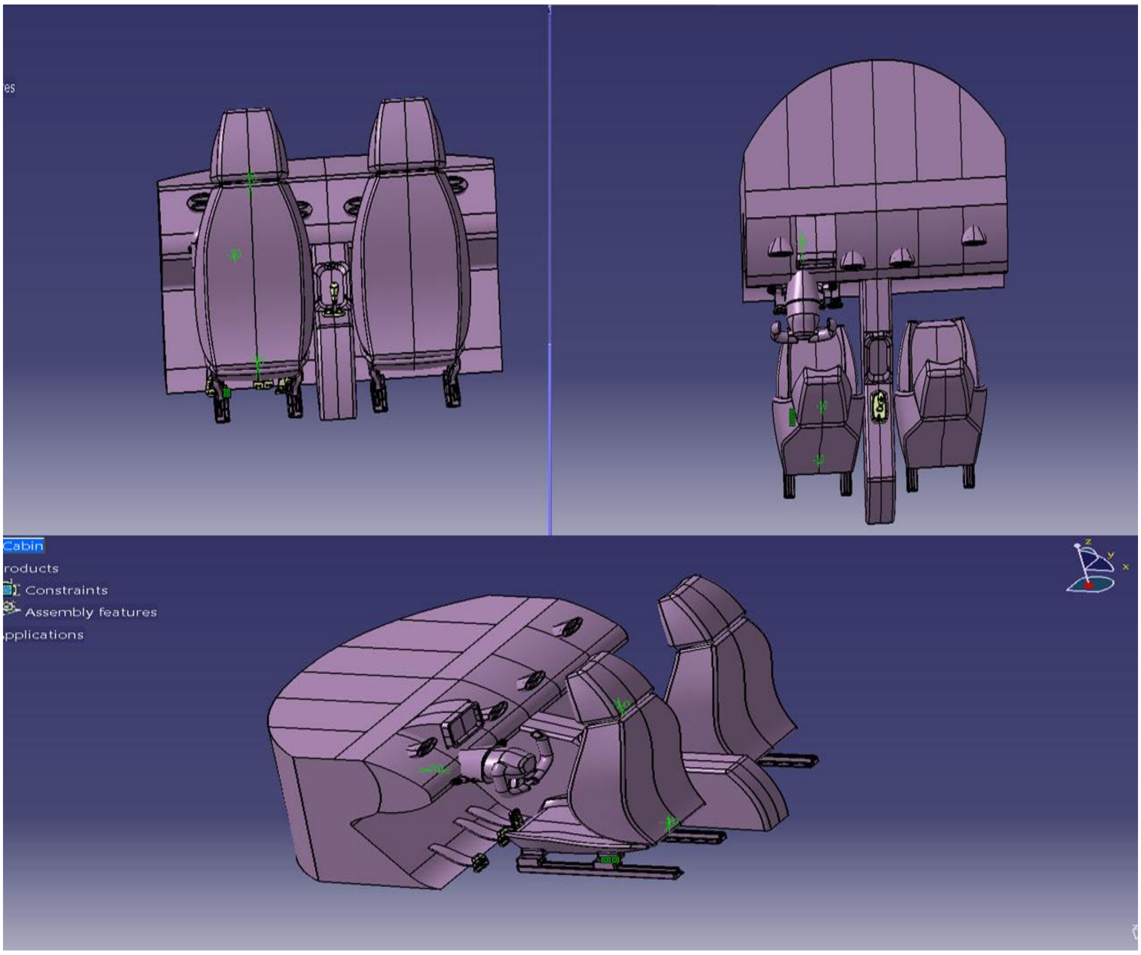
Picture 3.9.1: Exploded view of cabin’s assembly.



Picture 3.9.2: Definition of surfaces where the coincide constraint is imposed.



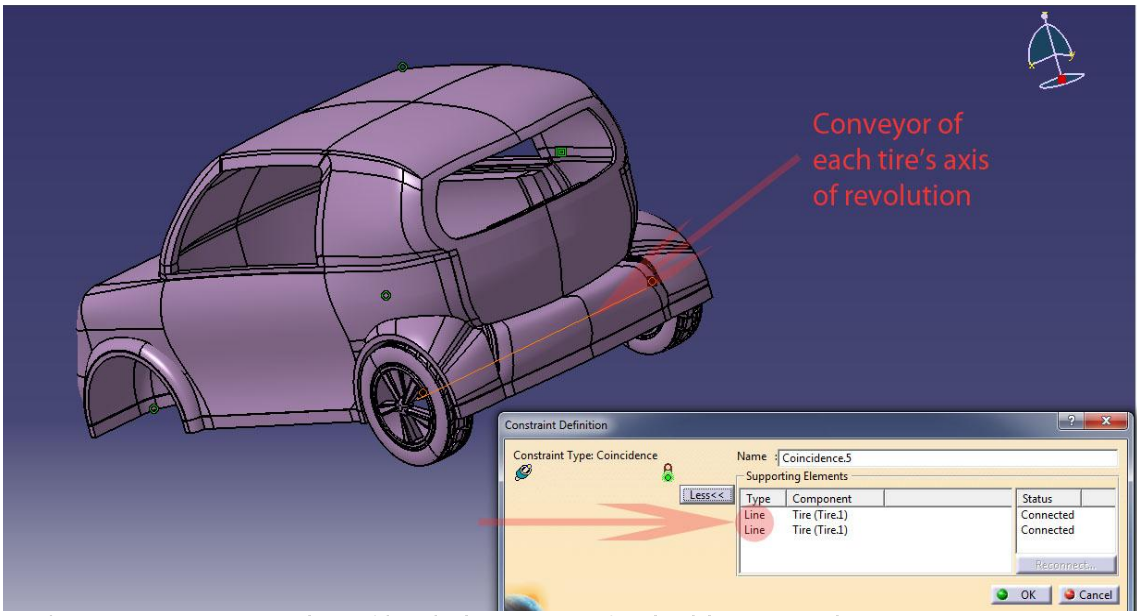
Picture 3.9.3: Steering wheel placement by implementing the coincide constraints.



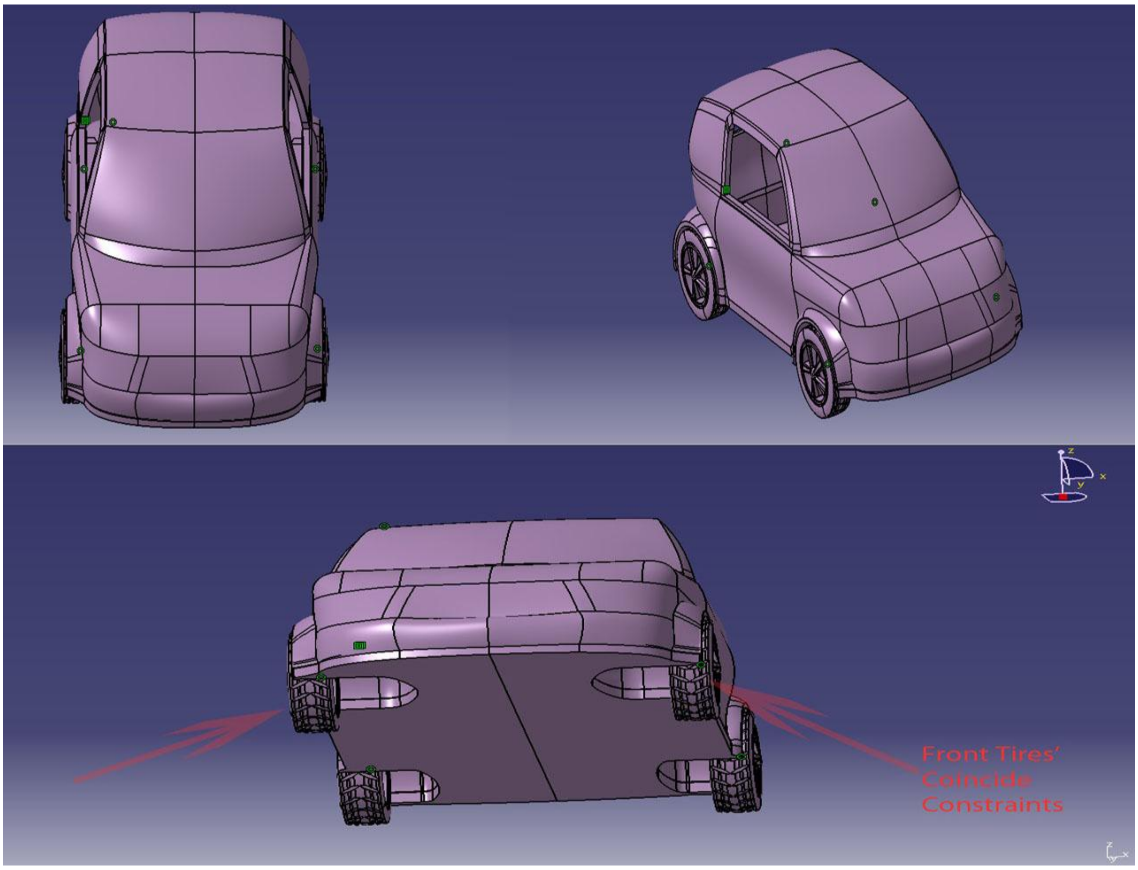
Picture 3.9.4: Final cabin's assembly.

3.10 CAR'S EXTERNAL SHELL WITH TIRES ASSEMBLY

This assembly followed after the vehicle's solid volume design. It consisted of the integration of five independent entities in total. For the completion of this assembly, a single tire subassembly was initially introduced inside the mechanical assembly platform. By exploiting the capabilities provided from the use of the manipulation tool, the first tire's volume was set to its desired location. The second tire's volume was set to its proper location by imposing a coincide constraint. This coincide constraint was imposed between the two axis of revolution which belonged to the two independent tires respectively. The remaining tires were positioned to their proper location using the exact same methodology.



Picture 3.10.1 Assembly of back tires – set of coincide constraints.



Picture 3.10.2 Complete assembly – vehicle body with tires.

4. DESIGN OF VEHICLE'S EXTERNAL SHELL

The design of this part constitutes the second primary module of the CAD operation for this thesis, including the design and assembly of the vehicle's cabin. As every single solid and surface model in this project, this solid volume was also designed as a parametric volume. The shell's external shape was designed to resemble that of a small urban car. The size of the vehicle was selected to approach the size of a two – seater urban car model. Its design was created using Catia's software generative shape design platform. In order to complete the shell's digital mock up, the passenger door solid was used as a reference.

4.1 DESIGN OF VEHICLE'S HOOD VOLUME

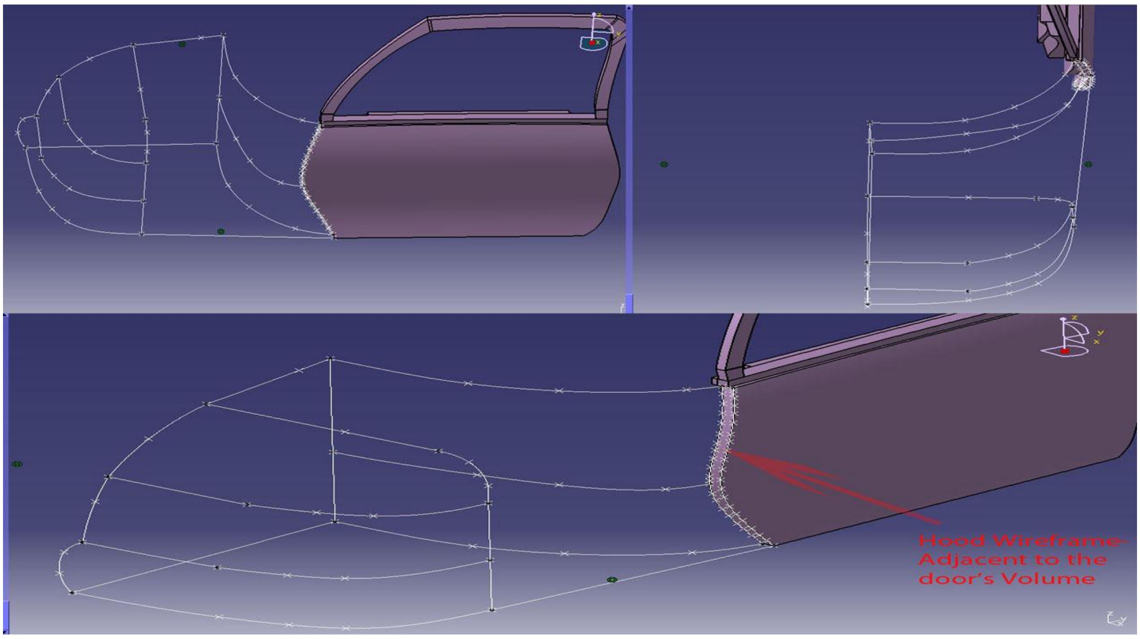
The hood's volume constituted the first feasible geometry for the determination of the solid. This volume was selected as a designing base for the rest volumes of the vehicle. Firstly, the hood's wireframe model was determined. The main attributes of the current wireframe model were its complexity (in comparison to all the previously designed wireframe models) and the fact that it was not defined as a single section. By contrast, a several number of subsequent sections coexisted inside the model. Additionally, the wireframe model (Picture 4.1.1) presented a symmetry with respect to the xz- plane. An important feature that had to be integrated in the model's design was the first degree continuity constraint that had to be imposed between the hood's wireframe and the passenger door solid part. This condition was fulfilled by designing the wireframe model adjacent to the door's external side surface. Next in the design process the hood surface model followed. By producing a series of multi- section and several fill surfaces after exploiting the previously designed wireframe model, this wireframe model was covered by a set of 11 independent surfaces. Right after its production, these surfaces were unified into a single surface with the use of *"join"* operation command (Picture 4.1.2). The second symmetrical set of surfaces derived by executing a *"symmetry"* operation command. As input inside the symmetry's command window, the previously produced surfaces were inserted as element, while a plane parallel to the yz – plane was defined as plane of symmetry (Picture 4.1.3). The hood's final solid volume was formed by executing the *"close surface"* (Picture 4.1.4) command. As input in the ''object to close'' option inside the command window the set of the hood's symmetrical surfaces was imported.

As for the dimensions' measurement, the main dimension components that were calculated for the design of the hood module, was the hood's length, height and width. The hood's height component was easy to identify as it was taken from an already existing passenger door part.

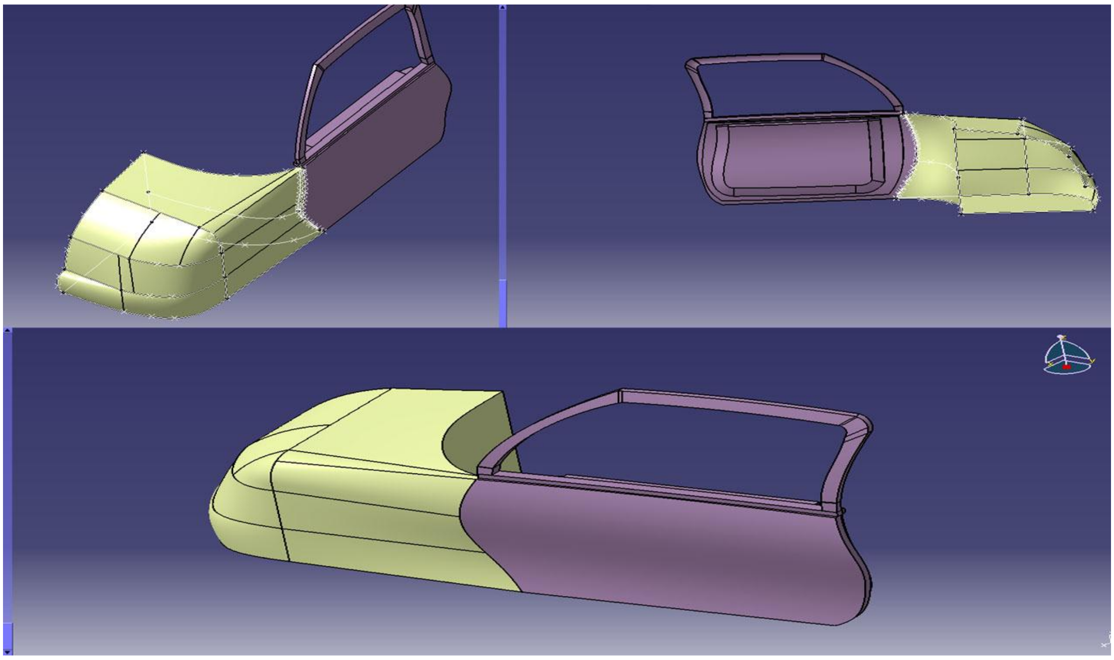
The exact value of each dimension component is depicted in Table 4.1.1 below.

Table 4.1.1

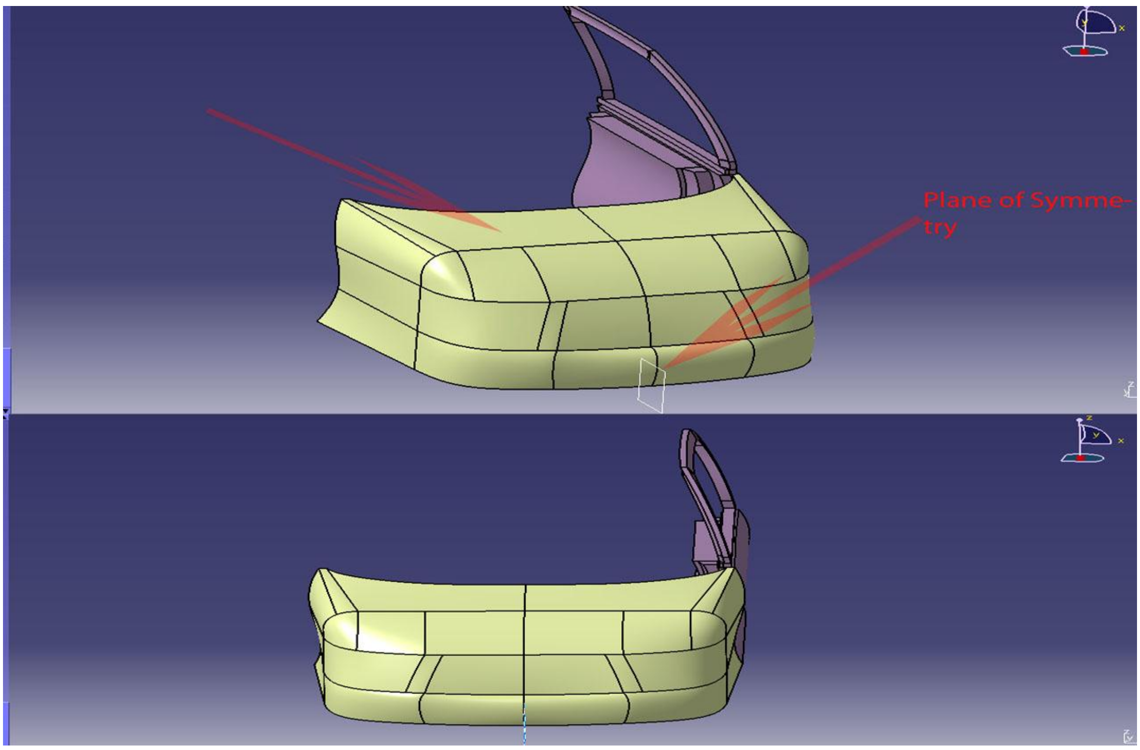
Dimension Components/ Numerical Values (mm)	
Height	600
Length	1052
Width	1500



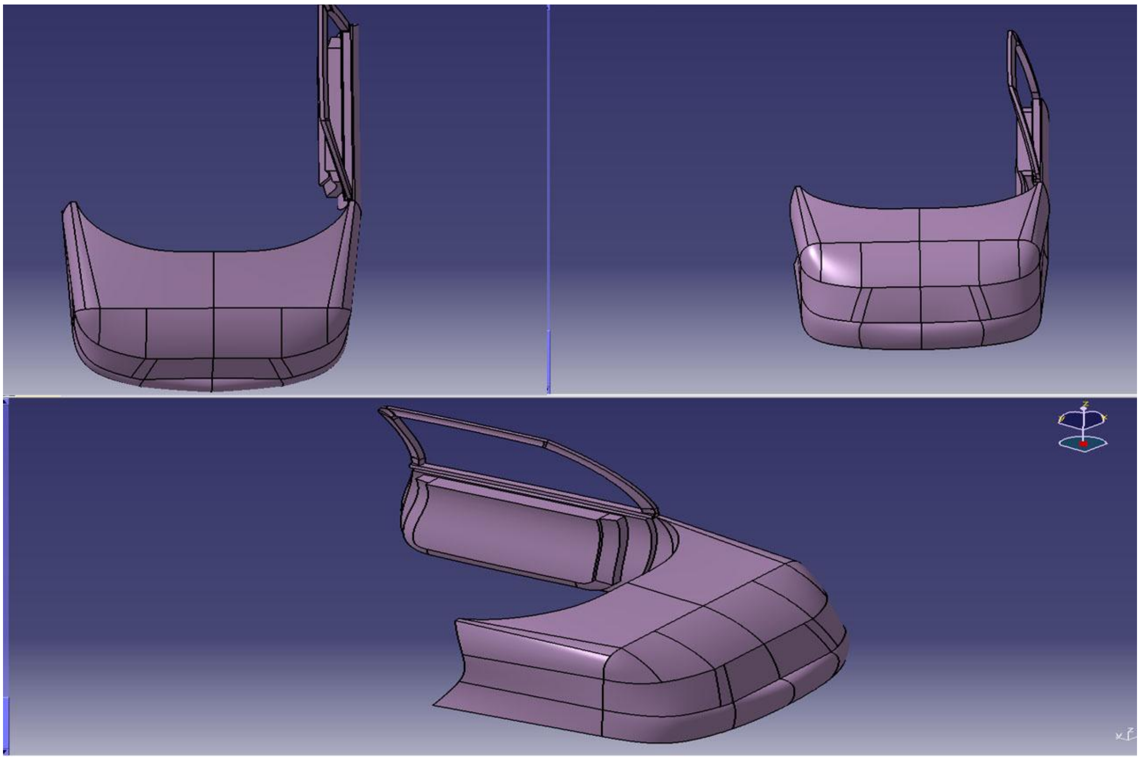
Picture 4.1.1: Hood's wireframe model.



Picture 4.1.2: Hood's surface model.



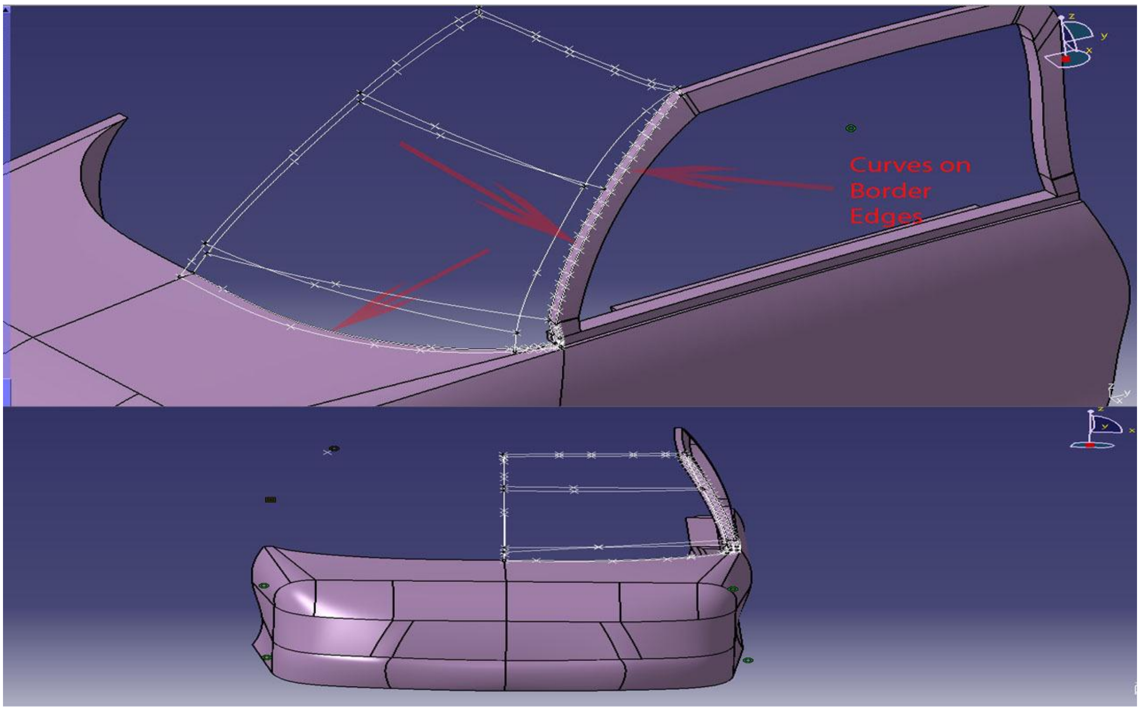
Picture 4.1.3: Hood's surface model.



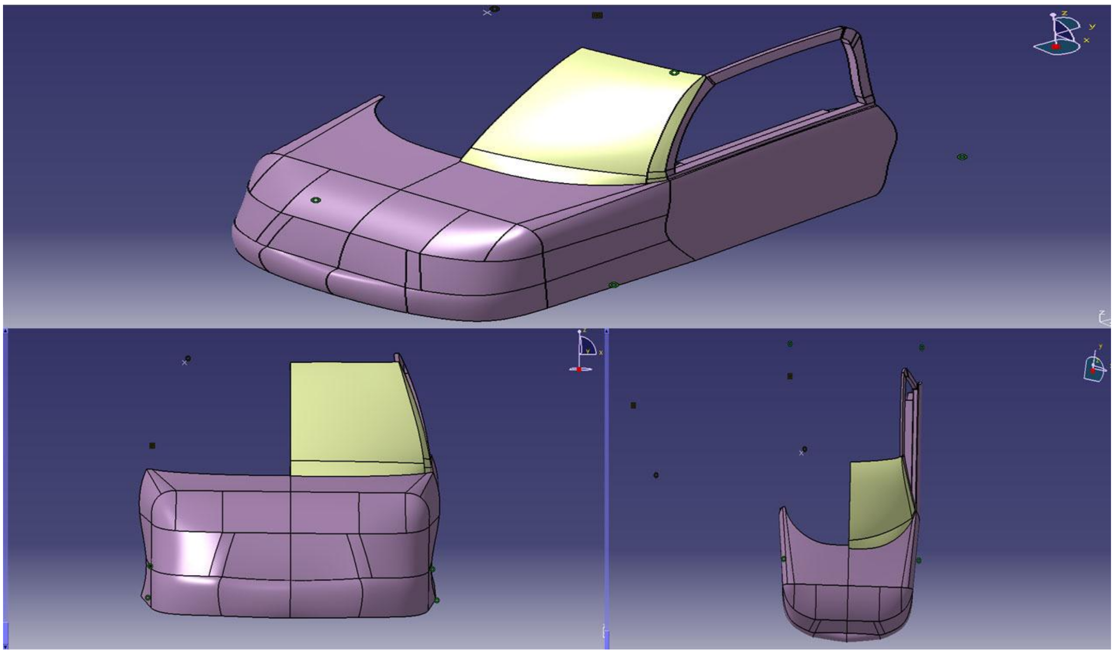
Picture 4.1.4: Hood’s solid volume.

4.2 DESIGN OF VEHICLE’S WINDSCREEN

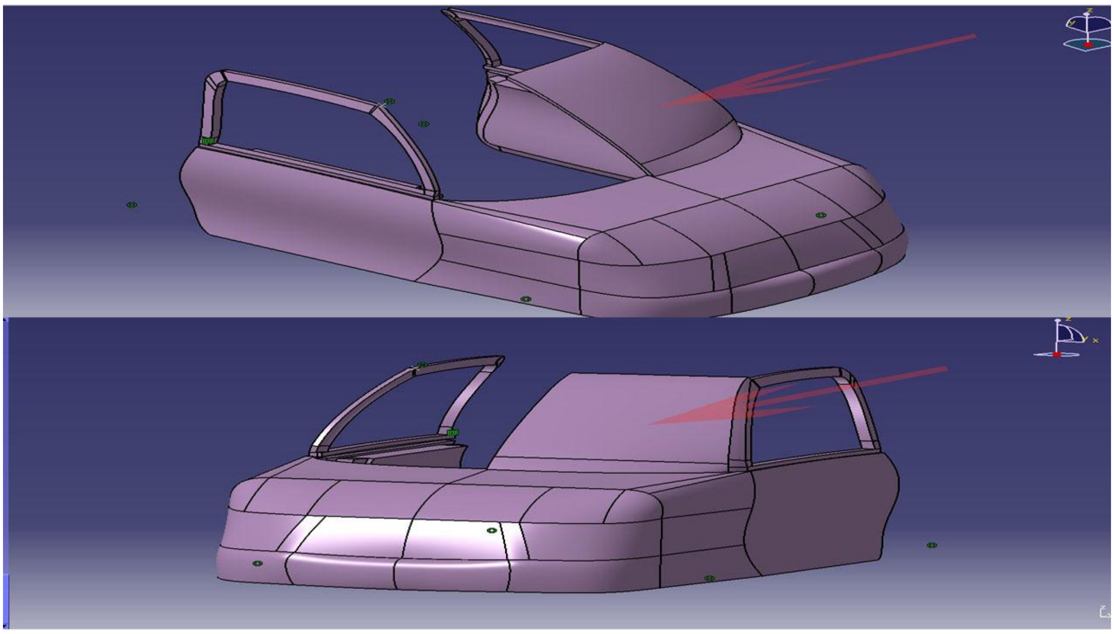
This solid constituted the second feasible option for the design of the vehicle’s solid volume. As with the previous hood volume, this current volume was designed as a parametric solid. For the construction of this certain volume, the door and hood solid volumes were used as references. Moreover, this current volume just like the previous ones, presented a symmetry with respect to yz plane. Consequently, only half of the total wireframe had to be produced. As a base for the production of the final volume the windscreen’s wireframe model was designed first (Picture 4.2.1). In order to secure a first degree continuity between the door and hood volume, a number of curves were designed on the border edges of each volume respectively. These curves were designed by joining a number of previously defined points adjacent to each border edge respectively. The points were defined using the ‘’point on curve’’ option inside the point’s definition command window. To achieve the door’s and hood’s shape with accuracy, a good number of points were set on each edge respectively. Next in sequence followed the design of the windshield surface model (Picture 4.2.2). By exploiting the previously produced wireframe model, the gaps that existed between the model's wires were covered by nine in total multi section surfaces and one fill surface. Then, the solid volume was formed after executing the "*close surface*" command (Picture 4.2.3). As input for the execution of the close surface operation, the previously defined set of surfaces was used. The second symmetrical half volume derived by executing a "*symmetry*" operation command. As input inside the command's element bracket the previously designed solid volume was inserted while as plane of symmetry a plane parallel to the yz –plane was selected (Picture 4.2.4).



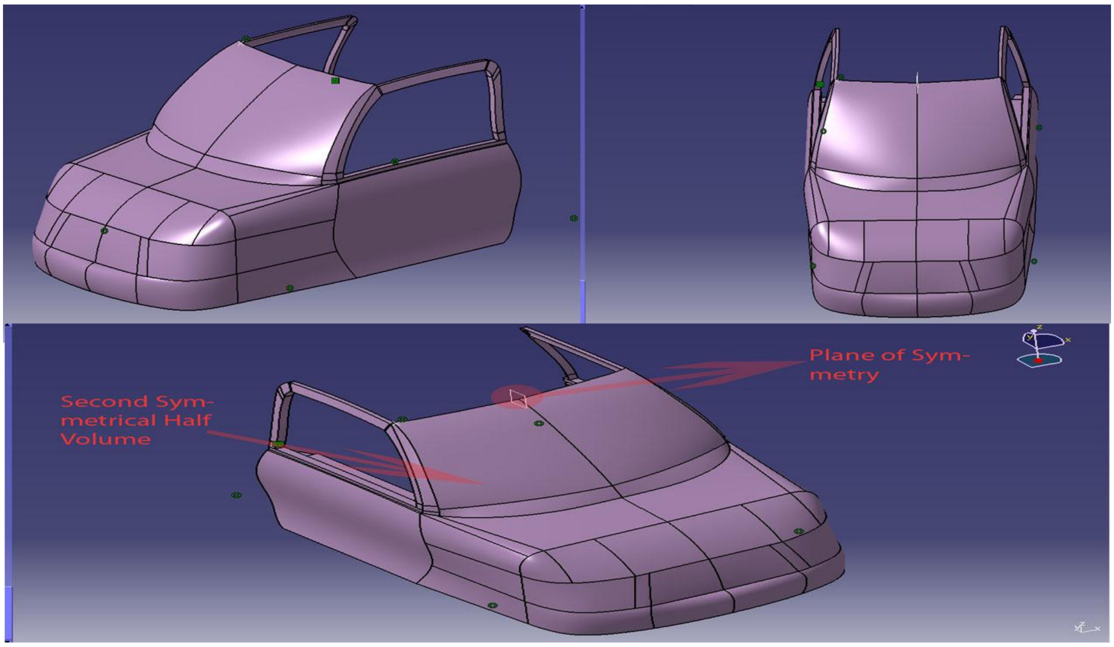
Picture 4.2.1: Windshield's wireframe model.



Picture 4.2.2 Windshield's surface model.



Picture 4.2.3 Windshield's solid volume.



Picture 4.2.4: Windshield's symmetrical volume.

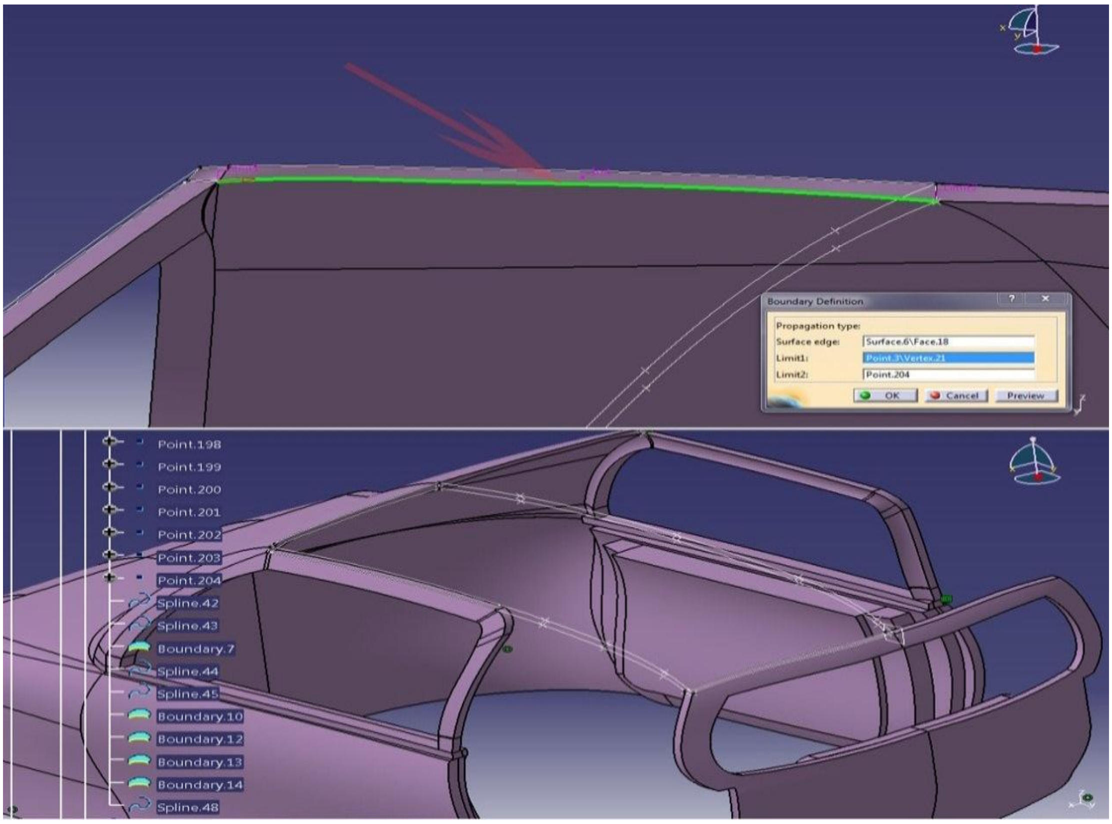
4.3 DEFINITION OF VEHICLE'S ROOF

The next volume to be created was the vehicle's roof. This solid volume, as the previous two volumes, was defined as a parametric solid. Moreover, it also presented a symmetry with respect to the yz plane. In order for the solid to be defined, it was necessary that the trunk solid part be introduced inside the designing platform. Moreover, for the proper design of the roof's solid volume the passenger's door, trunk and windscreen volumes were used as references.

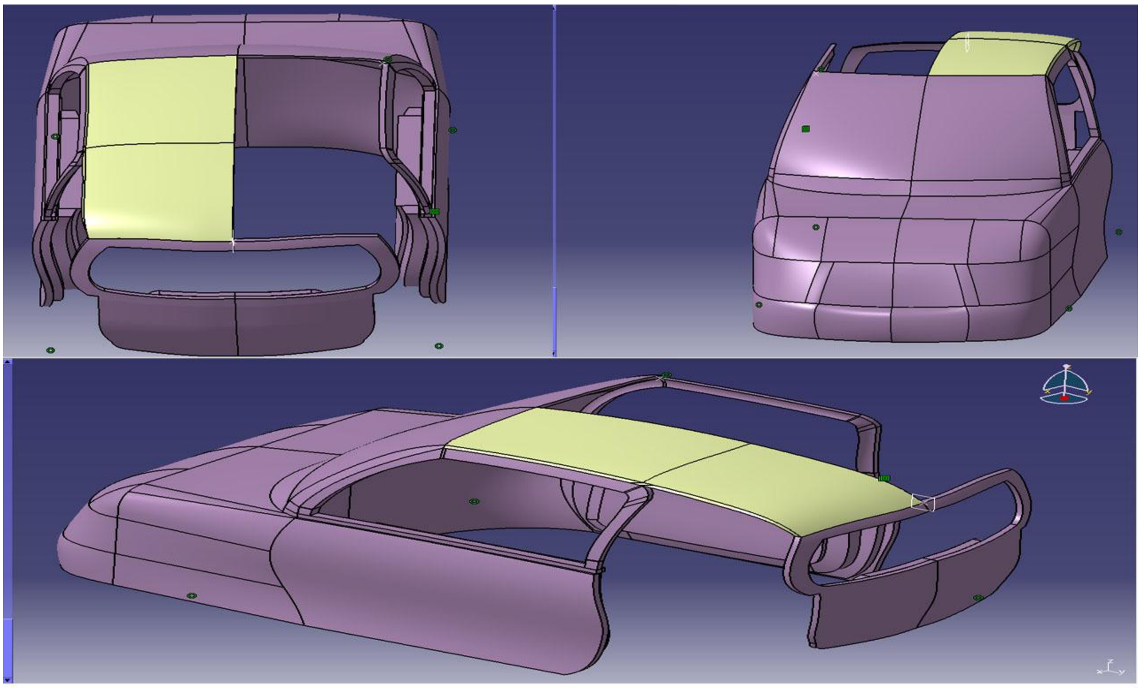
The wireframe model constituted the base for the solid volume construction (Picture 4.3.1). On the wireframe model definition a continuity constraint between the solid volume and the windscreen, trunk and door volume was imposed. At the current wireframe model a different technique was followed in order to measure the solid volumes' border edges with absolute accuracy. On each external surface a "boundary" operation was executed. With the execution of this certain command, it was made possible to obtain a border edge of each external surface without any discontinuities. Next in sequence the design of the roof's surface model followed. By using the previously defined wireframe model as a reference, the gaps between the wires were covered by six multi section surfaces in total (Picture 4.3.2). The set of surfaces was unified into a single entity with the implementation of bilateral connections between the surfaces. These connections between the surfaces were realized by using the "*healing*" operation. The final solid volume was created using the "*close surface*" operation command (Picture 4.3.3). As in the previous part's design, the second symmetrical half of the roof's solid volume was created by using the *"symmetry"* operation command (Picture 4.3.4). The primary dimension component that was mandatory to calculate was the roof's total length. The exact numerical value of the roof's length was obtained from similar small urban vehicles (Table 4.3.1).

Table 4.3.1

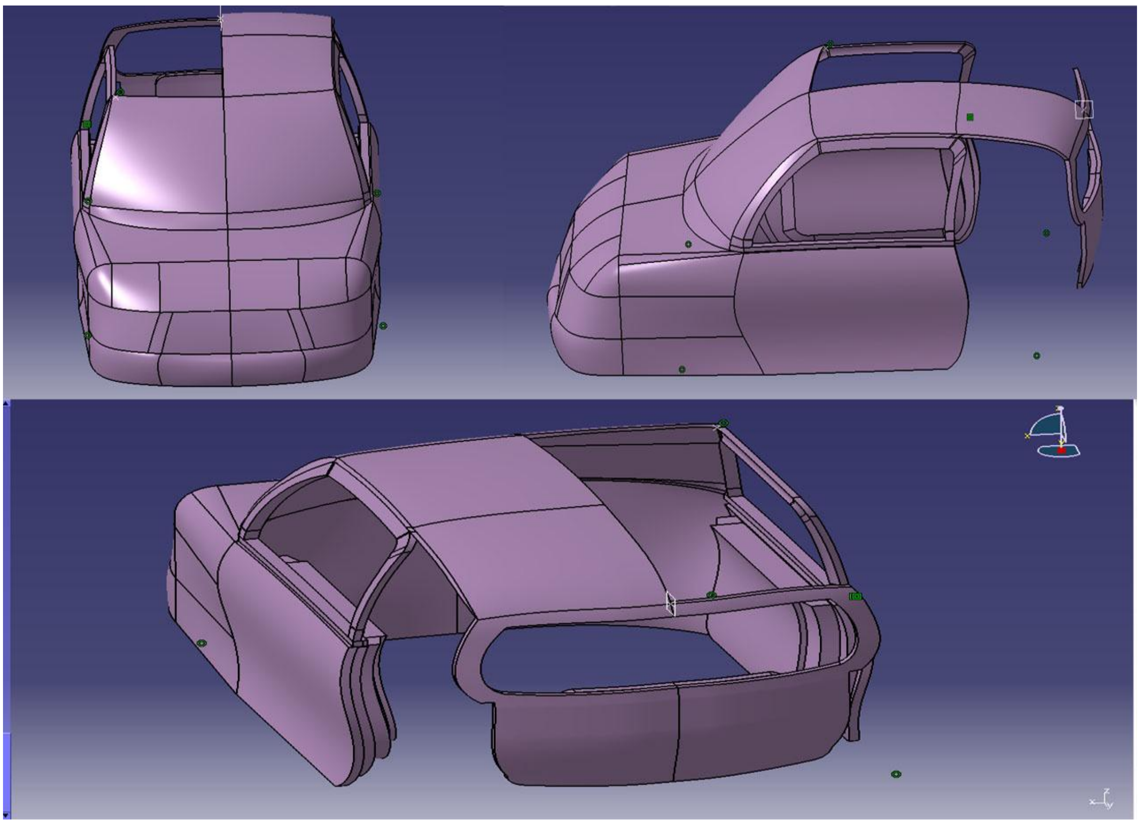
Dimension / Numerical Value (mm)	
Length	1373



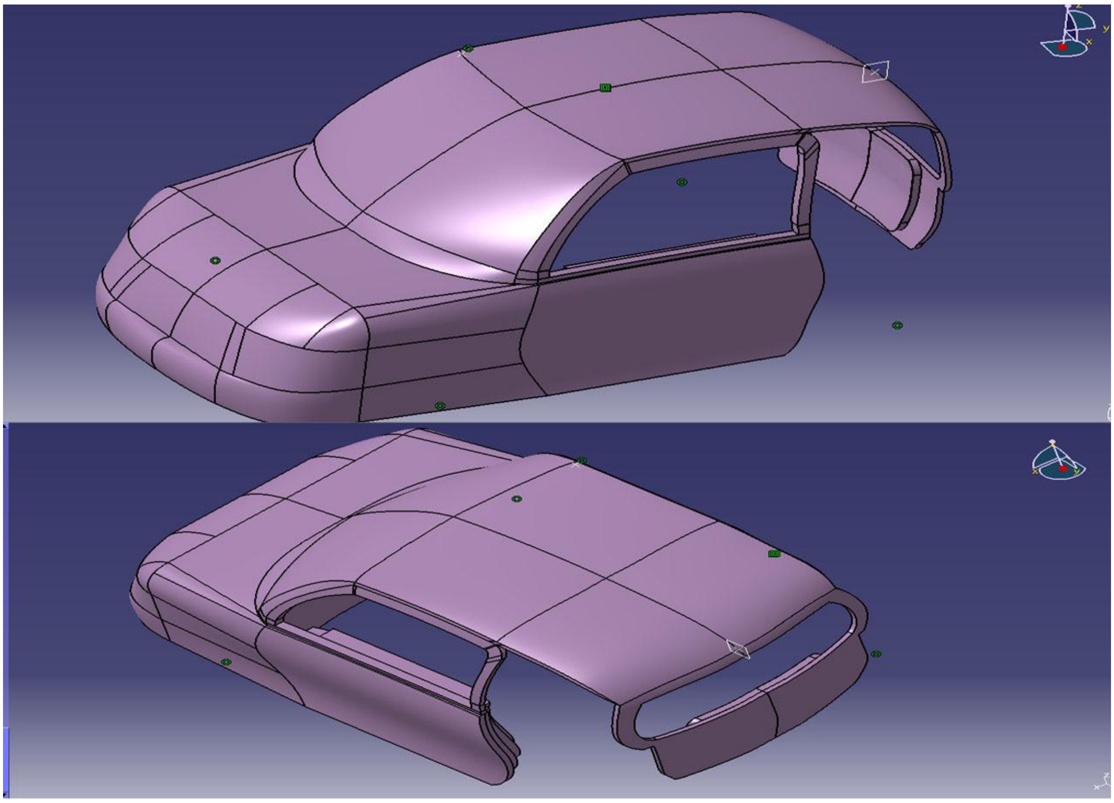
Picture 4.3.1: Roof's wireframe model.



Picture 4.3.2: Roof's surface model.



Picture 4.3.3: Roof's solid volume.



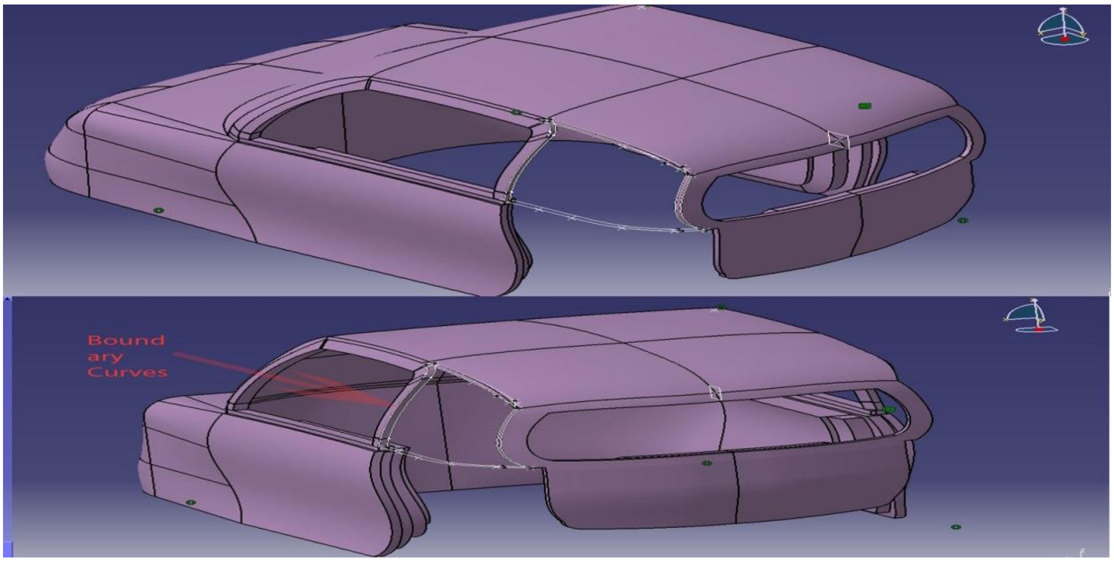
Picture 4.3.4: Roof's complete solid volume.

4.4 DEFINITION OF BODY'S SIDE SOLID VOLUME

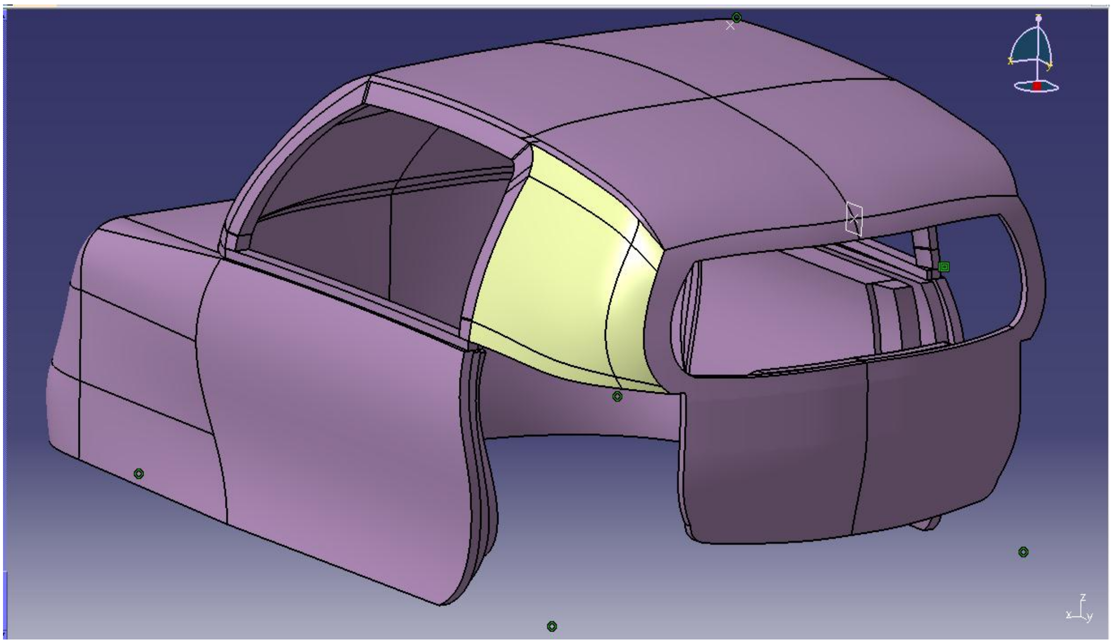
The next volumes designed were the vehicle's lateral solid volumes. The primary features of these solid parts were that they were defined as parametric volumes and presented a symmetry with respect to a plane parallel to yz plane. This volume was divided into three independent modules. Each module was separately defined.

At first, the wireframe model of the first module was created. As it can be inferred from Picture 4.4.1 of the model below, this wireframe presented a continuity with the door's and trunk's volumes. The border edges of the respective volume's external surface that had to be introduced into the wireframe's model design were imported as ''boundary'' curves. Then, by exploiting the wireframe model as a base, the surface model was produced (Picture 4.4.2). The model consisted of a multi-section surface and an extruded one. The multi section surface constituted the external surface of the model. The extruded surface derived by using the previous multi section surface as an input. The extrusion's direction coincided with the direction of x-axis. The total length of the extrusion was 25 mm towards the negative direction of x-axis. The final solid volume, was formed by executing the *"close surface"* operation command (Picture 4.4.3). As input inside the ''object to close'' option at the command's window the previously designed extruded surface was inserted. The second symmetrical volume was formed after the execution of the *"symmetry"* command (Picture 4.4.4). As plane of symmetry a plane parallel to the yz plane was selected. With the same methodology the second module's solid volume was created (Picture 4.4.5).

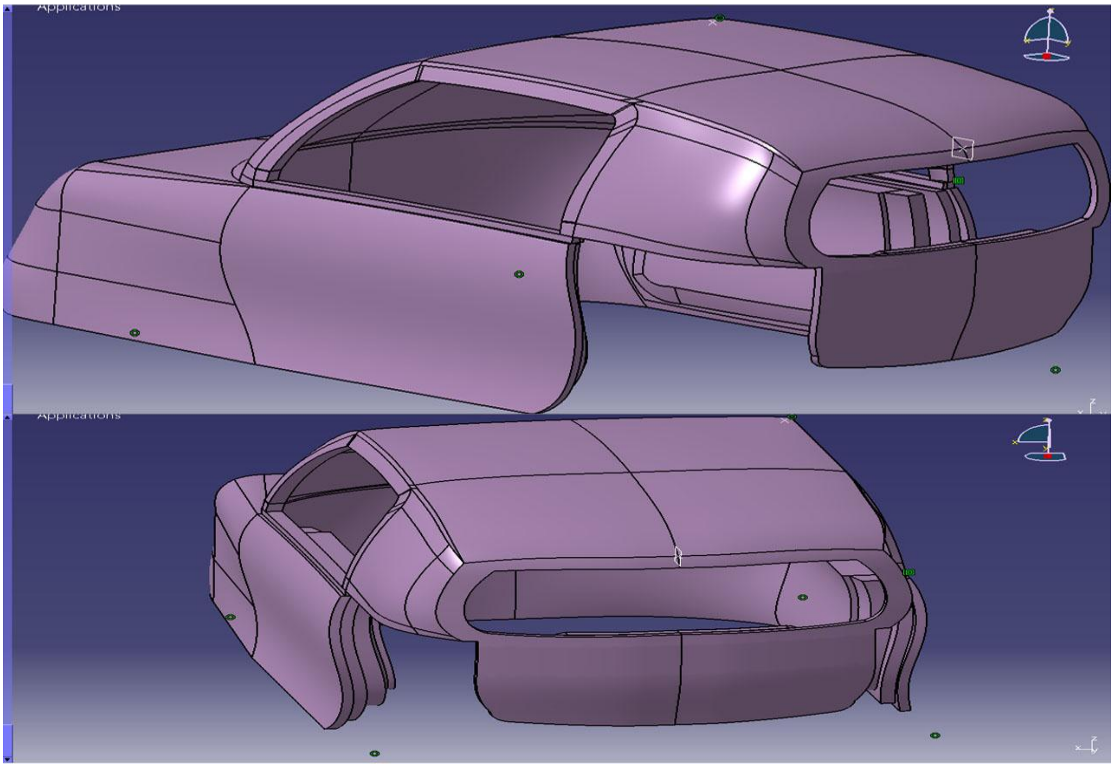
The third module that was created completed the lateral solid volume of the vehicle's 3-D digital mock-up. This current module corresponds to the vehicle's bumper volume. The main features of this volume coincide with those of the previously defined solid modules. Firstly, the module's wireframe model was determined. As it is depicted in Picture 4.4.9 below, the wireframe model falls within the continuity constraint between the passenger's door and the trunk's solid volume. Next in sequence, the surface model was determined (Picture 4.4.10). The model consisted of two multi section surfaces and one fill surface. Then, by joining together the previously designed set of surfaces with the use of "healing" operation command, the final bumper solid geometry was derived. The solid geometry was constructed by using the "*close surface*" operation command (Picture 4.4.11). The second symmetrical half was formed by executing a "*symmetry*" operation command (Picture 4.4.12).



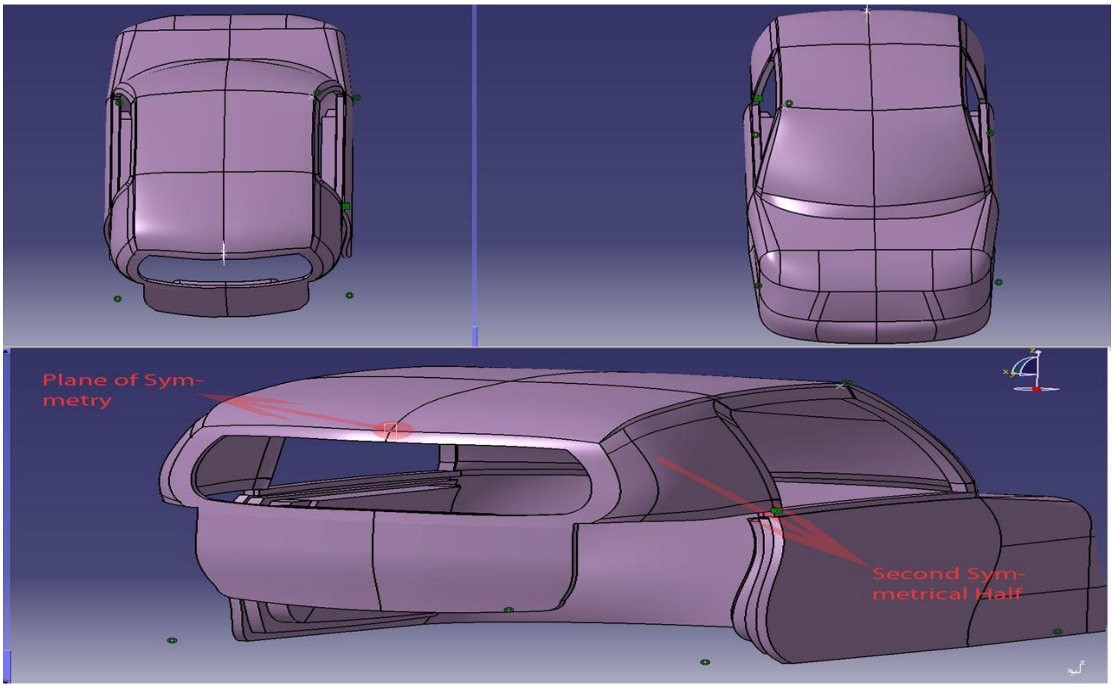
Picture 4.4.1: Lateral volume's wireframe model – first module.



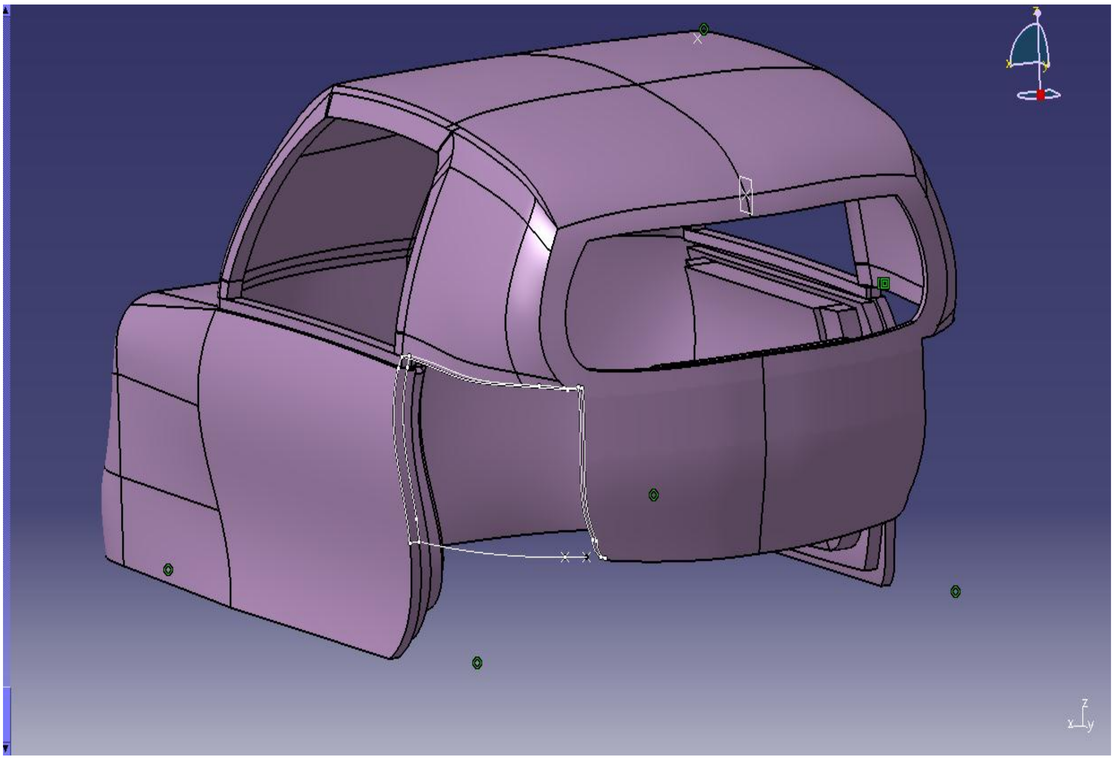
Picture 4.4.2: Surface model of body’s side first module.



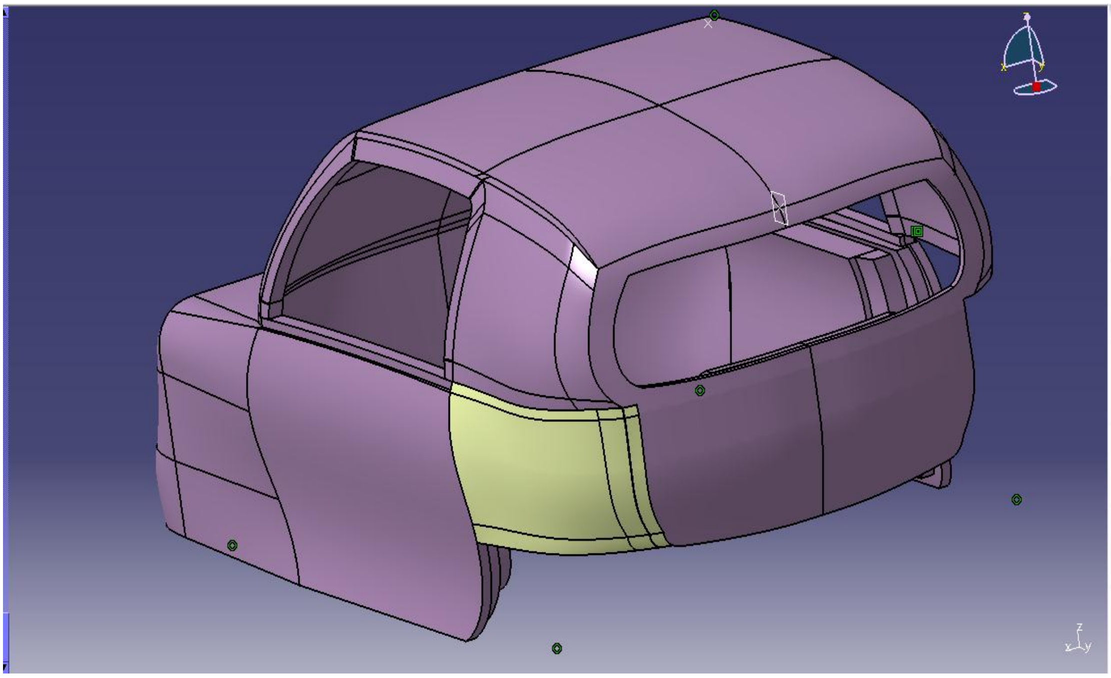
Picture 4.4.3: Creation of first volume of vehicle’s body side.



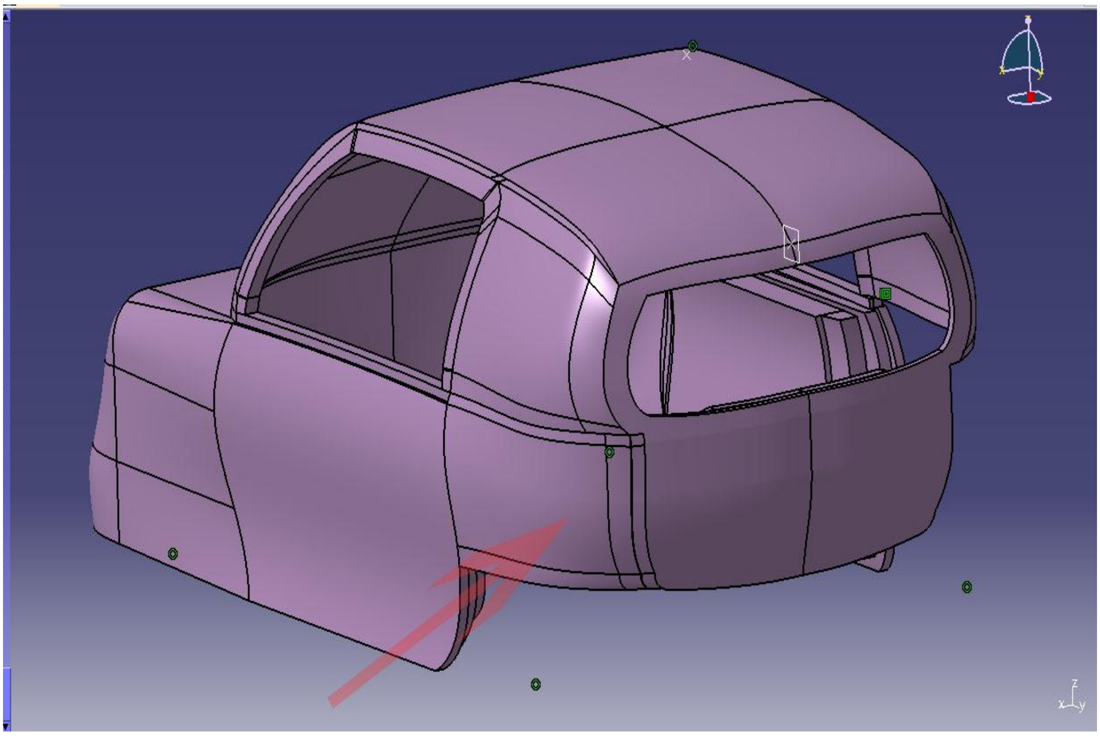
Picture 4.4.4: Creation of first module's symmetrical volume.



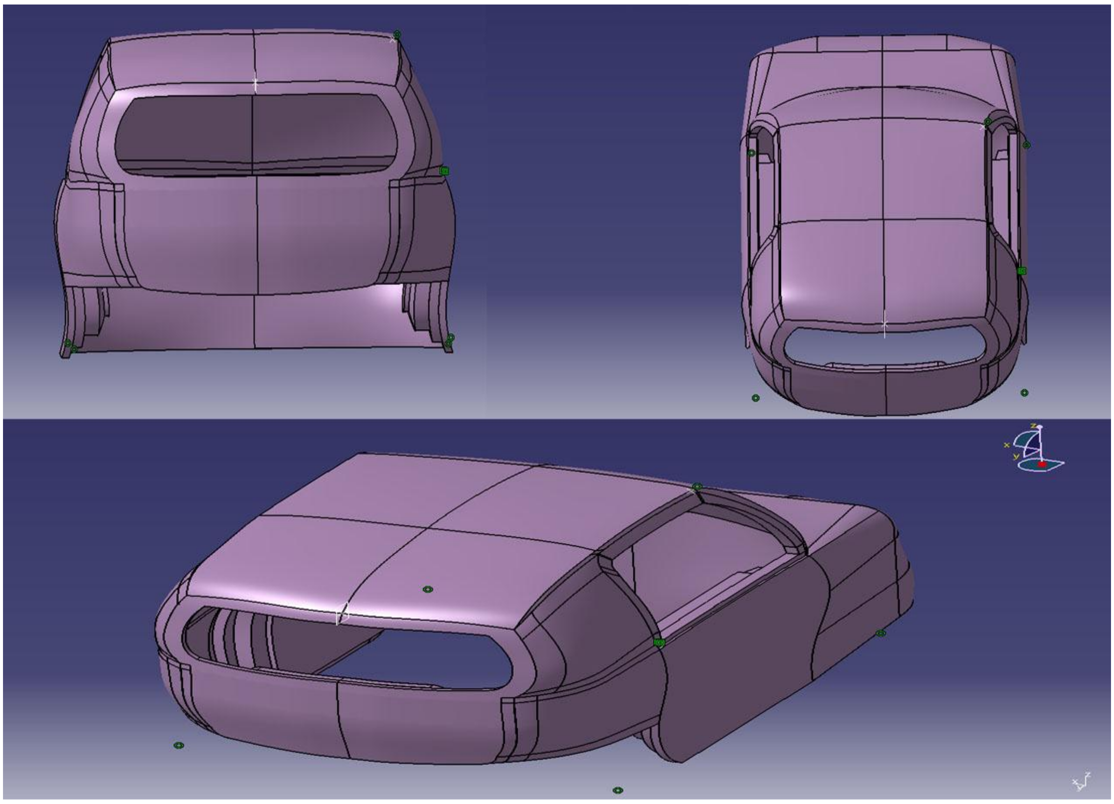
Picture 4.4.5: Creation of second module's wireframe model.



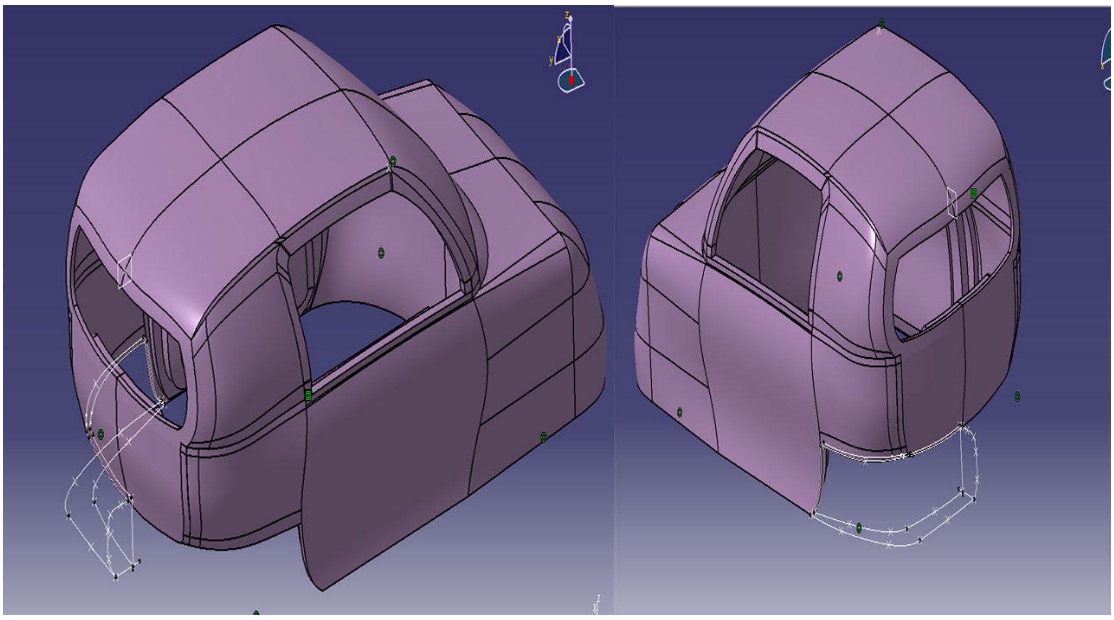
Picture 4.4.6: Creation of second module’s surface model.



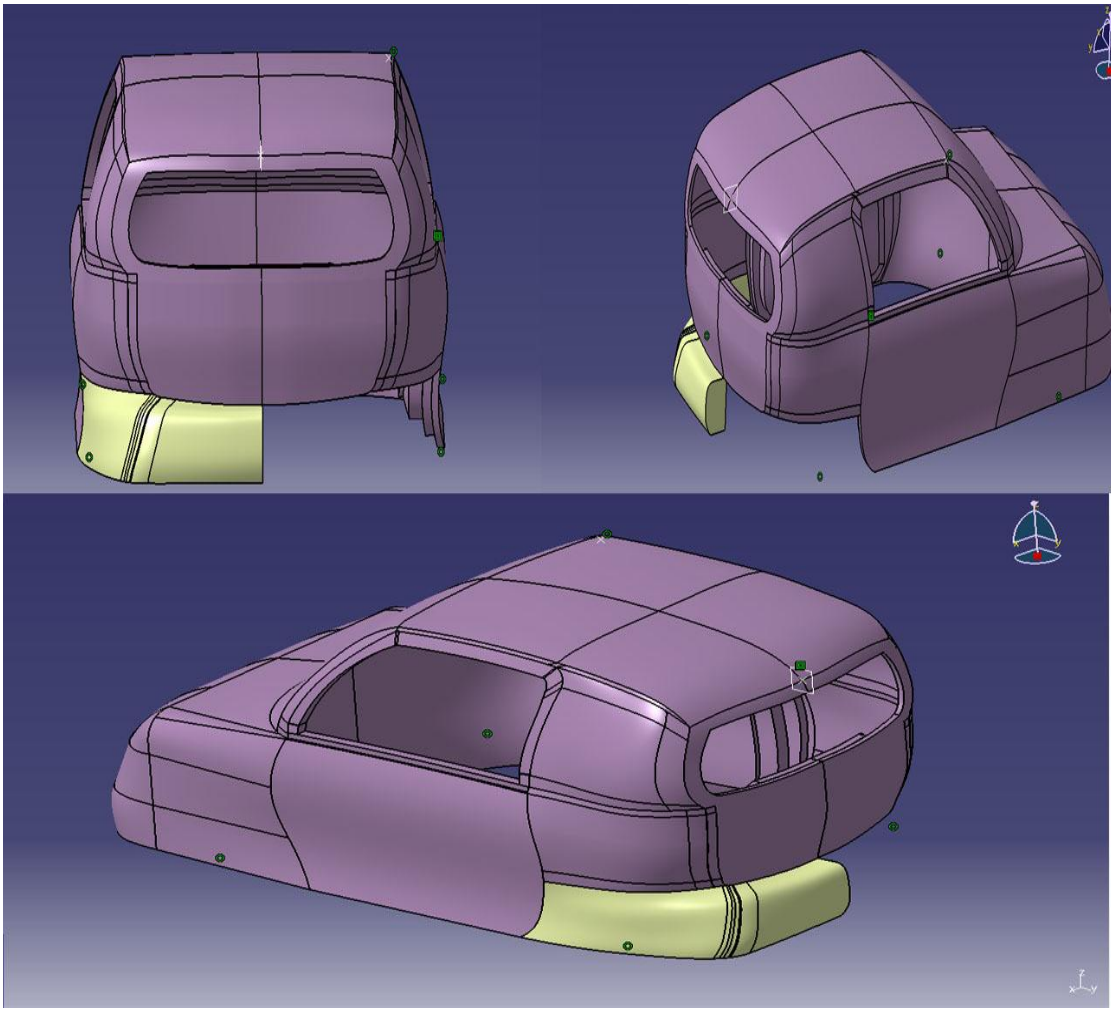
Picture 4.4.7: Creation of second module’s solid volume.



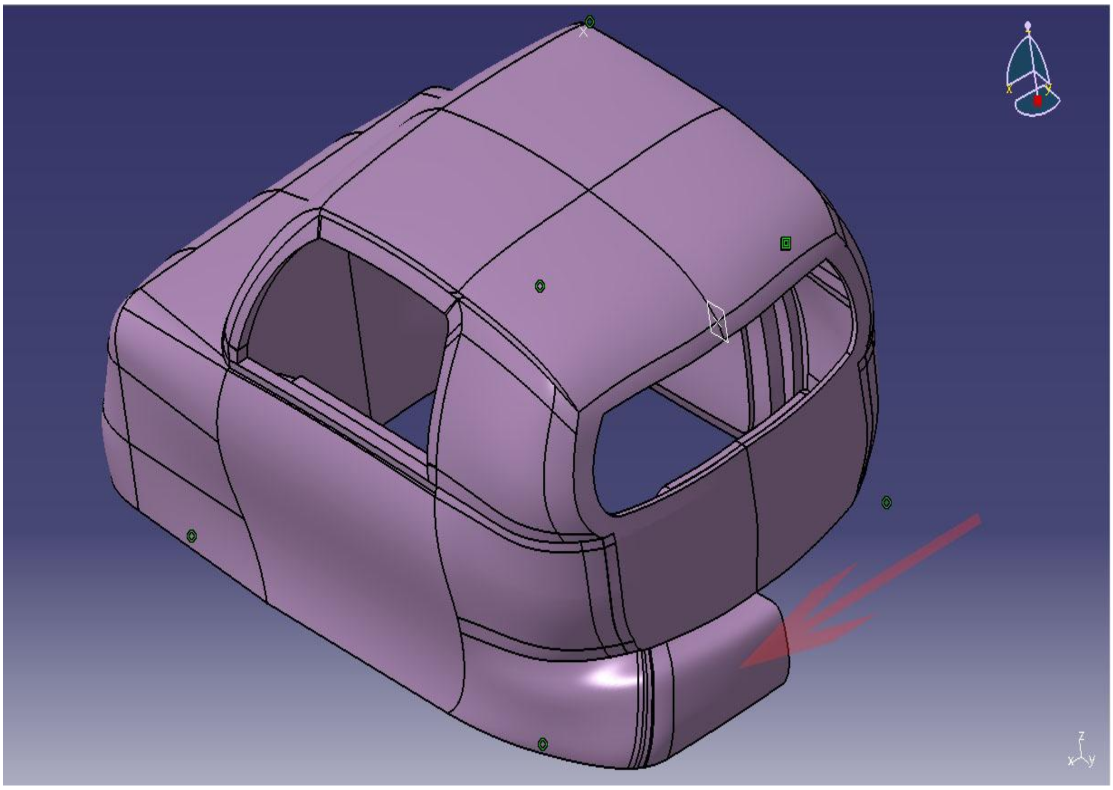
Picture 4.4.8: Definition of second module's second symmetrical volume.



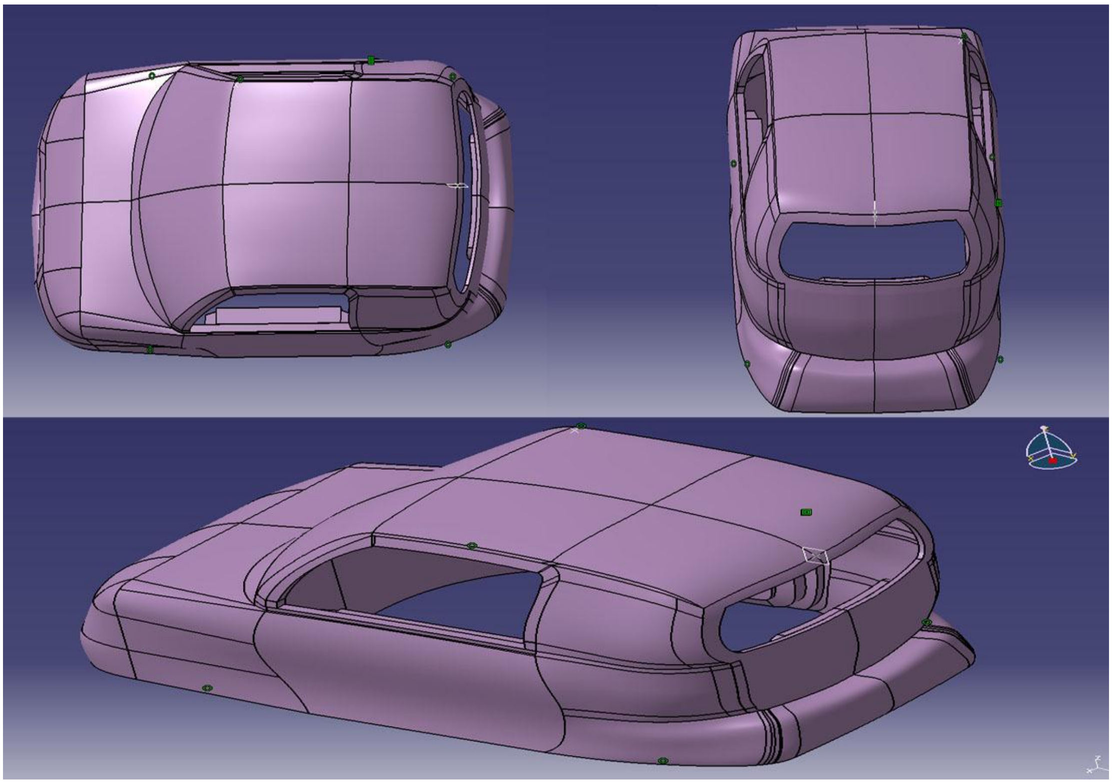
Picture 4.4.9: Creation of bumper's wireframe model.



Picture 4.4.10 Bumper’s surface model.



Picture 4.4.11 Bumper’s solid volume.



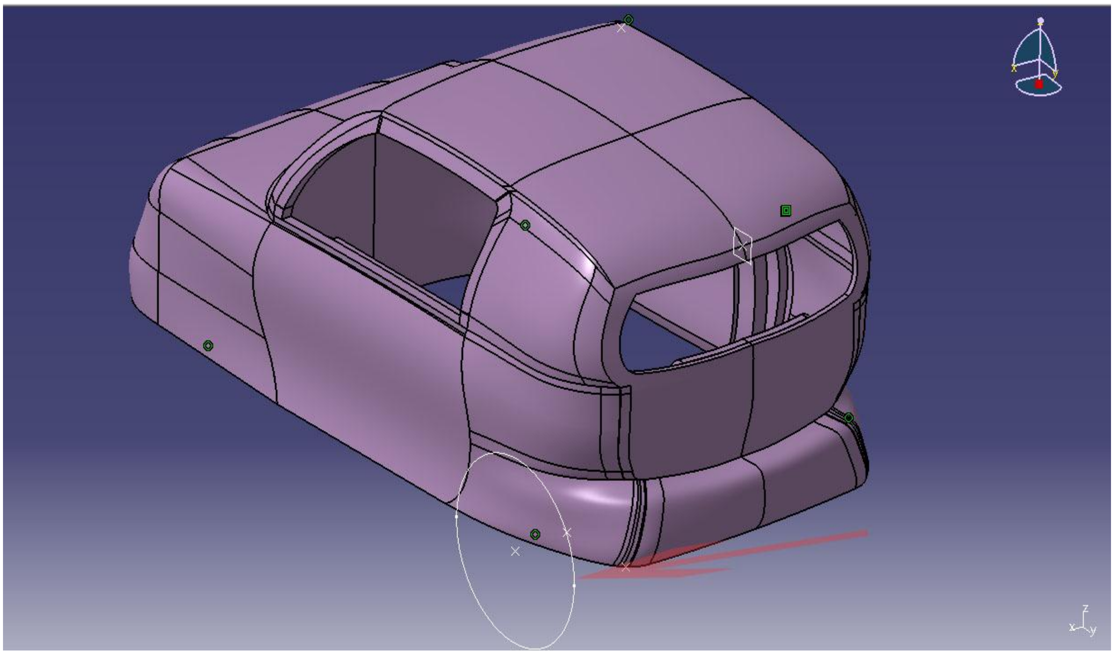
Picture 4.4.12: Creation of bumper’s symmetrical volume.

4.5 DESIGN OF TIRES' SOCKET HOLES

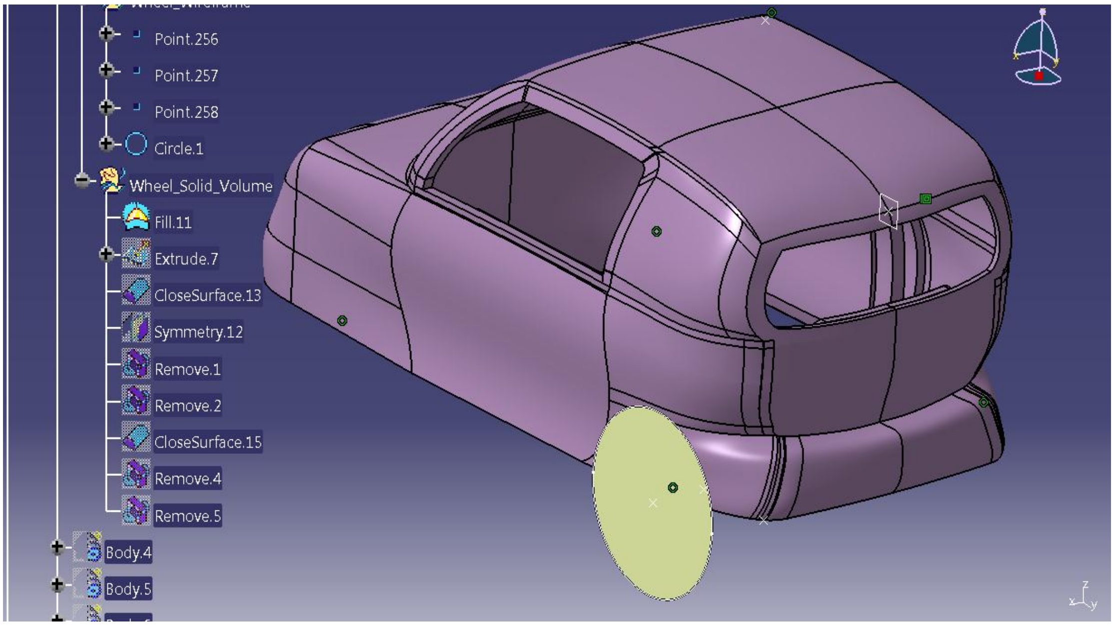
This current feature is not a solid geometry. It resembles more an operation than a solid geometry. The operation's primary scope, as it can be inferred from the title, was the design of an opening on the current vehicle's geometry. The functional purpose of this opening was the creation of a space inside the vehicle's volume for the creation of the canopies and the mounting of the tires.

For the creation of this feature, a similar volume removal technique was followed. This technique involved the removal of a certain part from the current volume body. This removal was executed using a "*Remove Volume*" Boolean operation. As it has already been mentioned, for the execution of a removal's Boolean operation, apart from the primary volume, an auxiliary solid volume also had to be constructed.

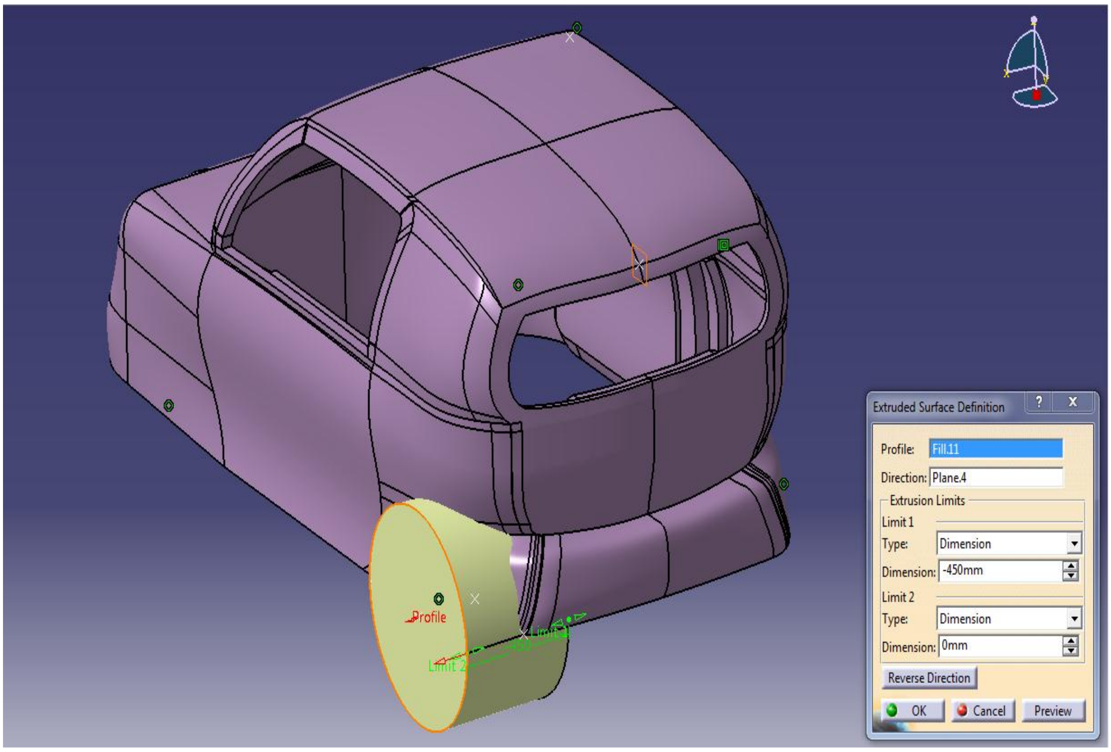
For the design of the auxiliary solid volume the wireframe model was defined first (Picture 4.5.1). The wireframe model was very simple and was formed by a single circle. Then, by using this circle as a reference a filled surface was created. This fill surface covered the gap that existed inside the circle (Picture 4.5.2). Then, by using the current filled surface as a reference, an additional extruded surface was created (Picture 4.5.3). The extrusion's direction was designed to be perpendicular towards the yz plane while its length was determined as 450 mm towards the negative direction of x-axis. The surface's length was defined as 450 mm in order to intersect the vehicle's solid volume. The auxiliary solid volume was finally designed with the execution of a "*close surface*" command on the previously defined extruded surface (Picture 4.5.4). The final solid volume derived by executing two remove volume Boolean operations in sequence. The number of Boolean operations that was used was determined by the number of solid volumes the auxiliary volume was intersected with. As it can be inferred from the picture below, the auxiliary volume intersected two of the vehicle's solid volumes (Picture 4.5.5). The symmetrical hole was created by creating a symmetrical auxiliary solid geometry and executing two sequential remove volume Boolean operations as before (Picture 4.5.6). The remaining two holes that were left to be designed on the front face of the vehicle's body, were constructed with the exact same procedure (Picture 4.5.7).



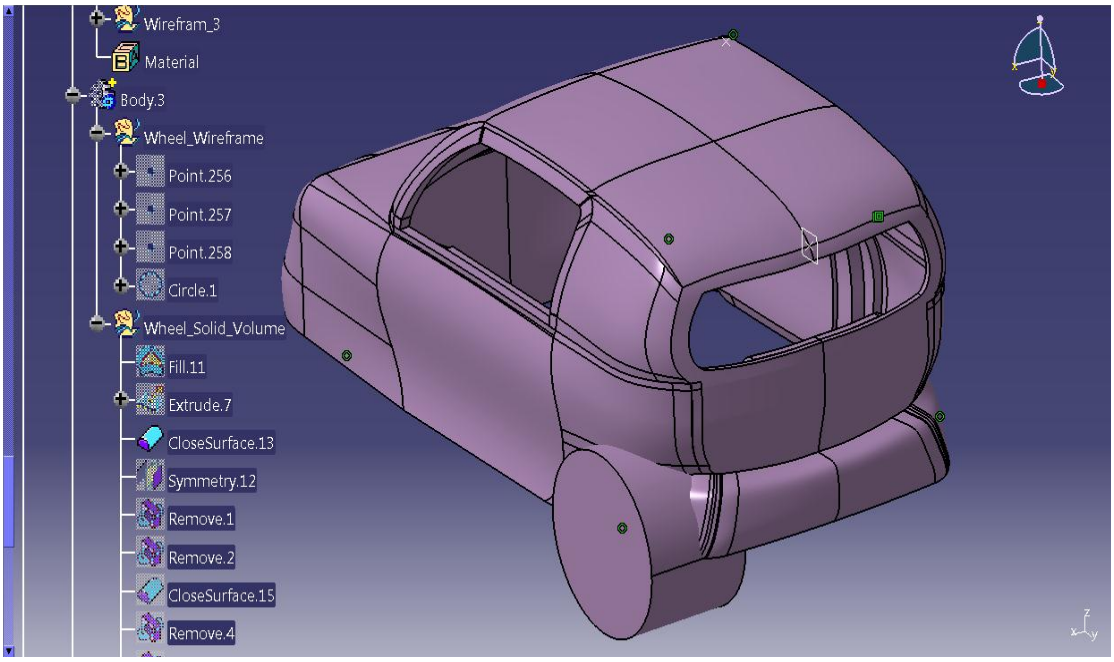
Picture 4.5.1: Creation of tire's hole wireframe model.



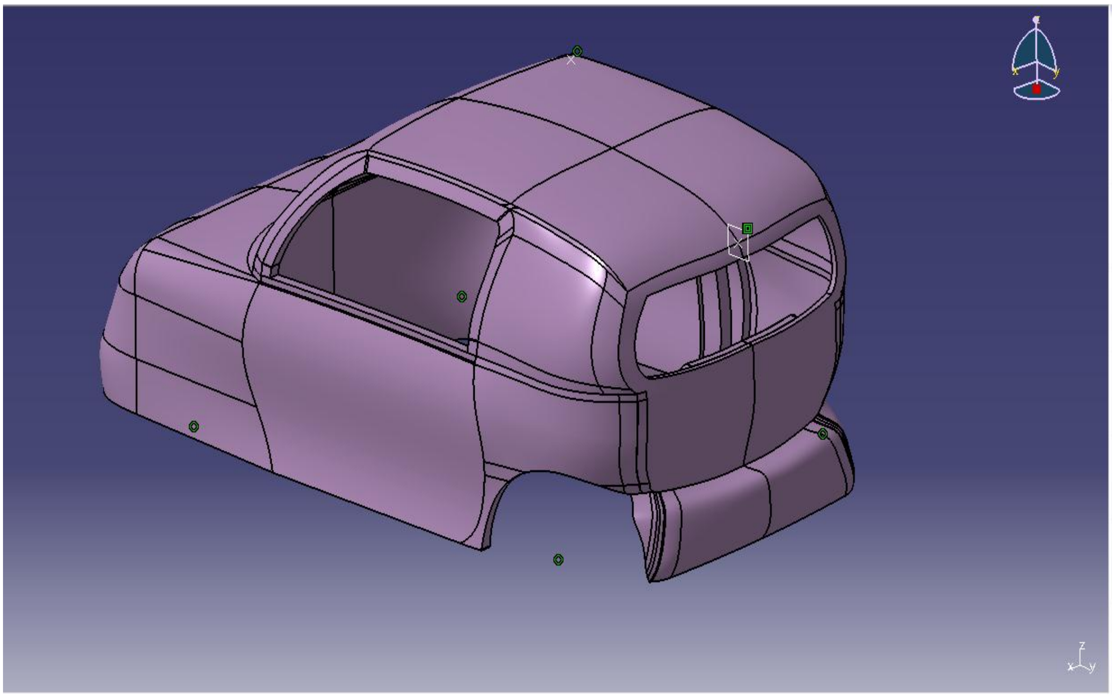
Picture 4.5.2 Creation of surface model from the previous wireframe model.



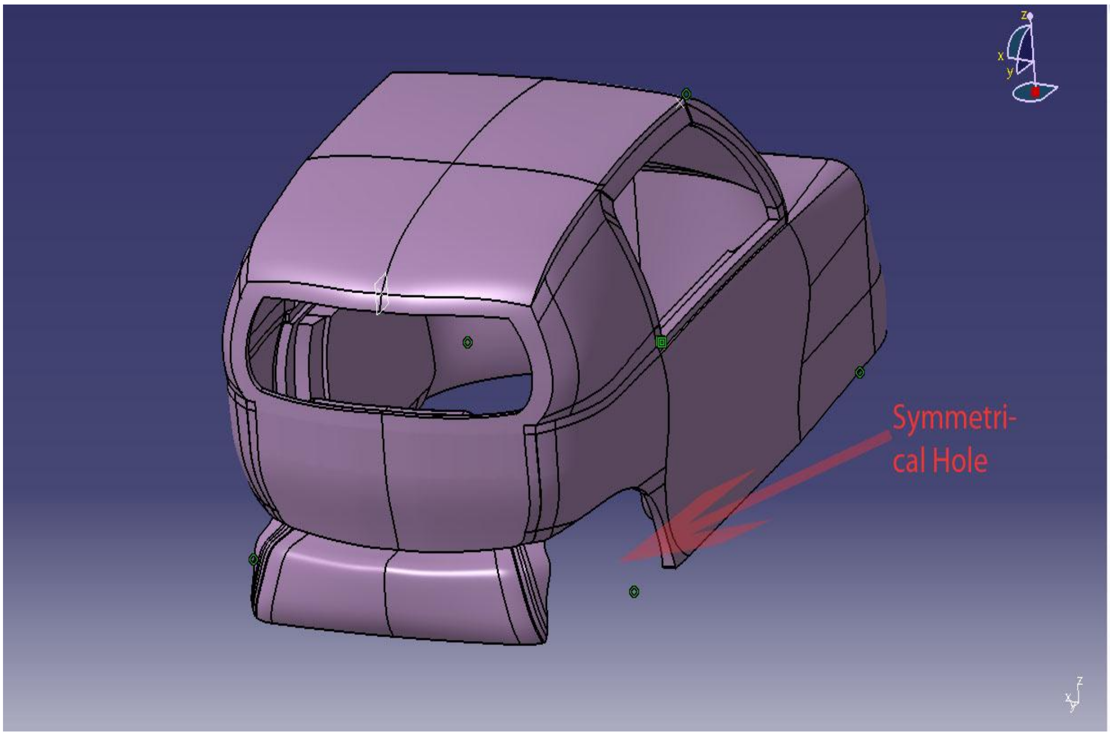
Picture 4.5.3: Creation of surface of extrusion.



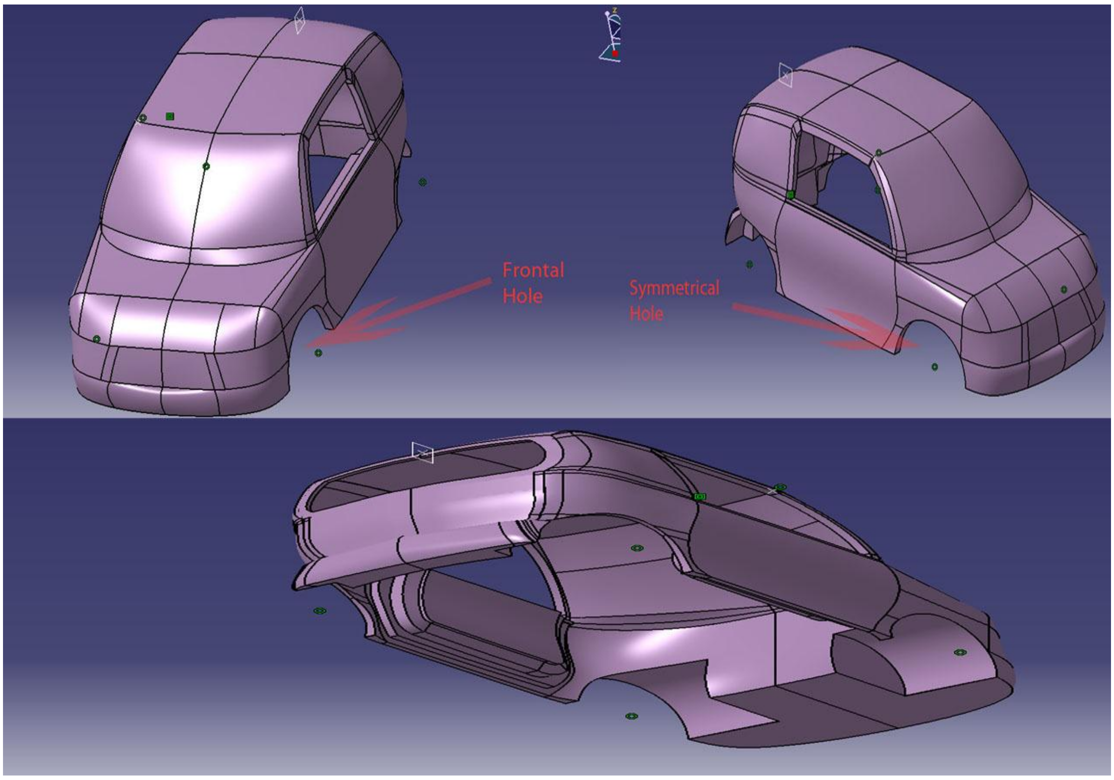
Picture 4.5.4: Creation of solid volume from the previous cylindrical surface model.



Picture 4.5.5: Creation of the socket's hole by executing the "*remove volume*" Boolean operation.



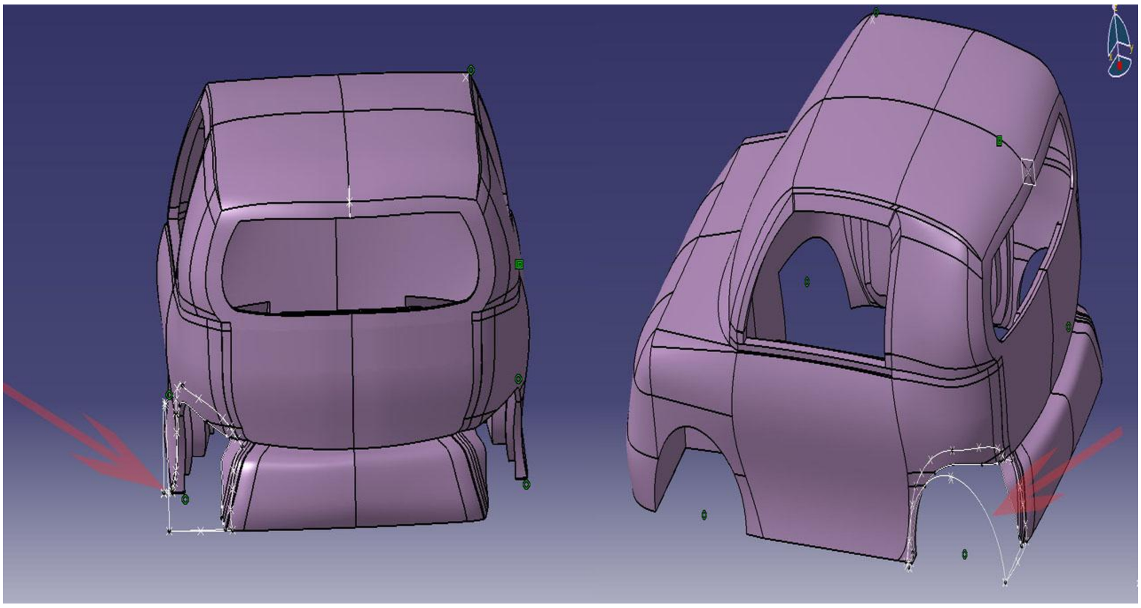
Picture 4.5.6: Creation of the symmetrical socket hole.



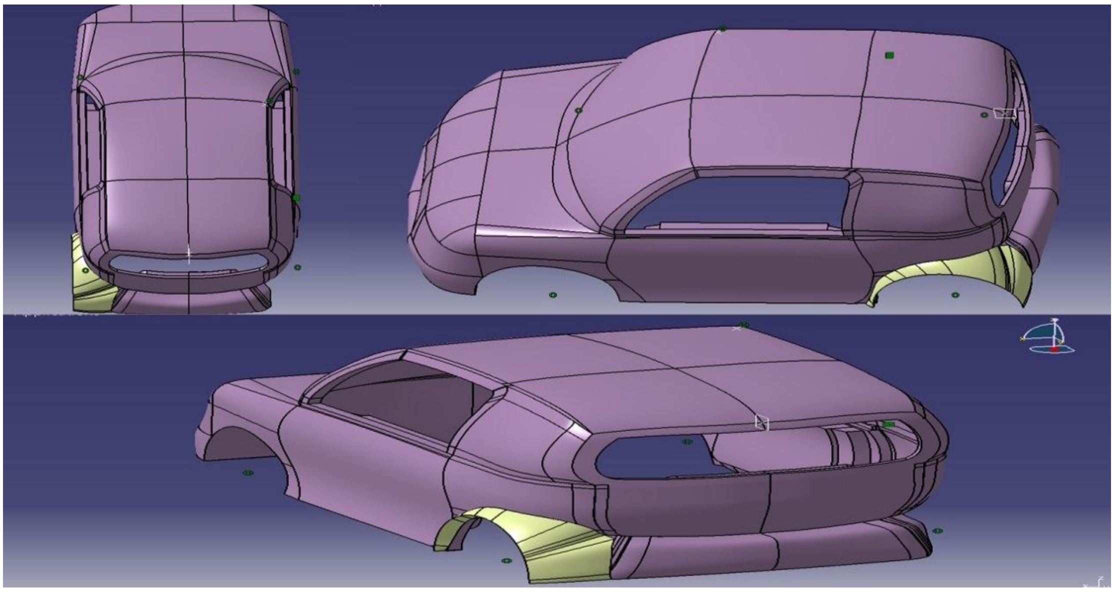
Picture 4.5.7: Creation of the frontal socket holes.

4.6 CREATION OF FENDERS

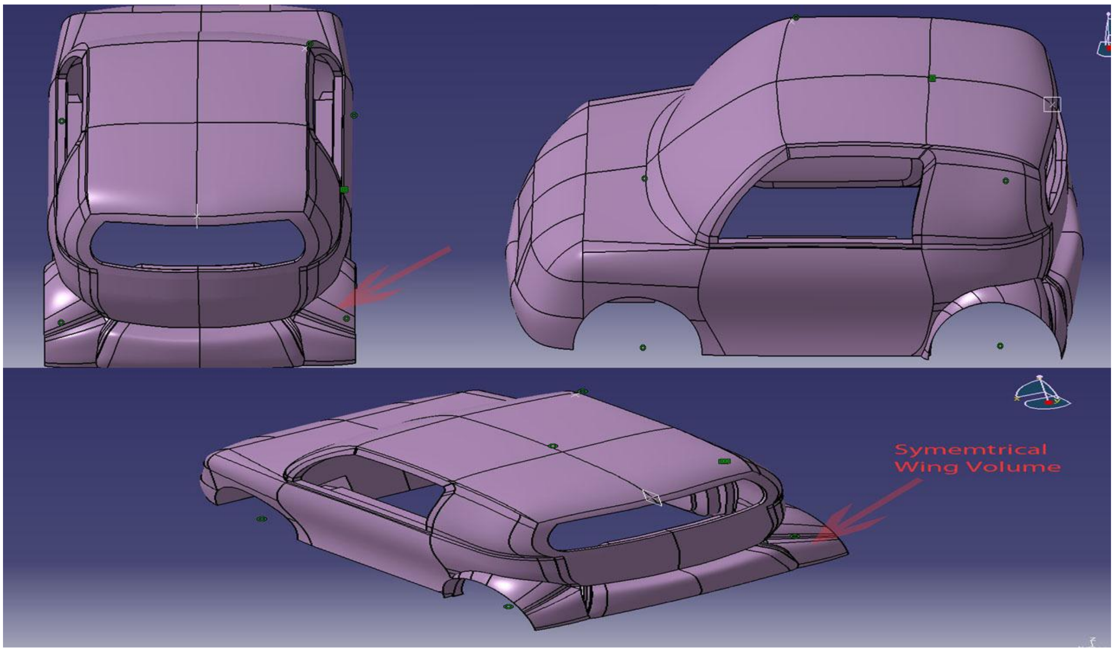
The next volumes that were created were the solid volumes of the fenders. These volumes constituted a cover of the tires' solid volumes. First in the design sequence came the fenders of the back tires. As it can be seen from Picture 4.6.1 below, the fenders' wireframe was more sophisticated than the respective wireframe models of the previous volumes. An important constraint that was followed in the wireframe model was the continuity of the current curves with the rest of the vehicle's solid volume. The model's curves that correspond to the volume's boundary edges derived by executing sequential "*boundary*" commands. Then, by using the current wireframe model as a base, the fender's surface model was created (Picture 4.6.2).This surface model consisted of one multi section surface and two filled surfaces. Subsequently, the set of the three surfaces was unified into a single entity by executing two sequential *"healing"* operations. The fender's solid volume was finally designed after a "*close surface*" operation command (Picture 4.6.3). By executing an additional "*symmetry*" operation command, the symmetrical fender volume was created. As a plane of symmetry inside the command's execution window the plane parallel to the yz plane was selected. By following the exact same methodology, the front tires' fenders were constructed (Pictures 4.6.4 – 4.6.7).



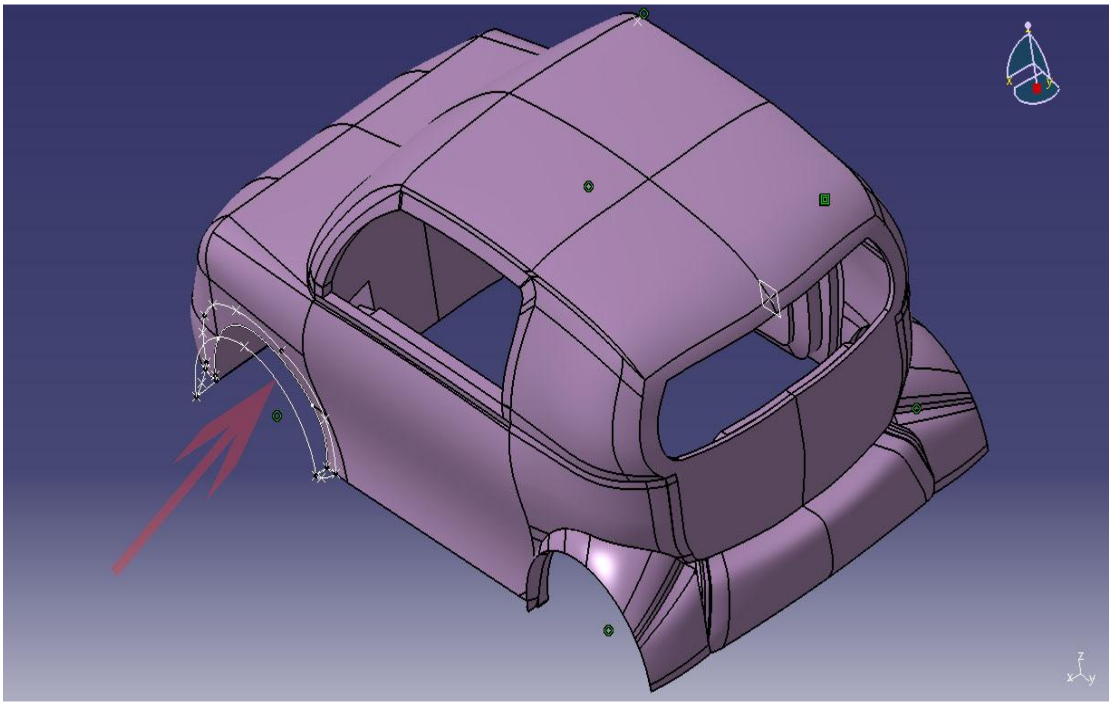
Picture 4.6.1: Creation of the rear fender's wireframe model.



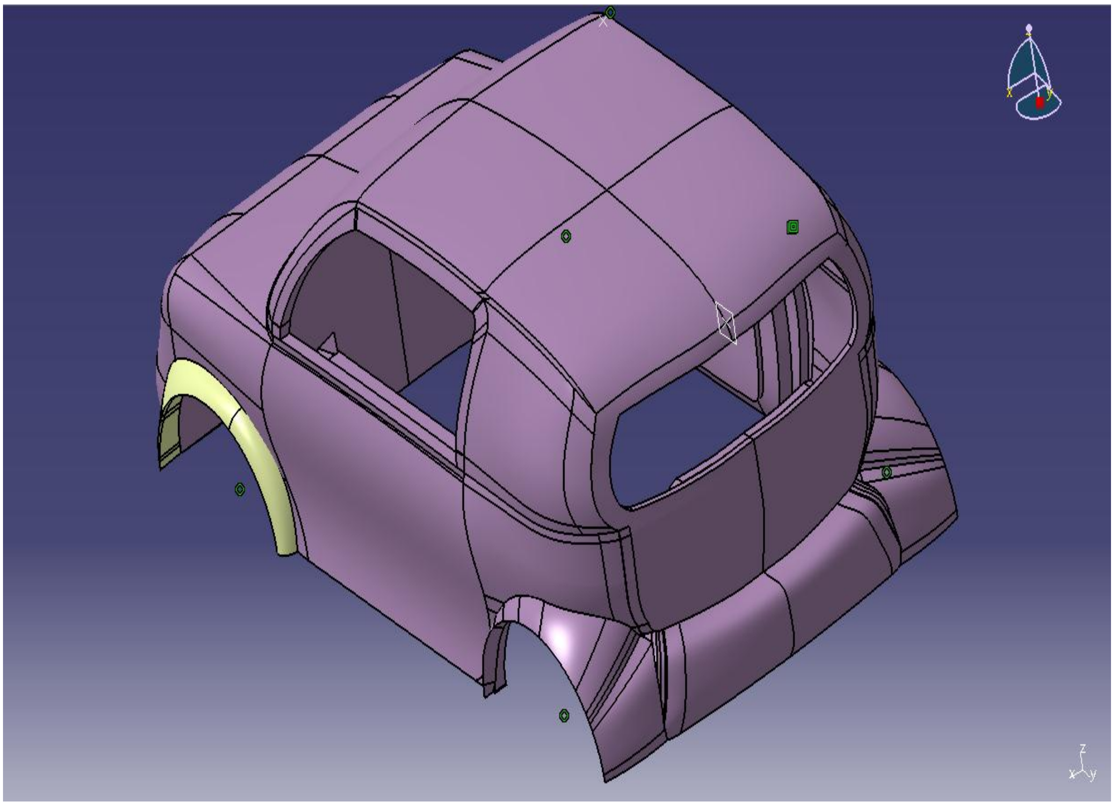
Picture 4.6.2: Creation of the rear fenders' surface model.



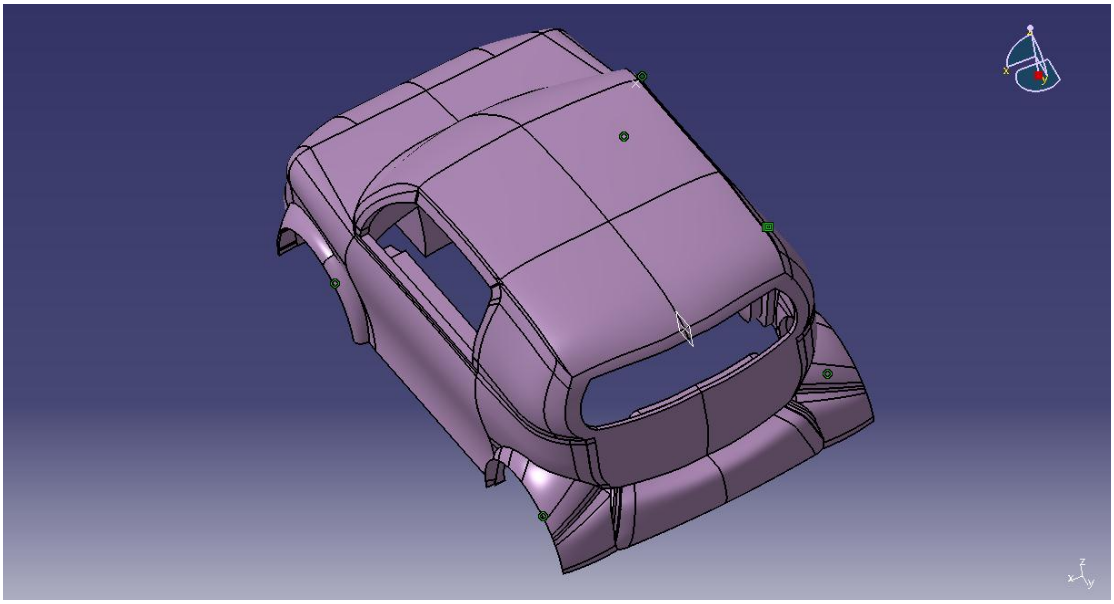
Picture 4.6.3: Creation of the rear fenders' solid volumes.



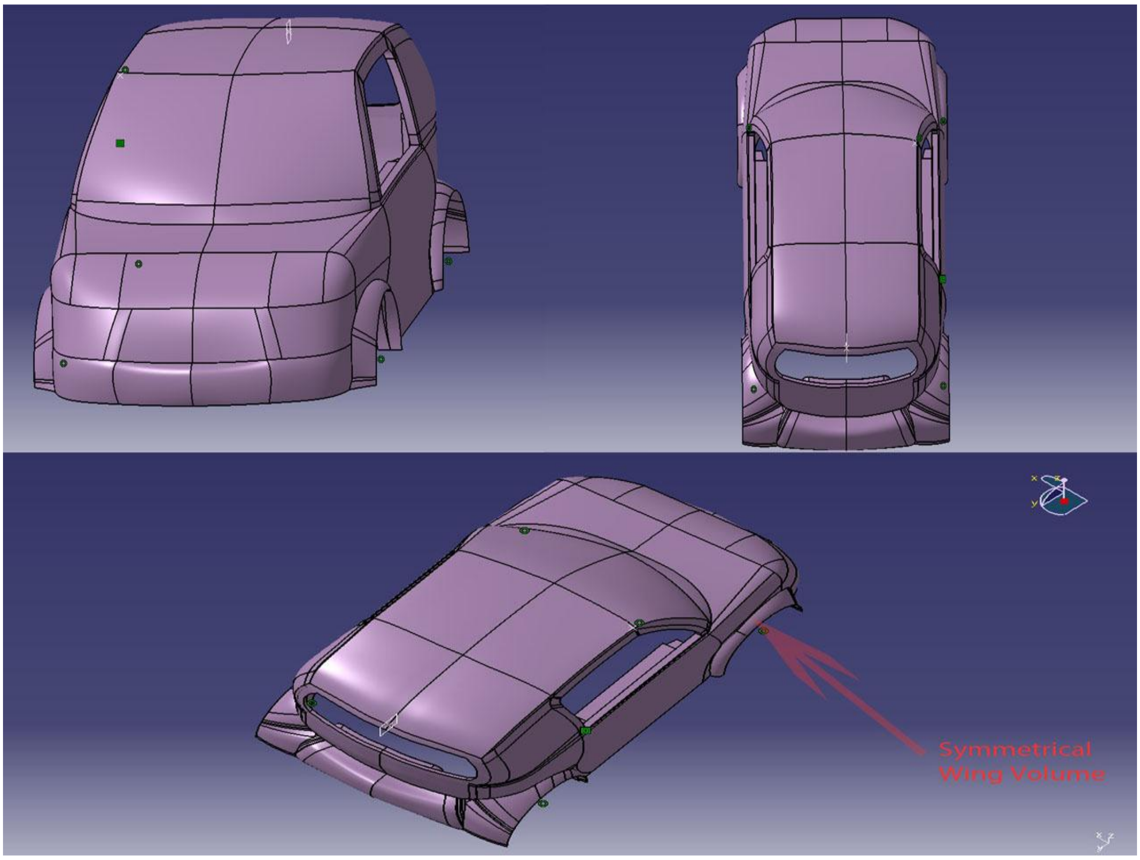
Picture 4.6.4: Creation of front fenders' wireframe model.



Picture 4.6.5: Creation of front fenders' surface model.



Picture 4.6.6: Front fenders' solid volume.



Picture 4.6.7: Creation of the front fender’s symmetrical volume.

4.7 CREATION OF TIRES' CANOPIES VOLUMES

After the definition of the openings on the vehicle's solid volume for the placement of the tires, the required space for the definition of the tires' canopies was also created. The functional purpose of this current volume was to determine the space where the tires are permitted to rotate. The complete solid volume was divided into two independent modules. The first module's solid, resembled the shape of a half – cylinder. The second solid module, resembled the shape of the canopy.

The wireframe of the first module of the back tire was defined first (Picture 4.7.1). This model was simple and consisted of a single semi-circle. This semi-circle was defined using three previously determined points as references. The reference points of the semi-circle arose after using three points which were defined on the solid volume of the vehicle. More specifically, these points were placed on the fender's border edge of the back tire. The precise position of these points is depicted in Picture 4.7.1 below. Then, by using the semi-circular profile as a reference, an extruded surface was defined (Picture 4.7.2). As a direction for the extrusion the perpendicular direction towards the yz axis was defined. The precise length of the extrusion was determined as 280 mm. The functional purpose of the extruded surface will become clear later in the analysis, and more precisely at the stage of the module's solid volume definition. The solid volume of the semi cylinder derived by executing a "*thick surface*" command. This operation was a volume definition command. For the execution of this command a certain number of parameters had to be defined inside the command's definition window. For the ''object to close'' parameter, the previously constructed extruded surface was selected. The total thickness of the derived volume was determined as 30 mm at the first offset option bracket. The direction of the material's addition was defined by an arrow which is shown in Picture 4.7.3 below.

Then, the second module's wireframe model was defined (Picture 4.7.4). This model is more complex than the previous one. It consisted of two concentric semi circles, one closed section, which was created perpendicular to the two semi-circles, and a single line. More precisely, the two semi circles lied on the yz plane and the closed section along with the line on the xy plane respectively. By using the previously defined wireframe model as a reference, a revolved surface was formed (Picture 4.7.5). As generative curve the closed section's profile was defined, while as guide curves the two perpendicular semi circles were selected. As axis of revolution the wireframe's single line was set. The surface model was completed with the creation of an additional fill surface. This surface was created to cover the gap that derived from the formation of the previous revolved surface (Picture 4.7.6). The final canopy's solid volume was formed by executing a "*close surface*" command (Picture 4.7.7). Then, after a "*symmetry*" operation command, the symmetrical solid volume was formed (Picture 4.7.8). The front canopies were created by following the same procedure (Picture 4.7.9).

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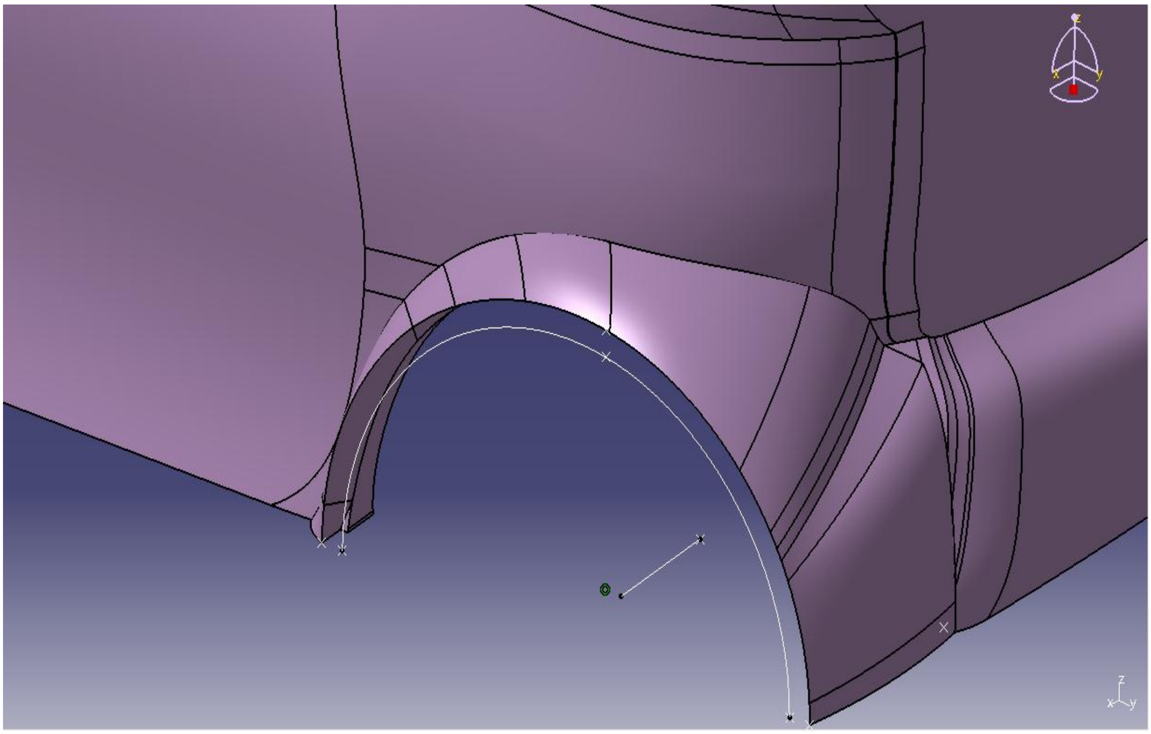
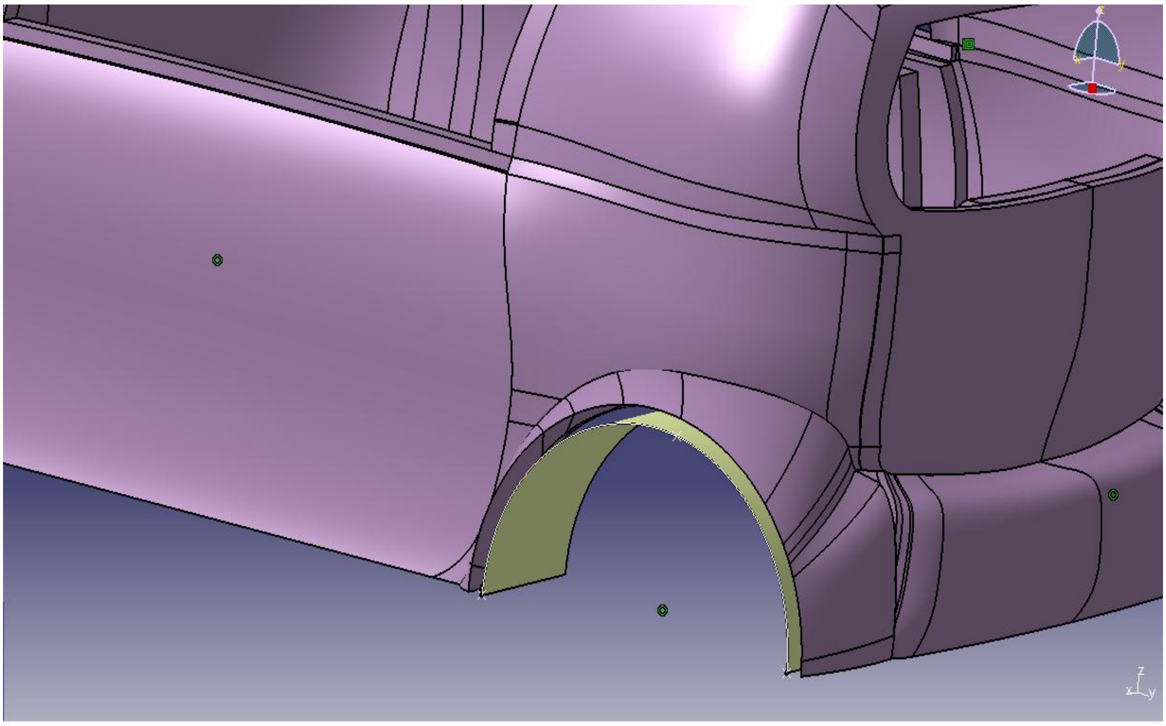
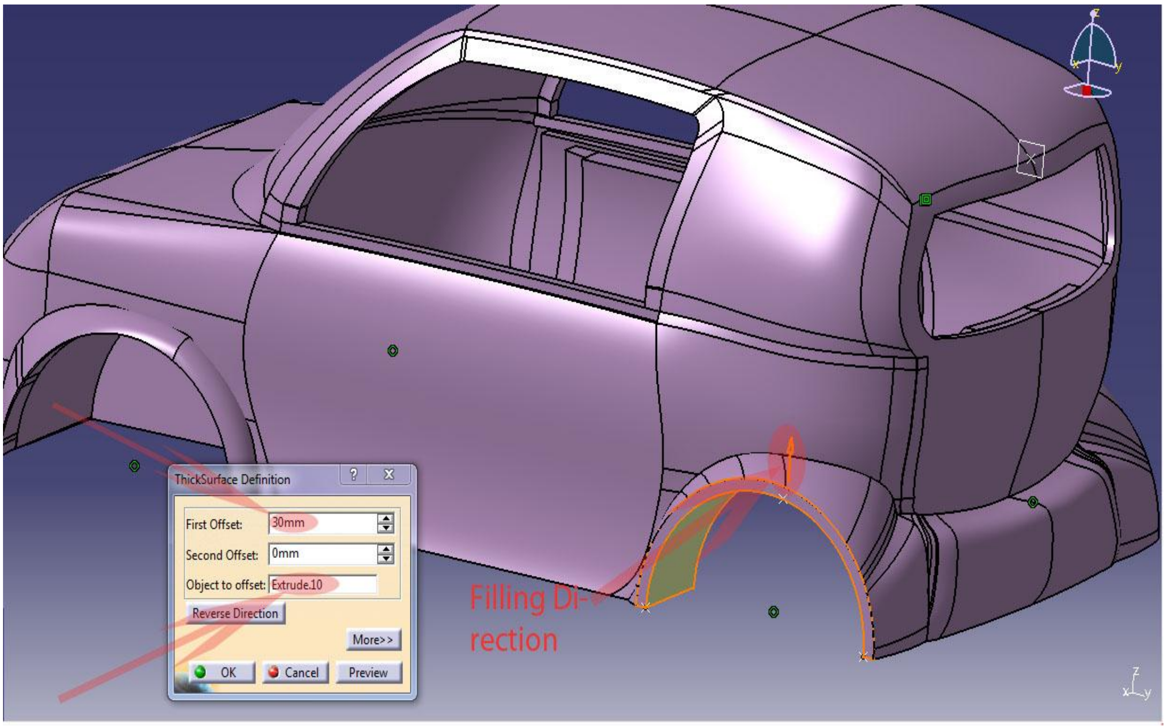


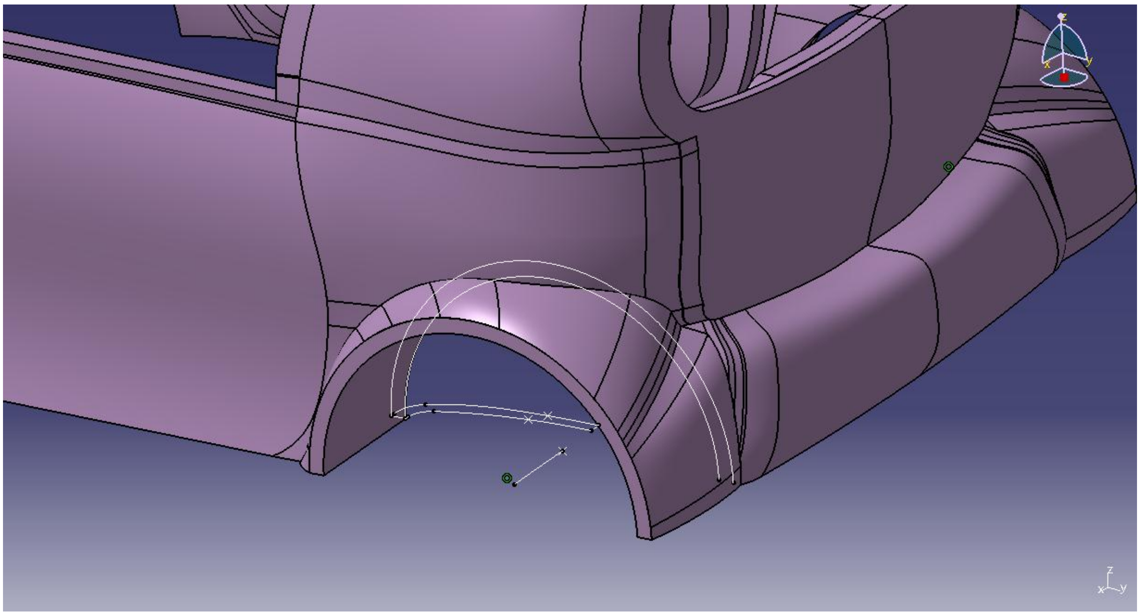
Picture 4.7.1: Tire’s canopy wireframe model.



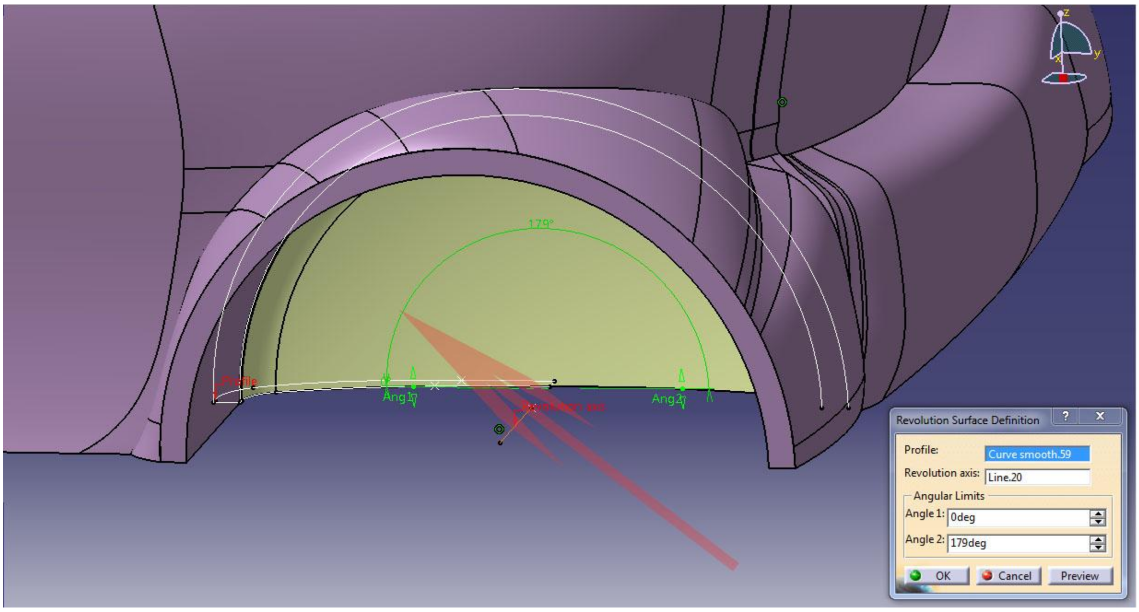
Picture 4.7.2: Creation of the surface of extrusion from the previous wireframe model.



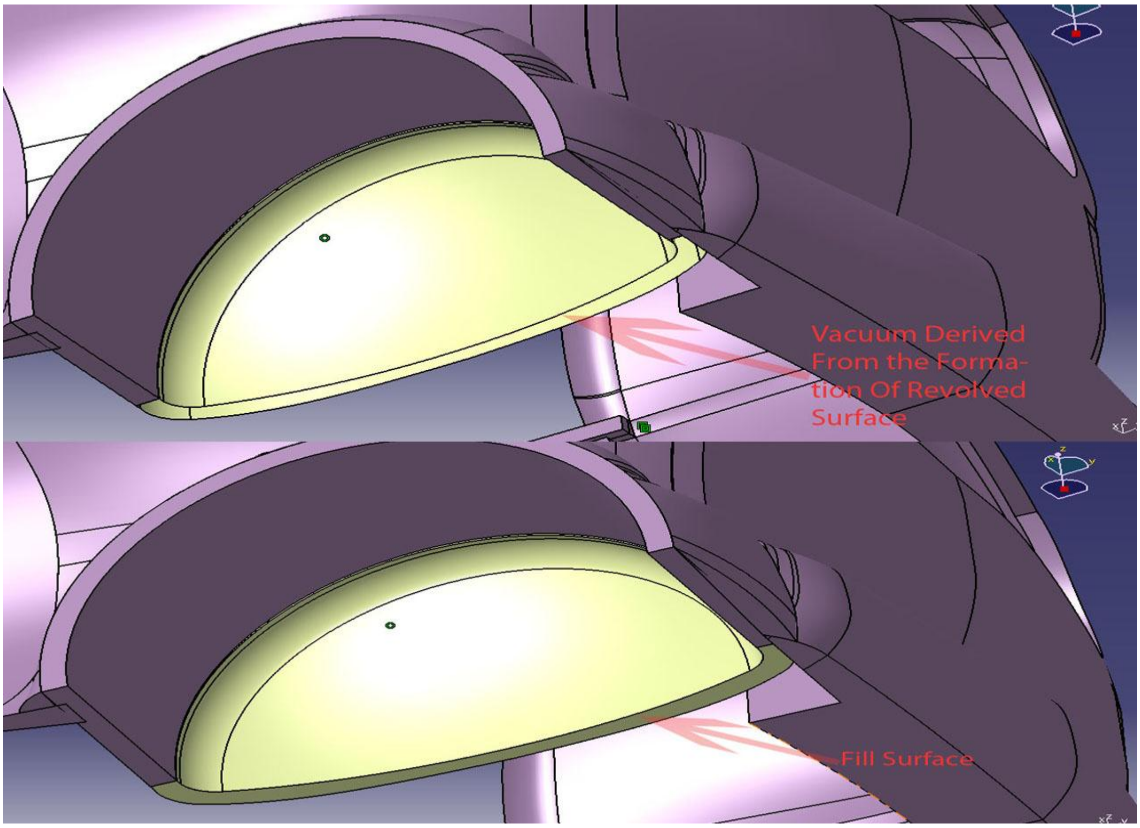
Picture 4.7.3: Creation of the canopy's first module solid volume.



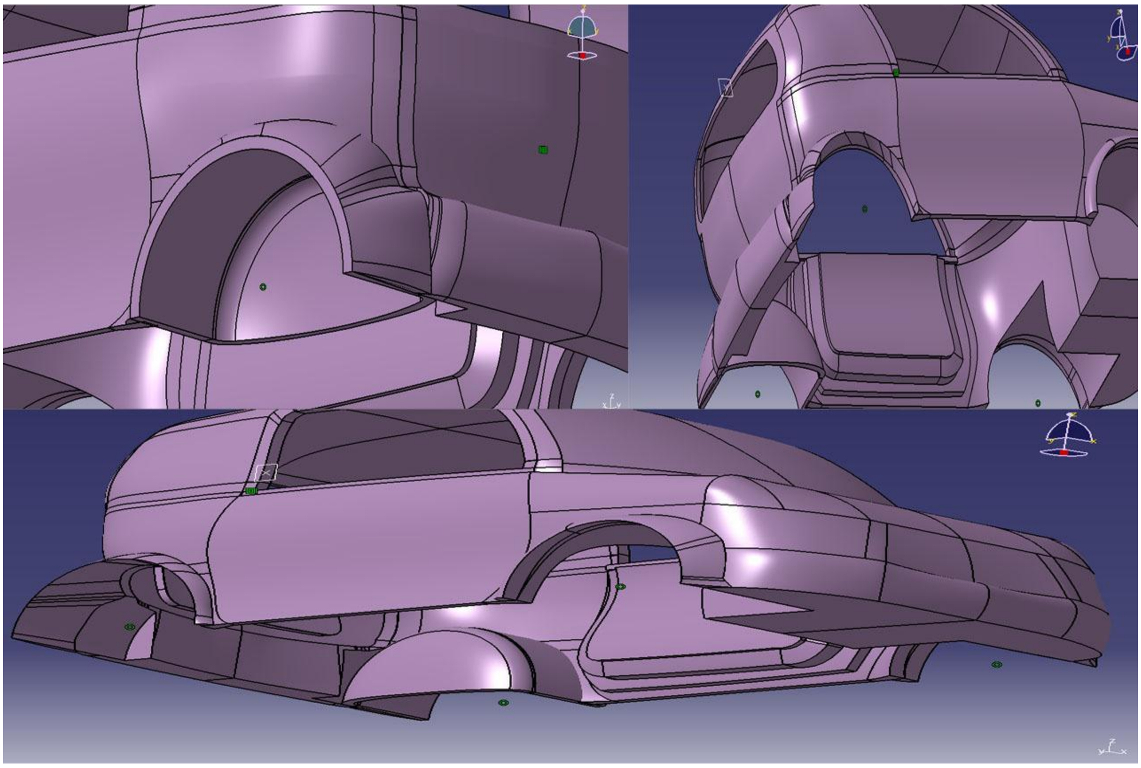
Picture 4.7.4: Creation of back tire's canopy second module's wireframe model.



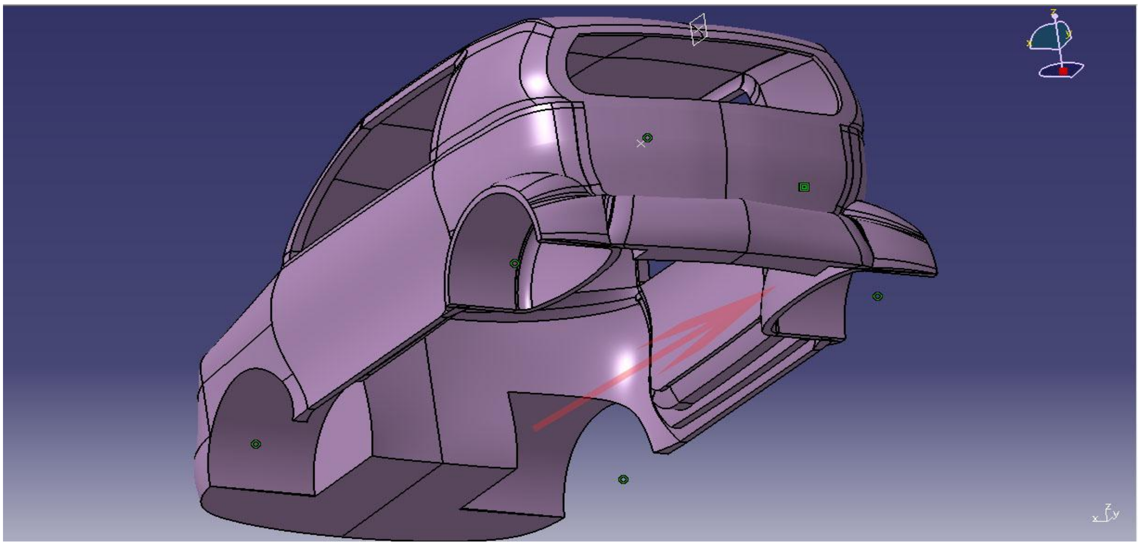
Picture 4.7.5: Creation of canopy's second module surface model.



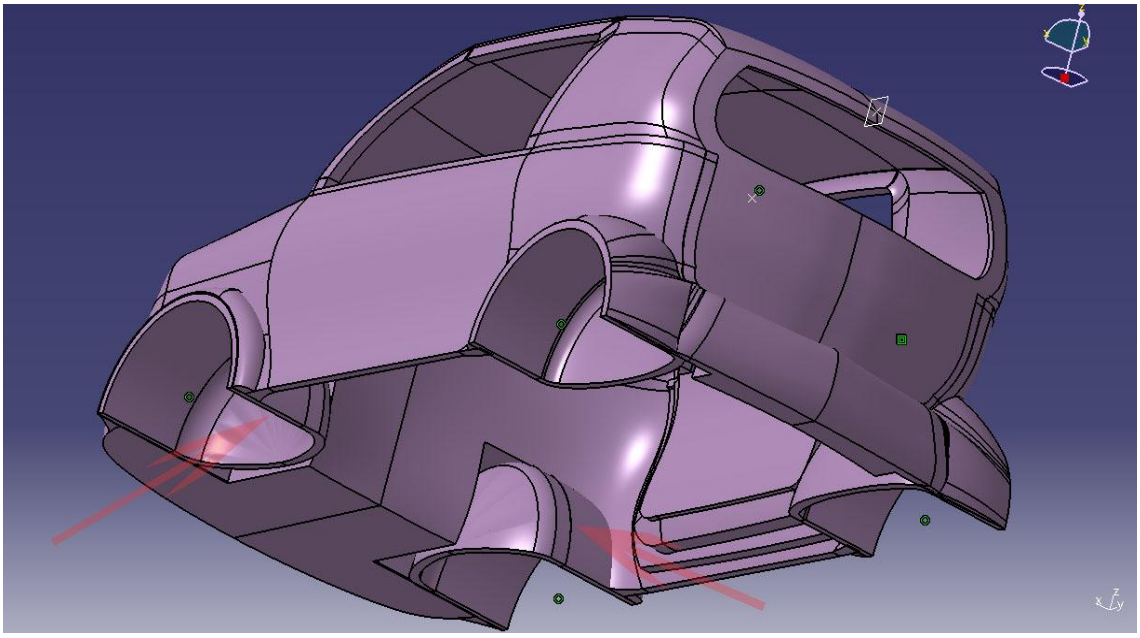
Picture 4.7.6: Creation of canopy's second module's fill surface.



Picture 4.7.7: Creation of canopy's solid volume.



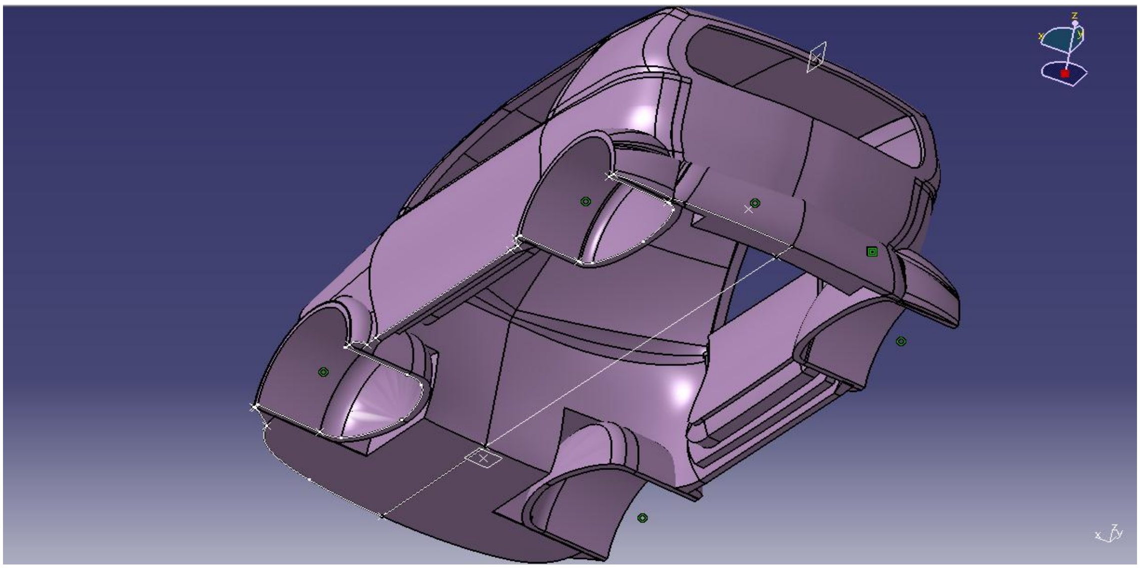
Picture 4.7.8: Creation of canopy's symmetrical volume.



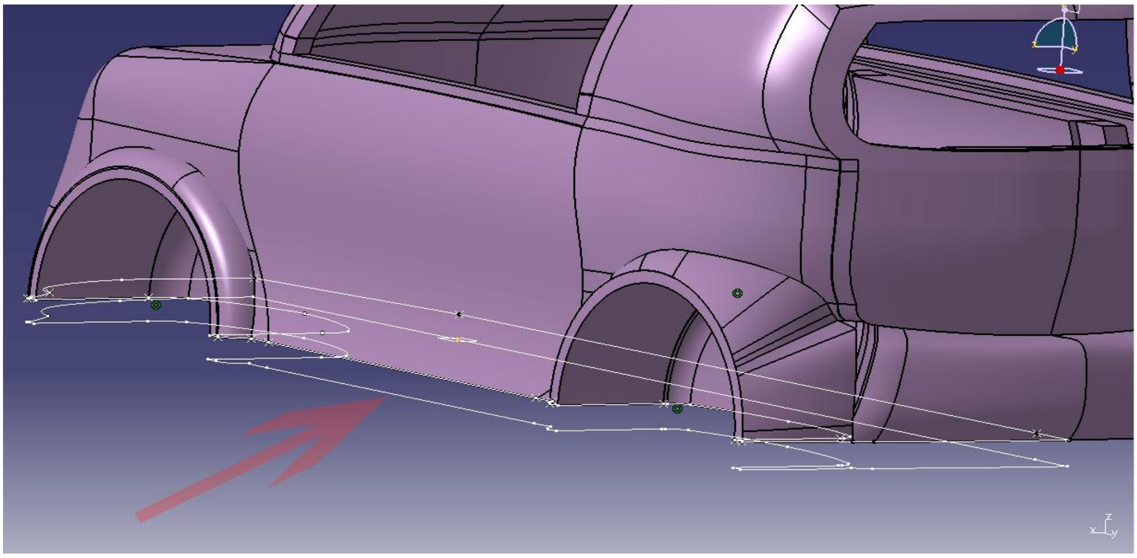
Picture 4.7.9: Creation of front canopies' solid volume.

4.8 DESIGN OF VEHICLE'S FLOORBOARD

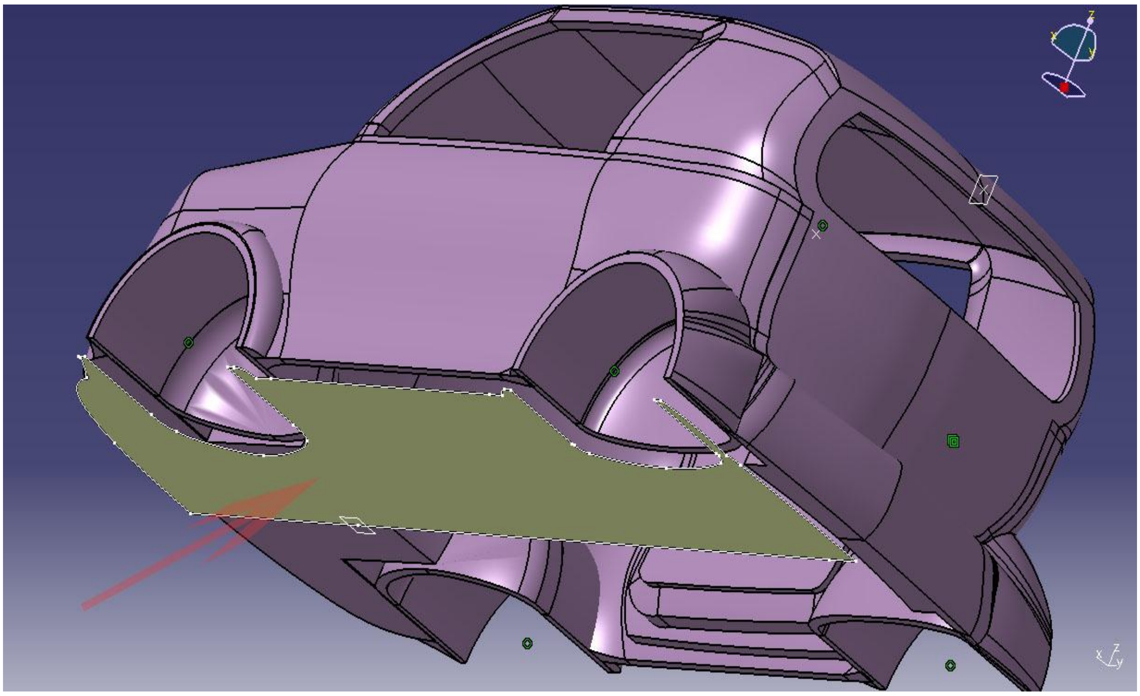
The final volume that remained to be designed in order for the 3-D digital model of the vehicle's external shell to be completed, was the floorboard volume. As with all the previous volumes, the floorboard's wireframe model was created first (Picture 4.8.1). The current volume's wireframe was sophisticated and it was defined so as to follow the vehicle's bottom volume contour as much as possible. For this reason, 18 sequential ''boundary'' operations were executes along with the design of 2 additional Splines. Finally, the bottom's closed contour was formed, with the design of a set of 20 wires in total. At this point it is important to note a specific detail that is necessary for one to understand the necessity of the following operation. This certain wireframe model was not planar. However, the planarity of the wireframe model had to be ensured in order for the surface model to be defined more efficiently. Consequently, the current closed section was projected onto a plane parallel to the software's xy - plane. The previous action was realized with the execution of a ''*projection*'' command. In addition, the wireframe's closed section was covered with a ''*fill*'' surface (Picture 4.8.3). Furthermore, by exploiting the previously designed filled surface, an additional extrude surface was constructed. The extrusion's direction was defined as the perpendicular direction towards the xy - plane. The extrusion's total length was determined 61 mm, slightly intersecting the vehicle's rest volume (Picture 4.8.4). The final solid volume derived from the execution of a "*close surface*" command on the floorboard's surface (Picture 4.8.5). The symmetrical volume was formed by executing a *"symmetry"* operation command. As plane of symmetry a plane parallel to the yz plane was selected (Picture 4.8.6). The vehicle's complete solid volume is depicted in Picture 4.8.7 below.



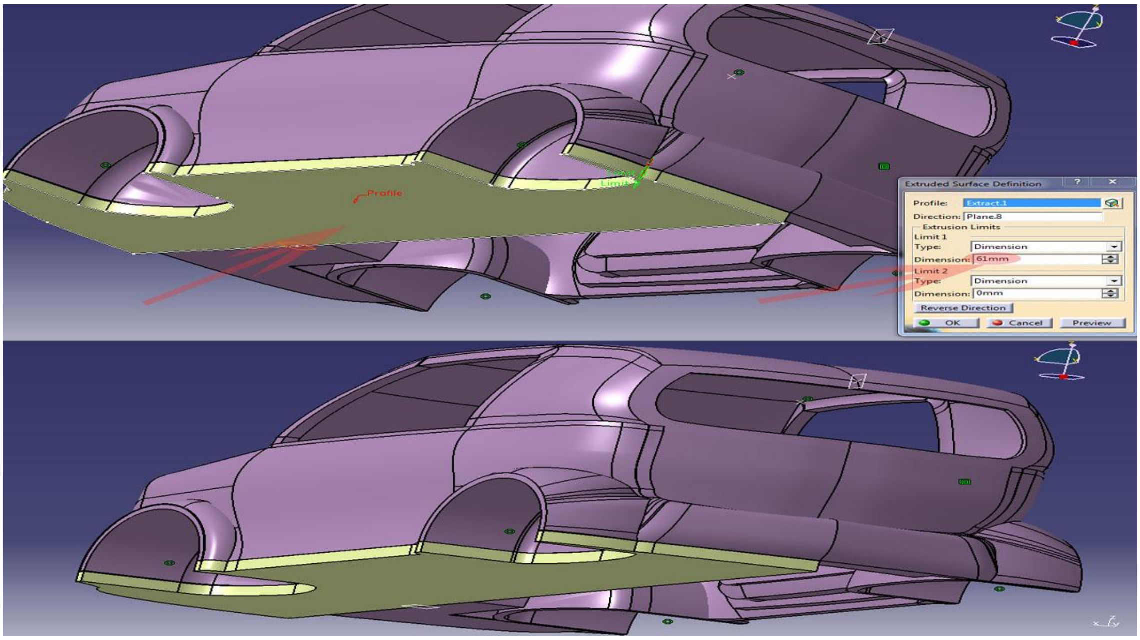
Picture 4.8.1: Definition of floorboard's wireframe layout.



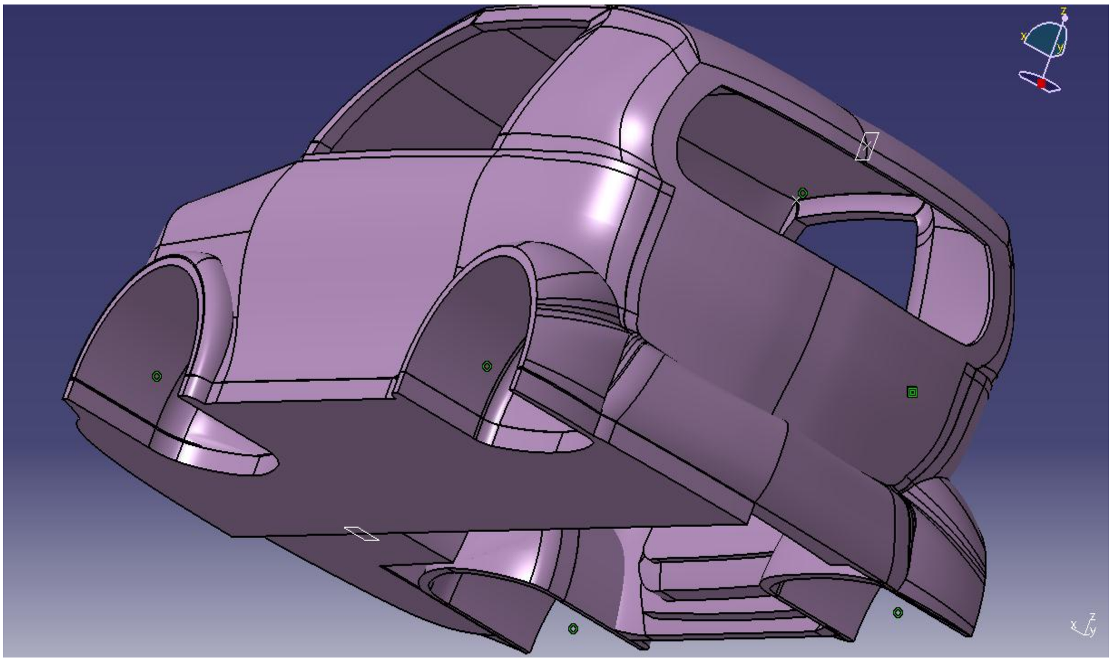
Picture 4.8.2: Creation of floorboard’s wireframe model as a projected section.



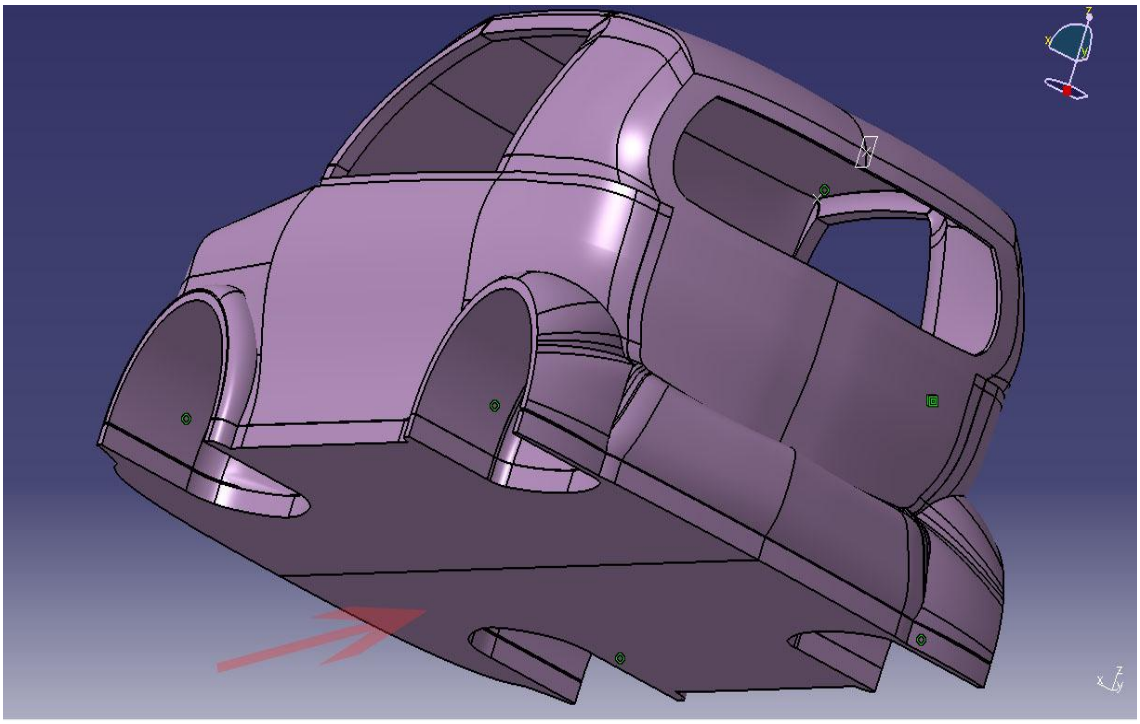
Picture 4.8.3: Creation of a surface model from the previous wireframe model.



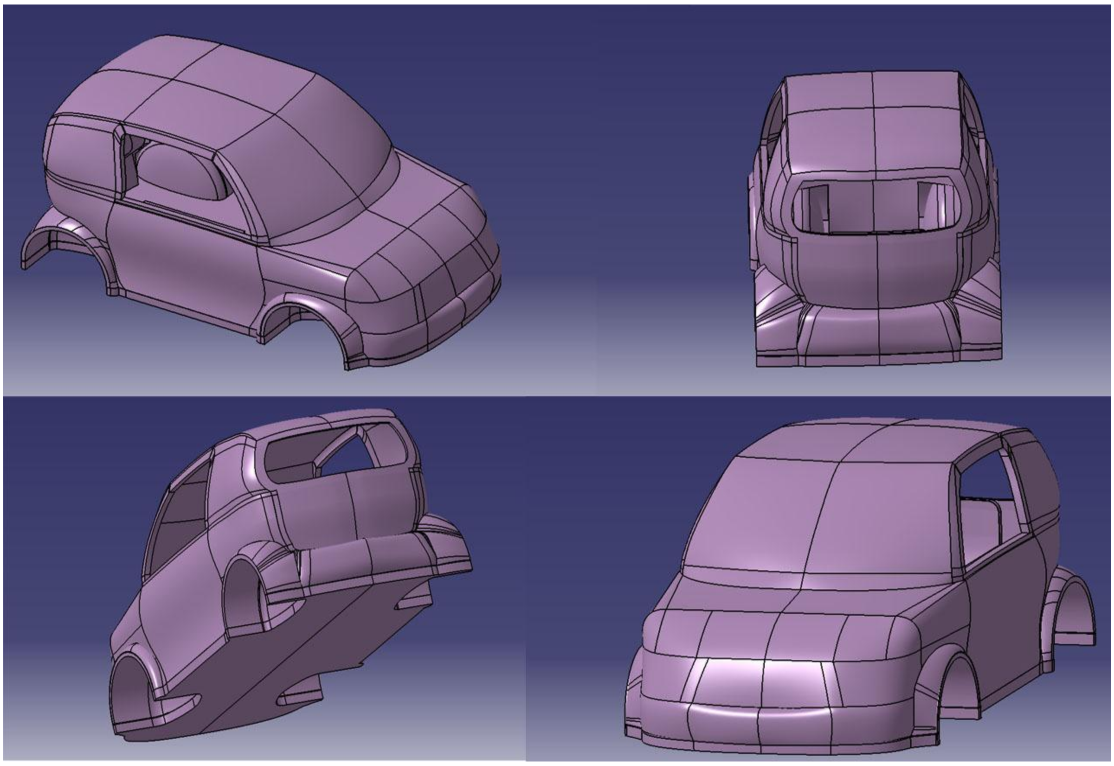
Picture 4.8.4: Floorboard's complete surface model.



Picture 4.8.5: Floorboard's solid volume.



Picture 4.8.6: Floorboard's second symmetrical volume.

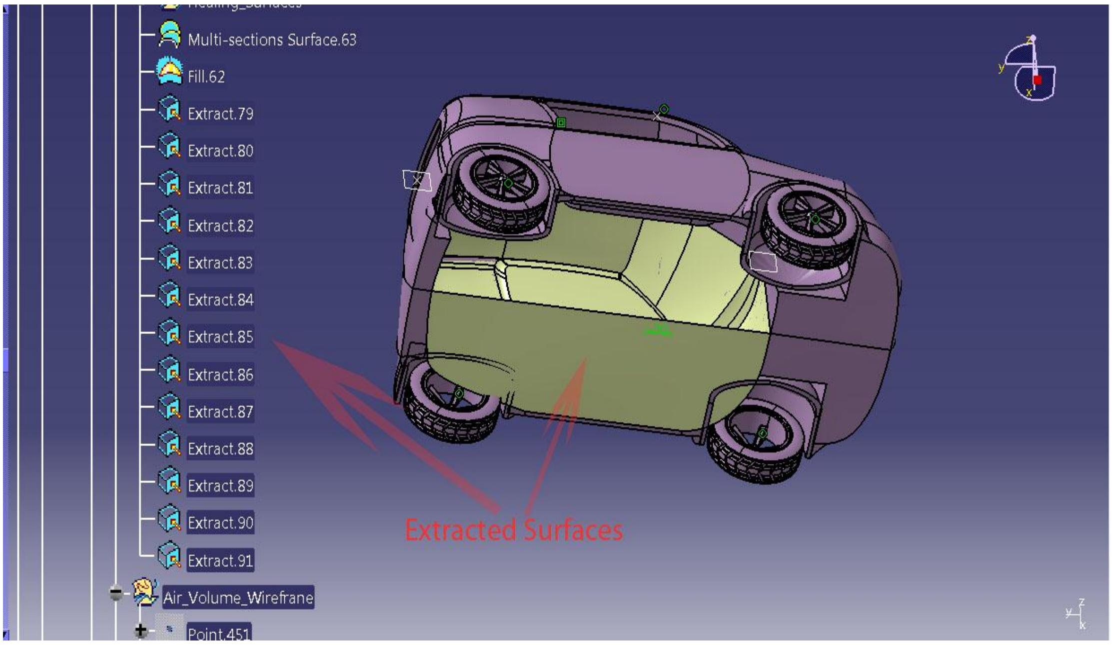


Picture 4.8.7: Complete vehicle's solid body.

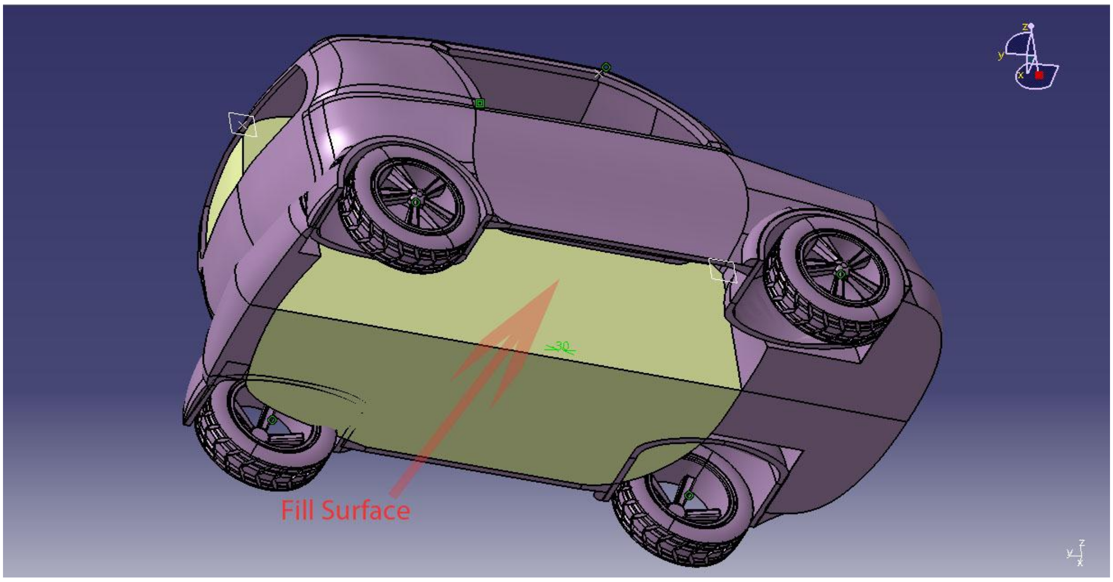
4.9 CONSTRUCTION OF INTERNAL AIR CONTROL VOLUME

This certain volume had to be constructed for the implementation of the aerodynamics analysis of internal air's flow inside the cabin. The air's flow is derived from the function of the vehicle's air conditioning system. As air inlet inside the cabin the dashboard's airduct volumes were designated.

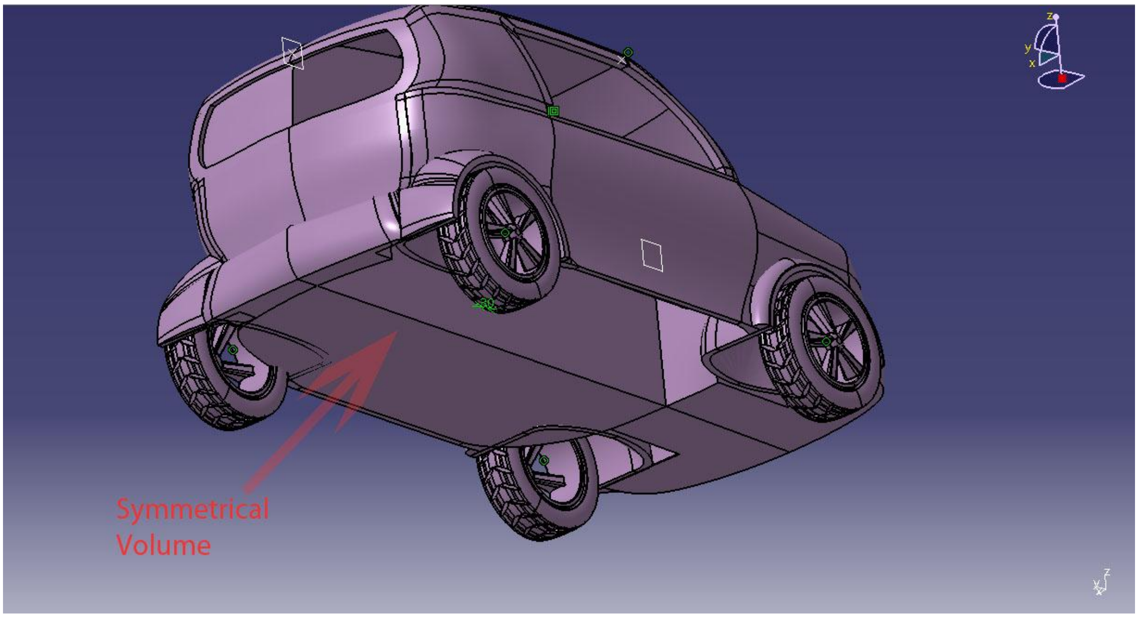
For the definition of the internal air volume inside the vehicle's digital mock up a completely different methodology was followed. Instead of approaching the vehicle's internal volume as a design of an independent volume that resembles the initial and then implement a series of corrections on the independent volume, the vehicle's internal surfaces were used as references for the design of the desired control volume. More specifically, on each internal lateral surface inside the vehicle's cabin an independent surface was determined. The creation of each single surface derived from the execution of the "*extract surface*" operation. Due to the existence of symmetry with respect to yz - plane on the vehicle's solid volume, only half of the vehicle's total lateral surfaces were used as references for the construction of the control volume's external surfaces (Picture 4.9.1). In order to create a closed set of surfaces for the construction of the control's volume first symmetrical half, an additional planar surface was defined. This surface was designed as a fill surface on the plane of symmetry by defining as contour the already existing surfaces' boundary curves (Picture 4.9.2). Next in sequence followed the unification of the previous set of surfaces by eliminating the slight discontinuities that existed between the designed surfaces. This connection was achieved by executing the "*healing*" operation command in sequence. Each healing operation was introduced on the existing surfaces in pairs. Then, by employing the previously designed set of surfaces, the final solid volume was formed by executing the "*close surface*" command (Picture 4.9.3). The second symmetrical half was created into the digital mock up by executing a "*symmetry*" operation command. The final control volume derived by extracting the volumes of the cabin's components from the initial control volume. This procedure was completed by executing sequentially *"remove volume"* Boolean operations. As base object inside the command's definition window the air control volume was consistently inserted while as volume to be removed each of the components' volume was introduced. The vehicle's internal air control volume is depicted in Picture 4.9.4 below. This volume can be used as a basis for the simulation of the air-conditioning flow inside the cabin by means of Computational Fluid Dynamics software.



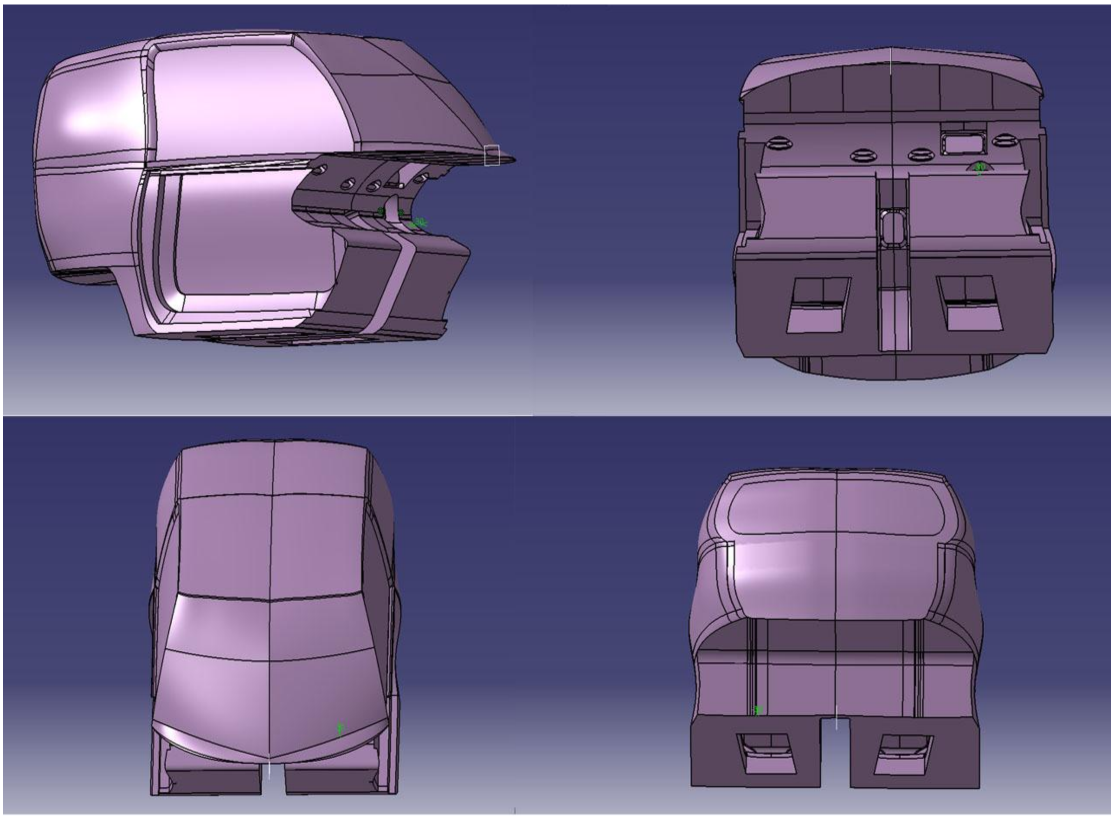
Picture 4.9.1: Internal air surfaces of control volume.



Picture 4.9.2: Creation of filled surface.



Picture 4.9.3: Creation of internal air control volume first symmetrical half.



Picture 4.9.4: Creation of internal air final control volume.

REFERENCES

[1] Catia V5R18 Tutorials Disc 1-4

[2] Catia V5R14 Tutorials Disc 1-5

[3] Morello, Rossini, Pia, Tonoli - The Automotive Body Volume I Components Design

[4] Morello, Rossini, Pia, Tonoli - The Automotive Body Volume II System Design