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Solid Modeling in Biomechanics and Engineering: SolidWorks-Based Interdisciplinary Design and Applications

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PREFACE

Unexpected challenges and problems continuously emerge throughout the course of life. This phenomenon is not limited to a specific field but manifests across all areas of discipline. The resolution of such problems has been achieved through imagination and holistic approaches to development. In this context, engineering assumes the responsibility of producing functional and practical solutions to the problems encountered. These solutions are shaped through discussions and brainstorming processes specific to each field of study. The design process, initiated based on the draft model, becomes tangible through solid modeling. At this stage, what stands before you as a response to the problem at hand is a solid model design. In engineering, before employing theoretical and mathematical approaches to problem-solving, it is essential to present or develop a representative design of the issue being addressed. This allows for more qualified and precise evaluations of the modeled problem. The subsequent stage involves analytical analysis, which may include mechanical, thermal, electromagnetic, or optical evaluations of the designed system or component. Therefore, the solid modeling of a part or system is understood to rely on multidimensional and detailed mathematical foundations. Although the design and solid modeling process constitutes the initial stage of a problem or project, conducting these processes with a multidisciplinary perspective offers significant contributions to optimizing the overall workflow. The common ground among various scientific disciplines lies in design and solid modeling, with mechanical engineering being at the forefront. Additionally, other engineering branches such as automotive, aerospace, civil, industrial, marine (shipbuilding), materials, energy, agricultural, and electrical–electronics engineering make extensive use of design and modeling practices. Beyond engineering disciplines, design and solid modeling have also become integral to health sciences. For instance, in dentistry, these applications are seen in dental implants, endodontic instruments, and prostheses, while in orthopedic medicine, they are utilized in prosthetic designs for the knee, arm, and skull. In cardiology, stents and other life-supporting prostheses are modeled and developed through engineering collaboration. Furthermore, reverse engineering applications also benefit greatly from solid modeling methodologies.

This book presents a step-by-step visual approach to the detailed solid modeling processes of components used across various scientific disciplines. In doing so, it provides practical and instructive examples for individuals trained in, or seeking to develop themselves within, different fields. Brief explanations accompanying each model clarify their real-world applications, serving as a guide for designers. Solid models are created using various CAD (Computer-Aided Design) software and stored in different file formats. The most commonly used formats include IGES, STL, DWG, STEP, SLDPRT, CATPART, PRT, and IPT. The solid models featured in this book were created using SolidWorks 2022, which was selected for its user-friendly interface and efficient modeling capabilities. Unlike traditional design manuals, this book not only explains how to model but also elaborates on where and why each model is used from a scientific and academic perspective. In particular, for readers interested in biomechanical design, the explanations of prosthetic and implant models aim to provide a comprehensive understanding of their structure and engineering background.

ACKNOWLEDGEMENTS

I would like to express my sincere gratitude to my beloved family, who have never withheld their support despite all challenges. I am deeply thankful to Assoc. Prof. Dr. Fahri SARAÇ from Süleyman Demirel University, who has always been by my side regardless of distance; to my colleagues Assoc. Prof. Dr. Melih YILDIZ and Dr. Fethi ŞERMET; and to my esteemed friend, Dr. Mustafa HAMAMCI.

INTRODUCTION

Biomechanics emerges as a multidisciplinary field that investigates the interactions between mechanical loads, constraints, and contact conditions with living tissues [1, 2]. The number of innovative approaches in prosthetic and implant applications developed and validated based on biomechanical principles has been steadily increasing, both in scientific research and commercial practice [3-5].To examine the mechanical interaction within biological tissues, biomechanics advances on the theoretical foundation of mechanics and employs finite element stress analysis (FEA) in a numerical environment before conducting experimental studies [6]. For such stress analyses, a physical solid model representing the problem is required. Prior to performing damage evaluations, the creation of a detailed and accurate solid model that faithfully represents the problem is essential. The implementation of solid modeling in modern engineering does not merely represent geometric visualization; it forms the fundamental basis of analysis, optimization, and manufacturing processes. Through the integrated design chain, a model created via Computer-Aided Design (CAD) software can subsequently undergo Computer-Aided Engineering (CAE) analyses (e.g., static, thermal, fluid, fatigue, or vibration) and ultimately be transferred to production via Computer-Aided Manufacturing (CAM) systems [7, 8]. This comprehensive chain process often termed the "design–analysis–manufacturing cycle" significantly enhances industrial efficiency. Among the primary CAD-based modeling tools used in engineering are SolidWorks, CATIA, Siemens NX, and Autodesk Inventor, though various alternative modeling platforms also exist. These software packages provide high precision in modeling complex geometries and integrating them into engineering analyses. In particular, SolidWorks has gained widespread adoption across industry and academia due to its user-friendly interface, parametric modeling capabilities, and adaptability to biomechanical systems.

In biomechanics, solid modeling applications require the support of specialized software. The biomechanical modeling process does not begin directly with a solid model; instead, it starts by obtaining patient-specific CT data, which are processed into STL (stereolithography) format as point-cloud representations. These imaging datasets are then refined and processed using programs such as Geomagic or Mimics to define anatomical regions, such as cortical and trabecular bone structures.

The resulting models are first converted into surface models, which are later imported into CAD software (e.g., SolidWorks) to be transformed into final solid models. Subsequently, the appropriate prosthesis or implant design is assembled with the target bone or tissue model within these CAD environments. The fully developed solid models are then analyzed under simulated biomechanical boundary conditions including applied loads, moments, and supports using finite element analysis software such as ANSYS Workbench, MSC Nastran, or Abaqus. Through these analyses, the stress distributions, deformation patterns, and fatigue life of the bone–implant or bone–prosthesis system is comprehensively evaluated. Therefore, solid modeling serves not merely as a design practice but also as a scientific tool for investigating biomechanical behavior.

In the present study, solid models of the most commonly used implants and prostheses in the human body have been designed and presented in detail. These include dental implants, dental prostheses, knee prostheses, hip prostheses, intravascular stents, and fixation plates used for femoral fractures. Each model was designed through a stepwise and detailed process, accompanied by explanations of their intended clinical applications and material selections relevant to their manufacturing.

1. *DETAILED DESIGN AND SOLID MODELING OF THE DENTAL IMPLANT*

The initial concept of dental implants traces back to the pioneering work of Brånemark and colleagues, who established the structural and functional relationship between healthy bone tissue and the implant surface [9]. The primary objective of dental implants is to serve as an alternative treatment modality aimed at resolving the complications associated with partial or complete tooth loss [10]. Indeed, numerous studies have reported approximately 95% success rates over a 5-year period, strongly supporting the clinical reliability of implant-based treatments [11]. As research on dental implants expanded, various design geometries and configurations were developed to optimize clinical outcomes [12]. Among these, the most commonly employed geometries are cylindrical and threaded (screw-type) implants [13]. Dental implants can be placed in both the mandible (lower jaw) and maxilla (upper jaw), and are typically fabricated from biocompatible materials, most commonly titanium and its alloys, due to their superior osseointegration capability and corrosion resistance [14, 15]. Once the implant is surgically inserted into the jawbone, the biological process of osseointegration begins [16, 17]. This process represents the direct structural and functional connection between the implant surface and the surrounding bone tissue. In other words, osseointegration refers to the direct, functional, and stable contact formed between the implant and the alveolar bone [18, 19]. The long-term success and biomechanical stability of dental implants are influenced by key mechanical parameters stress and strain [20]. Stress–strain analysis encompasses a set of methodologies used to evaluate the internal forces (stress) and deformations (strain) within a structure or material under external loading conditions [21]. In clinical practice, the most frequently encountered mechanical complications following implant placement include screw loosening and fracture, often associated with excessive loading [22]. Therefore, prior to clinical application or production, it is crucial to conduct damage and failure analyses of dental implants using both experimental and numerical approaches. Among experimental methods, photoelastic analysis and strain gauge measurements are widely used, while in numerical studies, the finite element method (FEM) remains the most prevalent analytical technique [13, 23]. Accordingly, to accurately assess the biomechanical performance of dental implants, it is first necessary to construct precise and detailed solid model representations of the implant systems.

1.1. Design Model of the Dental Implant

For the design of the planned dental implant, a part file is first created in the solid modeling software. At the beginning of the study, this part file is named according to the planned project; in this case, we will refer to it as "dental implant." To create the intended geometry, the "front plane" is initially selected from the feature tree section located on the left panel of the modeling interface, and a sketch window is generated from this plane. These procedures are illustrated in Figure 1-1. Subsequently, within the opened sketch window, the initial two-dimensional (2D) outline of the planned dental implant model is drawn (Figure 1-2). Since the solid model will later be generated by revolving the sketch, only half of the implant geometry is drawn in 2D. In the next stage, in order to convert the defined 2D sketch into a solid model, the drawing file is selected, and the 'Revolve Boss/Base' feature is accessed from the "Features" tab. At this point, the program prompts the user to define the axis of revolution. After completing the 2D sketch of the dental implant in the CAD environment, the subsequent procedures for converting the design into a solid model are illustrated in Figure 1-3.

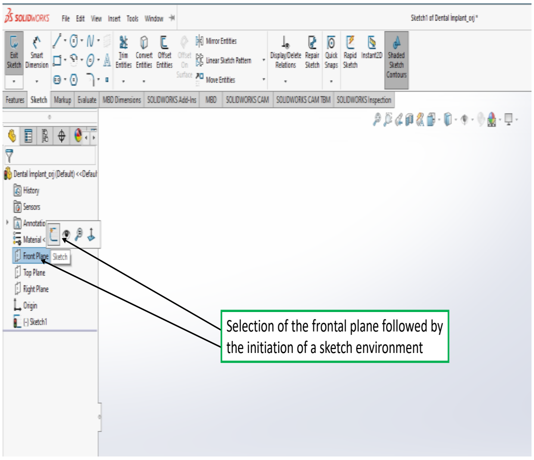


Figure 1-1. Selection of the front plane and creation of the sketch window

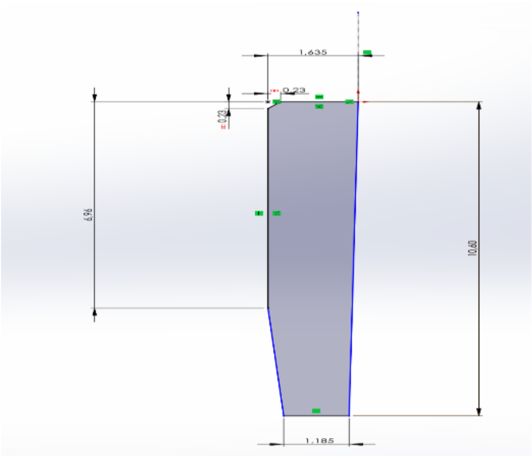


Figure 1-2. 2D sketch of the dental implant geometry

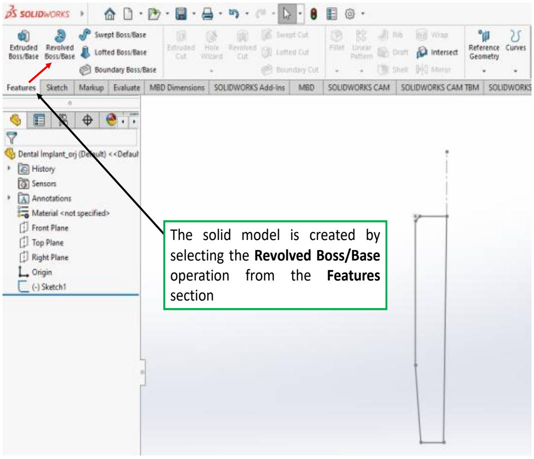
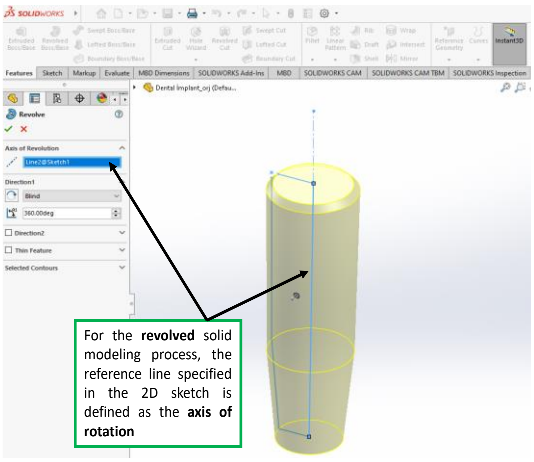


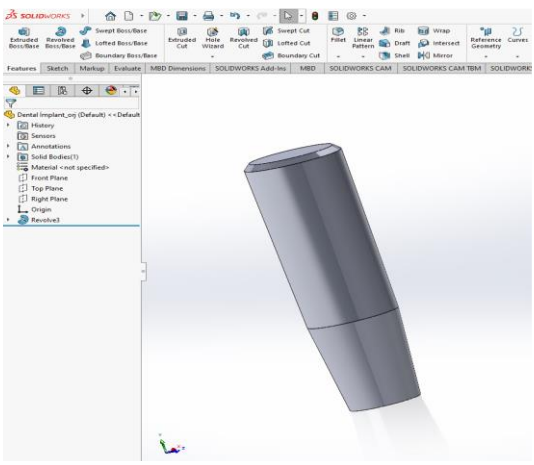
Figure 1-3. Step of transforming the dental implant design from 2D to 3D

The previously defined centerline in the 2D sketch is selected and confirmed as the axis of revolution. Upon validation of this reference, the software automatically generates the initial 3D solid model of the dental implant, as shown in Figures 1-4(a)–(b). In order to create the thread geometry of the dental implant on the generated initial solid model, a two-level (dual-step) sketch plane is constructed, as illustrated in Figures 1-5(a) and 5(b).

For defining this sketch plane, the Reference Geometry and Plane options are selected under the Features tab of the CAD software. To ensure proper geometric alignment and to accommodate manufacturing tolerances in the screw-thread region, an additional reference plane (Plane 2) is created together with Plane 1.

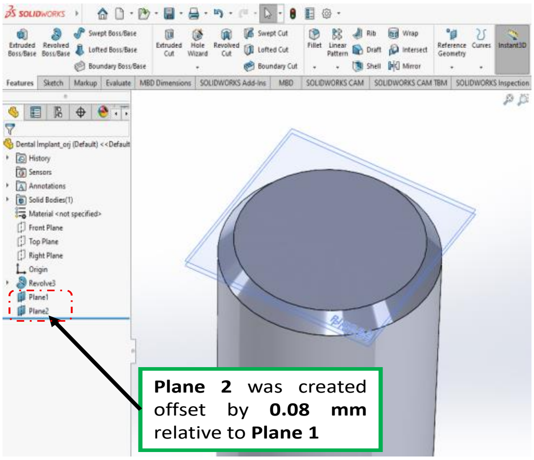


a)

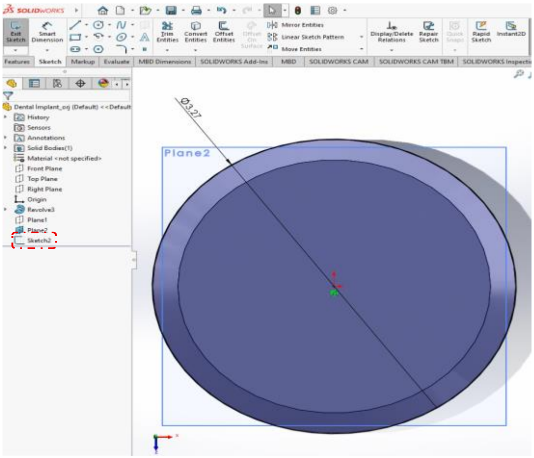


b)

Figure 1-4. Initial stage of the 3D solid modeling process of the dental implant: (a) Revolve operation from 2D sketch to 3D geometry (b) Generation of the preliminary solid model



a)



b)

Figure 1-5. Detailed sketching at the upper neck region of the dental implant:
(a) Construction of dual reference planes
(b) Generation of the circular profile for the thread initiation

On Plane 2, a new sketch window is created to define the starting profile for the helical path that the screw threads will follow.

In this sketch, a circle is drawn, representing the diameter of the dental implant. The circle is constructed based on a reference radius of R1.635 mm.

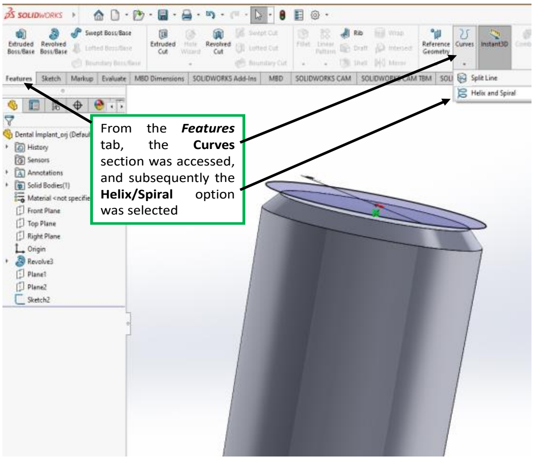


Figure 1-6. Generation of the helix at the upper neck region of the dental implant

The next step involves generating the helical curve, which defines the trajectory along which the screw threads will be formed. For this purpose, while the most recently drawn sketch is selected, the Curves section is accessed from the Features tab, and then the Helix or Spiral command is used to create the helical structure, as illustrated in Figure 1-6. At this stage, in order to define certain parameters and input values, a dialog window appears on the left panel of the CAD software interface during the Helix/Spiral command operation. In this window, the first definitions to be made are the height and angle values. Subsequently, within the parameters section, the variable pitch option is selected. Attention must be paid to the three-stage input of height values and the corresponding variations in diameter. The changes in these input parameters are illustrated in Figure 1-7. In the next step, to construct the screw threads, a plane perpendicular to the starting point of the helical spiral is defined (Figure 1-8). On this newly created plane, a sketch file is opened, and the thread profile geometry is drawn, as shown in Figure 1-9.

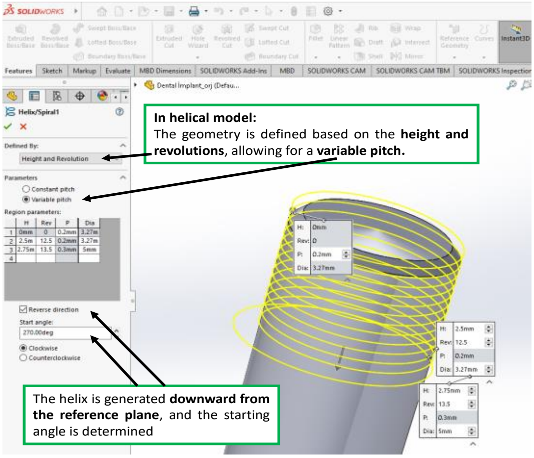


Figure 1-7. Design details of the helix at the upper neck region of the dental implant

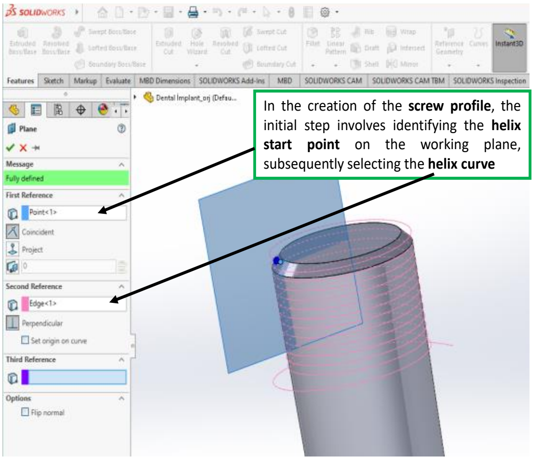


Figure 1-8. Creation of the thread profile plane

In the profile design process, a polygon is selected from the geometric shapes available in the sketch window, and in the dialog box that appears, the number of sides is set to three. The goal is to obtain a triangular profile with an inscribed circle. The key step in the profile design is aligning the geometric center of the drawn profile with the helical spiral curve by defining a pierce point, as shown in Figure 1-10.

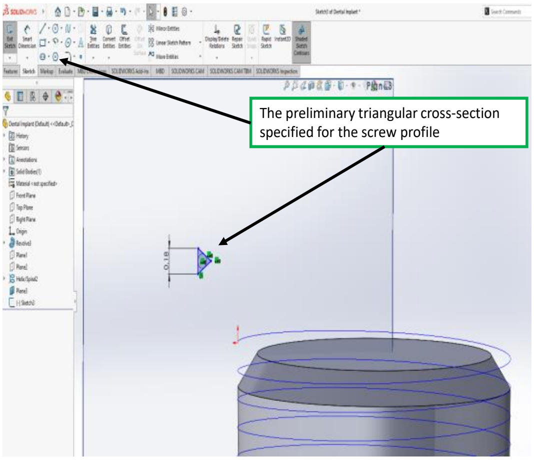


Figure 1-9. Sketching of the preliminary thread profile

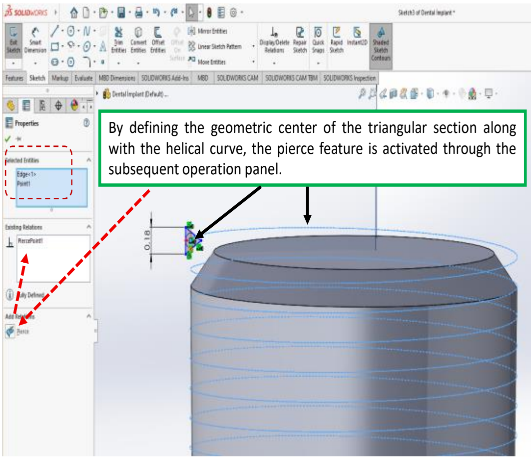


Figure 1-10. Alignment of the screw thread profile with the helical path design

Next, to achieve the proper thread form, a fillet radius of 0.03 mm is applied to the sharp inner corner of the triangle. From the triangle's central axis, a rectangular profile (height: 0.16 mm) is drawn, using the midpoint of the base as a reference. This additional geometry is then merged with the triangular profile to ensure geometric continuity, as illustrated in Figure 1-11.

Unnecessary segments of the thread profile are trimmed, and fillet operations are applied to smooth the remaining sharp edges, as shown in Figure 1-12. As a result of these steps, the two-dimensional thread profile design for the upper neck region of the dental implant is completed.

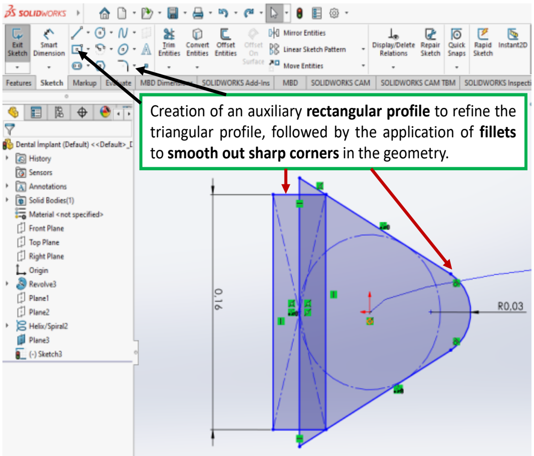


Figure 1-11. Development of supplementary geometrical features in the thread profile detailing process

After completing the two-dimensional design of the thread profile, the next step is to create the thread geometry on the surface of the solid model that represents the body of the dental implant. In this process, the previously designed thread profile is swept along the helical path and simultaneously cuts through the solid model, thereby forming the screw threads on its surface. To perform this operation, the Swept Cut command located under the Features tab is selected to initiate the process, as illustrated in Figure 1-13. When the Swept Cut command is selected to generate the thread profile on the dental implant, a dialog window appears. In this window, the required selections include the sketch profile and the path to be followed. The two-dimensional thread profile previously designed for the dental implant is selected as the cutting profile, while the helical spiral is defined as the path along which the sweep will occur.

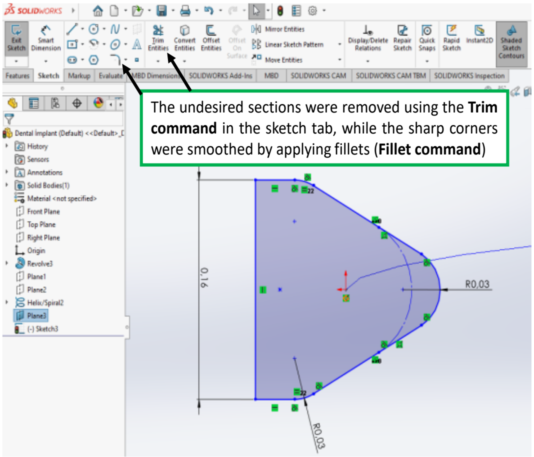


Figure 1-12. Completed 2D thread profile of the upper neck region of the dental implant

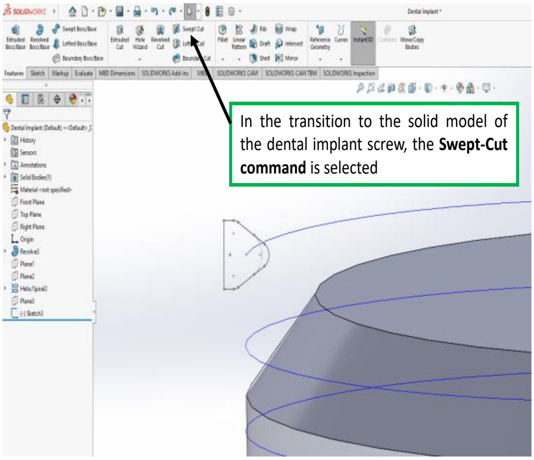


Figure 1-13. Generation of the solid thread geometry at the neck region of the dental implant

Once these parameters are confirmed, the cutting operation is executed, as shown in Figure 1-14. The final appearance of the threaded region after the cutting process is presented in Figure 1-15.

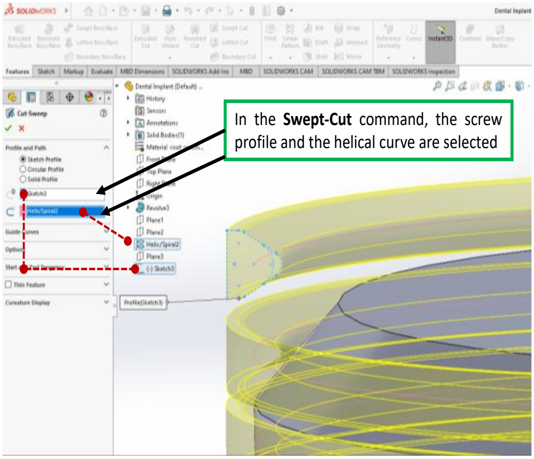


Figure 1-14. Implementation of the Swept Cut operation for generating the thread profile on the dental implant neck

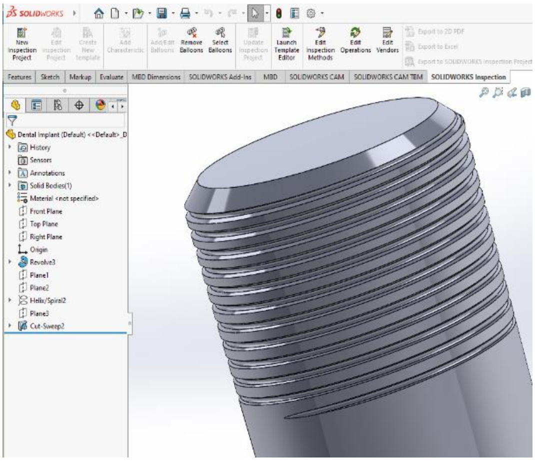


Figure 1-15. Completed thread profile geometry at the neck section of the dental implant

In the next stage of the dental implant design, an additional and unique structural feature was planned below the neck region to enhance the osseointegration effect. For this purpose, plane 4 is created so that it is positioned toward the outermost edge of the dental implant.

To construct plane 4, the reference geometry option is accessed from the Features tab, and the plane command is selected. The Front Plane is defined as the reference, and since the radius of the dental implant is 1.635 mm, this value is entered as the offset distance to generate Plane 4, as shown in Figure 1-16. Subsequently, a two-dimensional design of the new surface feature intended to improve the osseointegration performance is created, as illustrated in Figure 1-17. In the following step, the solid model of this 2D design is generated using the Cut Extrude command. During this process, appropriate fillet radii are applied, and a slight inward taper is introduced to the geometry to achieve a smoother transition and a more realistic bone–implant interface, as depicted in Figure 1-18.

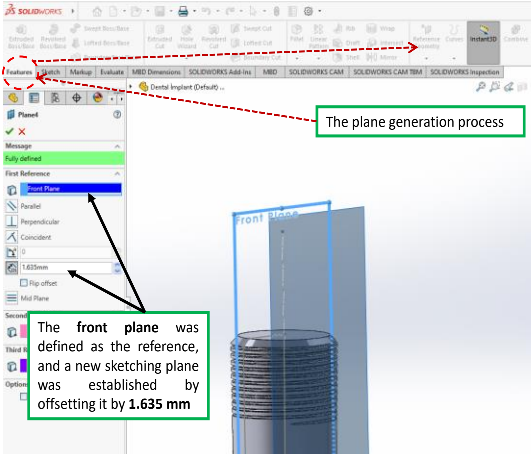


Figure 1-16. Establishment of a new reference plane below the neck region of the dental implant

In the solid model shown in Figure 1-18, an inward cut of 0.15 mm was applied with a taper angle of 27 degrees. The Cut Extrude command was used to create this solid model beneath the thread section of the dental implant, aiming to improve the osseointegration capability of the neck region.

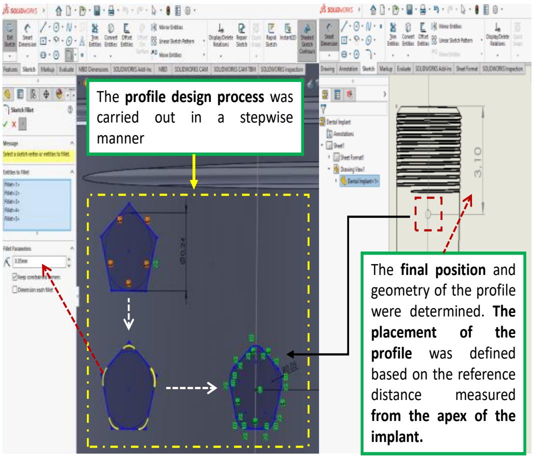


Figure 1-17. Process of creating the two-dimensional design on the lateral surface of the dental implant

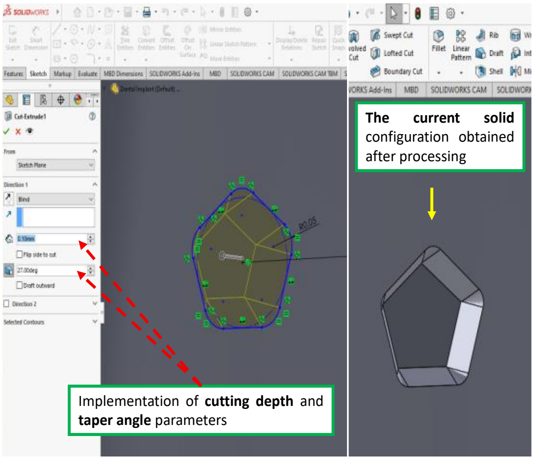


Figure 1-18. Pentagonal solid model created on the lateral surface of the dental implant

To smooth the sharp edges of the generated solid model, the Fillet command in the Features tab was applied, as illustrated in Figures 1-19 and 1-20. A fillet radius of 0.03 mm was applied in the models illustrated in Figures 1-19 and 1-20.

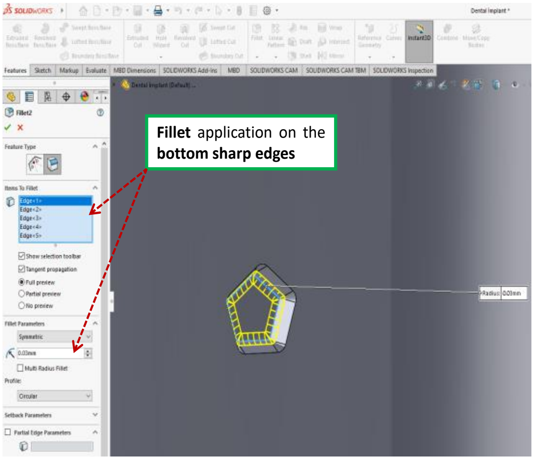


Figure 1-19. Fillet application to smooth the lower edges of the pentagonal solid model

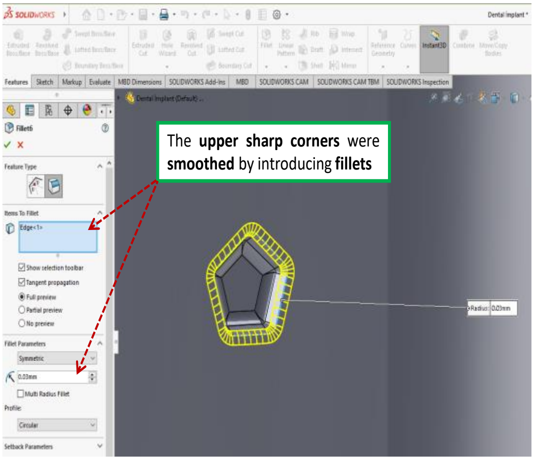


Figure 1-20. Fillet application to smooth the upper edges of the pentagonal solid model

Through these adjustments, the modeling process for this stage was completed, as presented in Figure 1-21. To enhance the osseointegration effect of the dental implant, the pentagonal form created on its lateral surface was replicated multiple times around the cylindrical implant body by forming an arched pattern along the side surface.

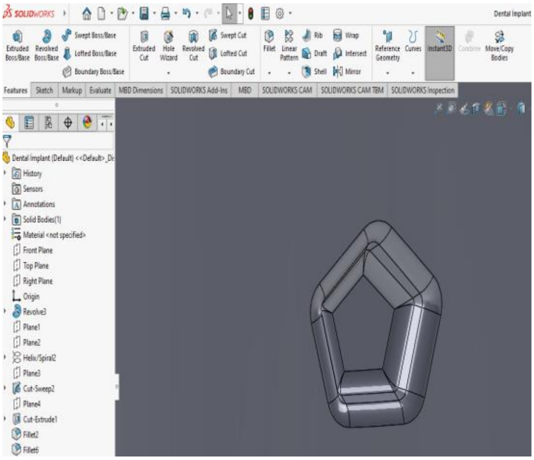


Figure 1-21. Completed pentagonal solid model following the fillet operations

For this purpose, the Circular Pattern feature located in the Pattern section of the Features tab was utilized, as illustrated in Figure 1-22.

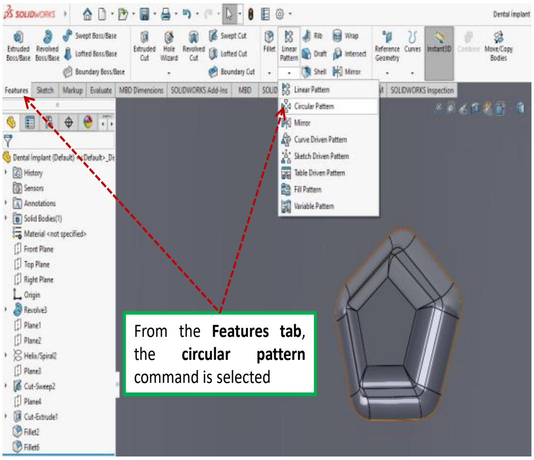


Figure 1-22. Selection of the circular pattern for replicating the pentagonal solid model

When the Circular Pattern dialog window is opened, several input fields appear that allow defining the direction, angle, and number of instances for the pattern. To specify the direction, the circular edge located on the upper surface of the dental implant is selected.

The angle is then set to 45°, and since the pattern is distributed around a circular form, the number of instances is defined as eight. These parameter settings are illustrated in

Figure 1-23. The final configuration of the arched patterned structure resulting from the circular duplication process is shown in Figure 1-24.

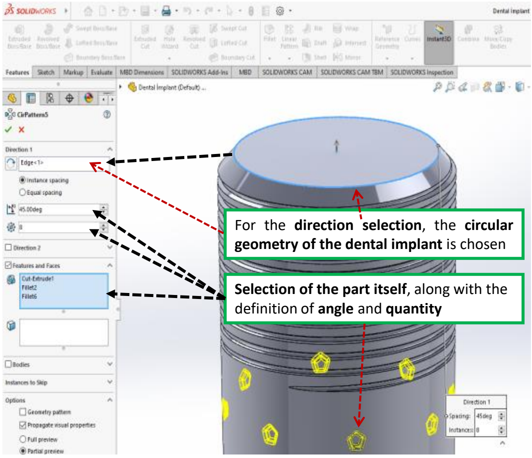


Figure 1-23. Application steps of the circular pattern process for replicating the pentagonal solid model

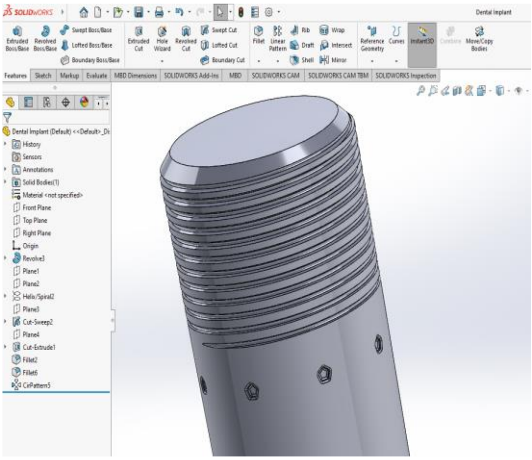


Figure 1-24. View of the pentagonal solid model after the circular pattern operation

The next stage involves generating the threaded solid model for the lower and middle body regions of the dental implant. This procedure follows the same step-by-step approach previously applied in the modeling of the upper neck section. As the first step, a reference plane is defined at the bottom portion of the implant (Figure 1-25).

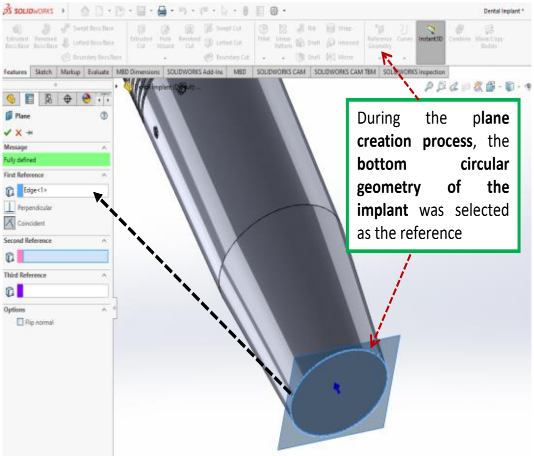


Figure 1-25. Definition of a reference plane on the bottom surface of the dental implant

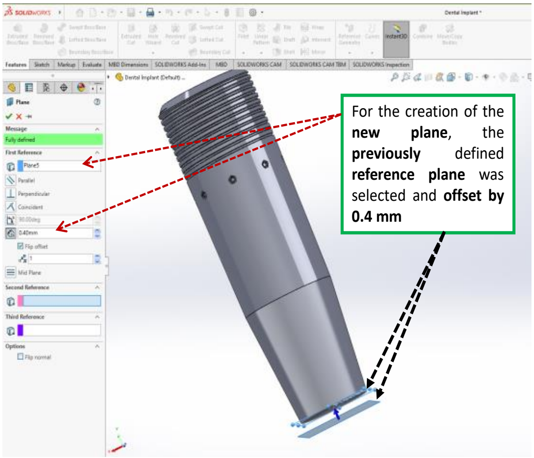


Figure 1-26. Definition of an additional reference plane on the bottom surface of the dental implant

To improve the thread emergence and formation, an additional plane (plane 5) is created by offsetting 0.4 mm outward from the bottom surface, as shown in Figure 1-26. A new sketch window is then opened on this plane, where a circular geometry is drawn (Figure 1-27).

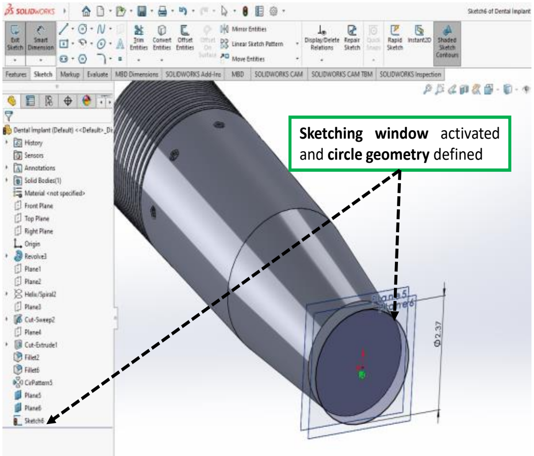


Figure 1-27. Drawing of the 2D circular geometry on the bottom surface of the dental implant

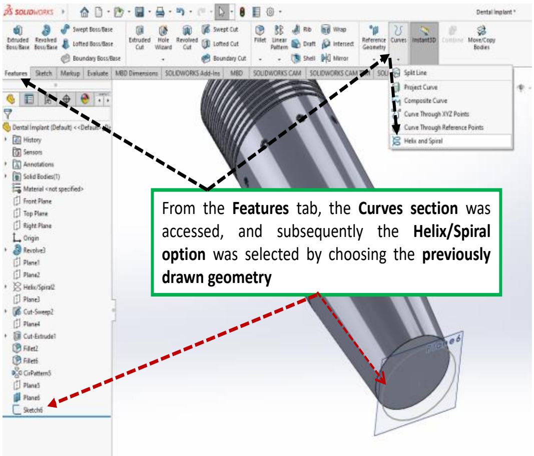


Figure 1-28. Creation of a helix from the new geometry on the bottom surface of the dental implant

The circle corresponds to the same diameter as the implant's bottom base surface. Using this circular sketch, a helical spiral is generated (Figure 1-28). In the helix/spiral parameters, both height and pitch values are defined, with the pitch (advance per turn) set to 0.4 mm. After adjusting these parameters and applying additional refinements, the helical spiral is successfully created, as illustrated in Figure 1-29.

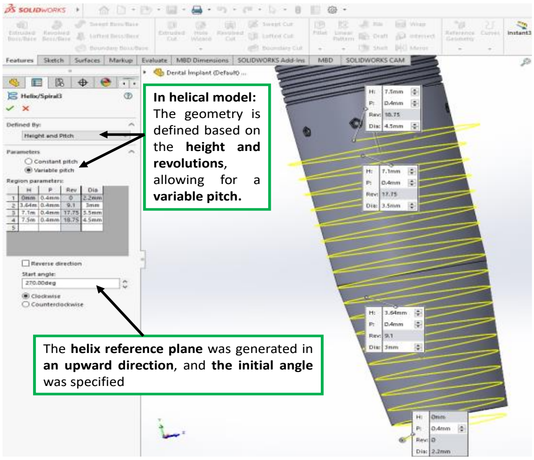


Figure 1-29. Detailed view of the helix generation from the newly defined geometry at the base of the dental implant

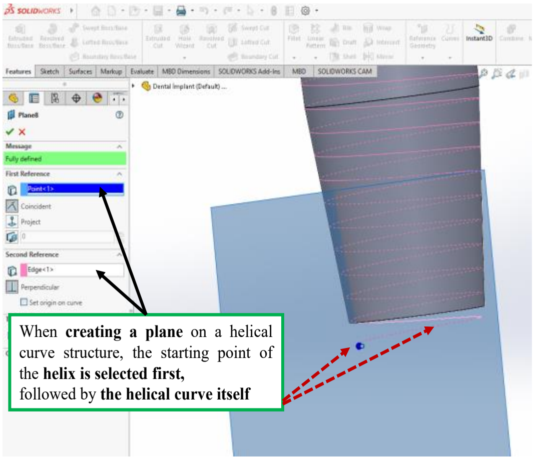


Figure 1-30. Creation of a plane perpendicular to the helical curve

In the next step, a plane perpendicular to the helical curve is created to design the thread profile corresponding to the generated helical path. For this purpose, the starting point of the helix and the helical curve itself are selected to define the orientation of the new plane (Figure 1-30). A sketch window is then opened on this plane, and an equilateral triangle is drawn by selecting the polygon tool for the thread profile (Figure 1-31).

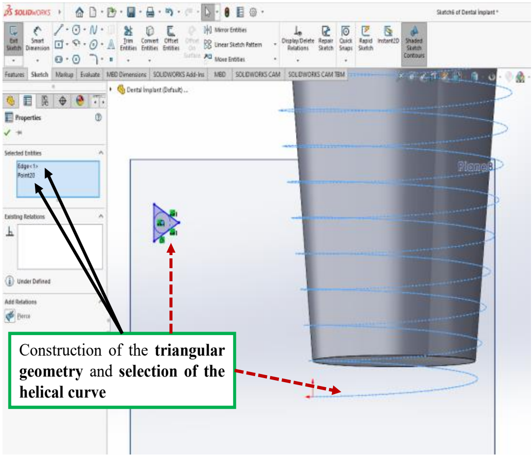


Figure 1-31. Alignment of the triangular profile with the helical path

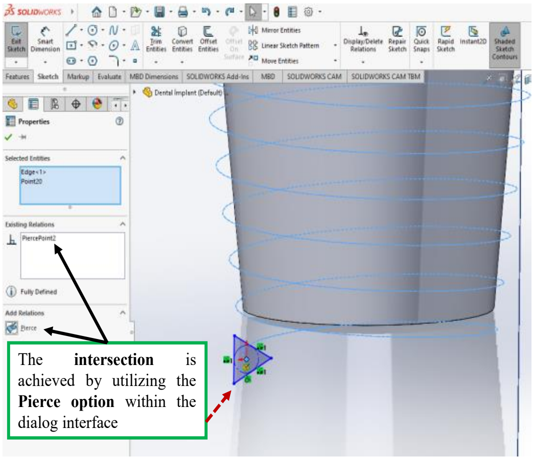


Figure 1-32. Coincidence of the triangular geometry and the helical path

Next, the geometric center of the triangle and the helical curve are selected, and in the dialog window, the "Pierce" constraint is applied to align the two entities (Figures 1-31 and 1-32). In this way, the geometric center of the triangular profile is coincident with the starting point of the helical curve (Figure 1-32). The triangular geometry selected for the thread profile represents the initial step of the thread design. The detailed configuration of the thread profile design is shown in Figure 1-33.

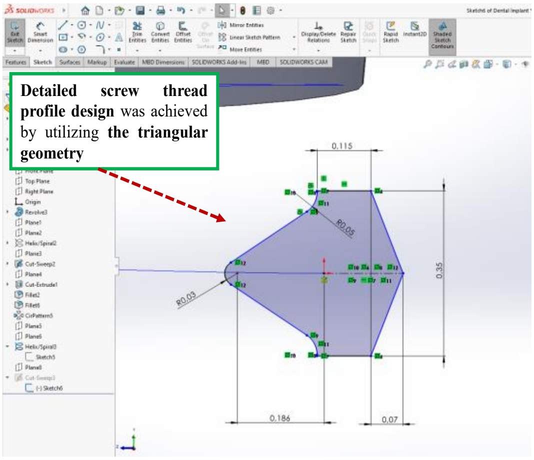


Figure 1-33. Detailed design development from the triangular geometry to the screw thread profile

With the completion of the detailed design that defines the final form of the thread profile, the next step involves generating the solid model of the thread on the dental implant using the Swept Cut command, as shown in Figure 1-34. All modeling steps for the dental implant have been successfully completed at this stage. Following this, the upper section of the dental implant where the abutment screw will be mounted can be further modified or detailed depending on the requirements of the alternative design project. The final form of the dental implant model, based on the current design, is presented in Figure 1-35.

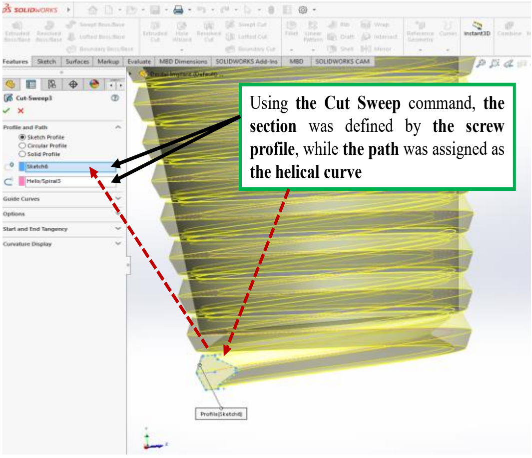


Figure 1-34. Generation of the 3D solid model based on the 2D screw thread profile design

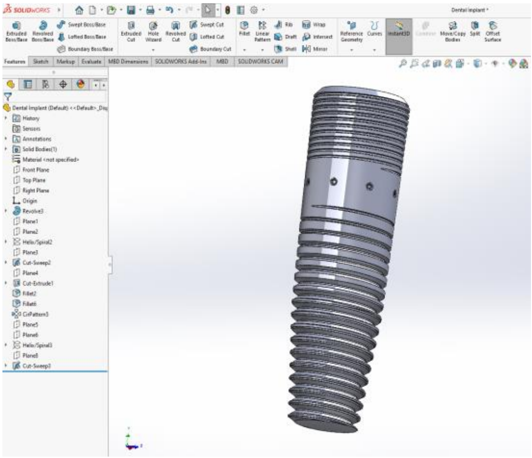


Figure 1-35. Completed solid model of the dental implant design

Figure 1-35 presents the final 3D solid model of the dental implant after the completion of the full design process.

2. *DETAILED DEVELOPMENT OF THE INTRAVASCULAR STENT AND ITS DESIGN MODEL*

Intravascular stents, which are frequently preferred in the treatment of vascular diseases, have established themselves as an effective and critical therapeutic option in the field of cardiovascular surgery [24]. Cardiovascular stents, which are frequently used in the treatment of vascular diseases, have become an effective and critical therapeutic method in cardiovascular surgery. Specifically, heart diseases, which account for approximately 600,000 deaths annually, represent one of the leading causes of mortality. At the core of these diseases lies atherosclerosis, a prevalent cardiovascular condition characterized by the accumulation of cells, lipids, connective tissue, calcium, and other substances within the inner layer of the arterial wall. This accumulation forms a fatty deposit known as an atheroma, which can lead to arterial stiffening, wall rupture or erosion, and ultimately a reduction or complete blockage of blood flow [25]. The development of this condition may begin in early life and continue into adulthood. Among the treatment methods, angioplasty is widely used, and this technique is based on inserting a balloon-tipped catheter to dilate the narrowed vessel segment, especially in wide-necked complex aneurysms [26]. Although this procedure helps expand the vascular lumen and restore blood flow, restenosis the re-narrowing of the vessel may occur within days or weeks following angioplasty. Due to the high recurrence rate of such complications, research has been directed toward developing permanent solutions. Consequently, stents expandable metallic mesh structures designed to remain permanently within the vessel have been introduced and widely adopted [25]. The primary purpose of using stents is to provide mechanical support and maintain vessel patency, serving as an essential interventional approach to vascular occlusions [24]. In the early stages of stent development, the materials used were primarily 316L low-carbon medical-grade stainless steel, resulting in uncoated metallic stents. In recent years, with the advancement of technological and material-science innovations, new generations of stent designs have emerged, including shape-memory intelligent stents, drug-eluting stents, and biodegradable stents. The materials utilized in these next-generation stents include polymeric materials, nitinol (NiTi shape-memory alloy), titanium-based alloys, and magnesium-based materials. Although stents are manufactured in various designs and material compositions, their primary objective remains the same: to provide an effective solution for vascular occlusions and to restore normal blood flow. Therefore, the biomechanical behavior of stents within the human body continues to be an active and critical area of research in current scientific studies [27, 28]. As with other biomechanical implants and prostheses used within the human body, stents are also evaluated prior to manufacturing by performing finite element stress analyses under conditions that simulate biomechanical environments. For this purpose, the development of accurate solid model designs of stents represents a critical initial step. In the present study, we will examine in detail how a stent design is developed, providing a comprehensive explanation of each stage of the modeling process.

2.1. Intravascular Stent Design Model

For the design of the planned stent, a part file is first created in the solid modeling program. At the beginning of the study, this part file is named according to the planned project in this case, it will be called **'Vascular Stent'.** To begin the design process, the Front Plane is selected from the Feature Tree section located on the left side of the modeling interface, and a new sketch window is opened on this plane. These steps are illustrated in Figure 2-1. Next, in the opened sketch window, the initial two-dimensional (2D) outline of the planned vascular stent model is drawn. The subsequent step is to convert the defined 2D rectangular sketch into a solid model. To accomplish this, while the sketch is selected, the 'Extruded Boss/Base' command is accessed from the Features tab (Figure 2-2). At this stage, the program prompts for the extrusion distance, which is set to 6 mm, as shown in Figure 2-3. Following the commencement of the preliminary solid modeling of the stent, a sketch plane is created on the wide surface of the generated solid (380 mm × 520 mm).

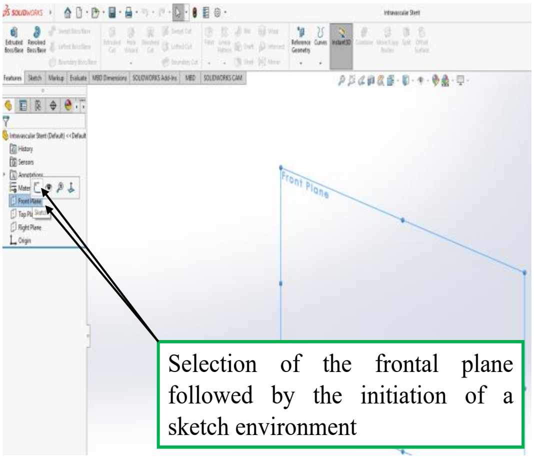


Figure 2-1. Selection of the front plane and creation of the drawing window

On this sketch plane, an original design forming the lattice structure which plays a crucial role in the stent configuration is drawn (Figure 2-4).

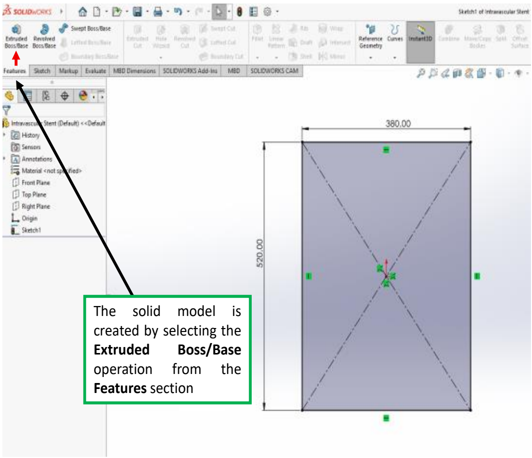


Figure 2-2. Beginning of the 2D design of the stent model and transition to the 3D model

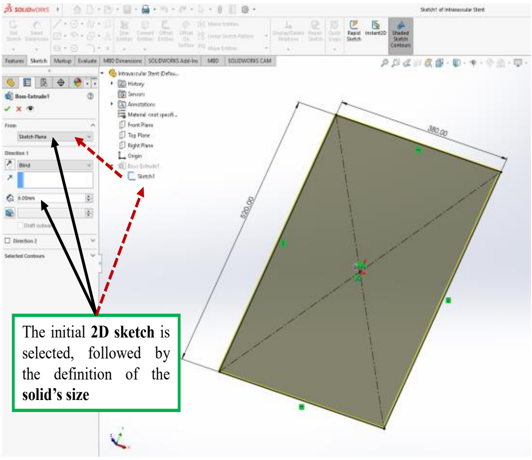


Figure 2-3. Beginning of the 3D solid modeling of the stent model

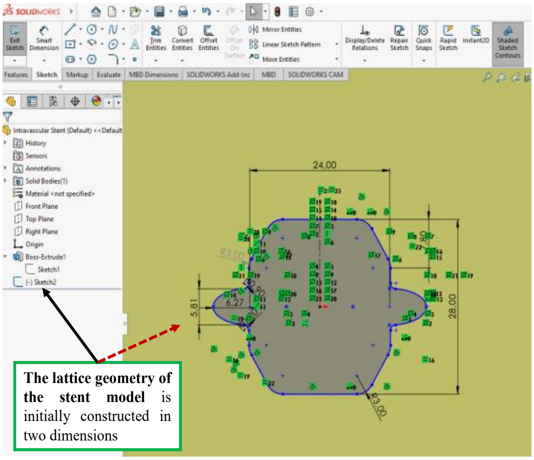


Figure 2-4. Design and dimensioning of the 2D lattice geometry of the stent model

Subsequently, the ***cut-extrude*** command is used to convert the generated lattice structure into a solid model (Figure 2-5). After creating the solid model of the lattice geometry of the stent, the geometry is duplicated over the main solid surface to form the complete stent model.

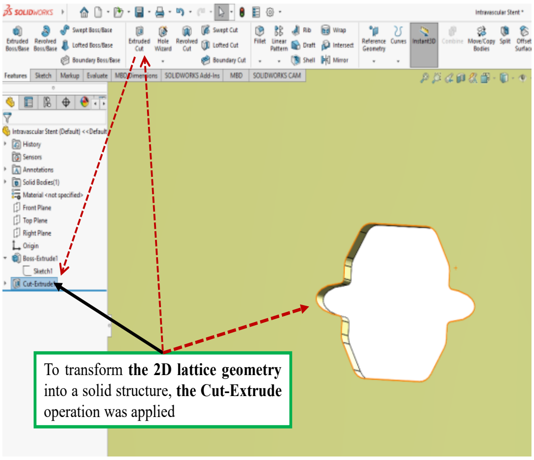


Figure 2-5. 3D lattice solid model of the stent

For this purpose, the linear pattern feature under the features tab is accessed, and then the **fill pattern** option is selected to make the necessary adjustments for filling the entire surface (Figure 2-6).

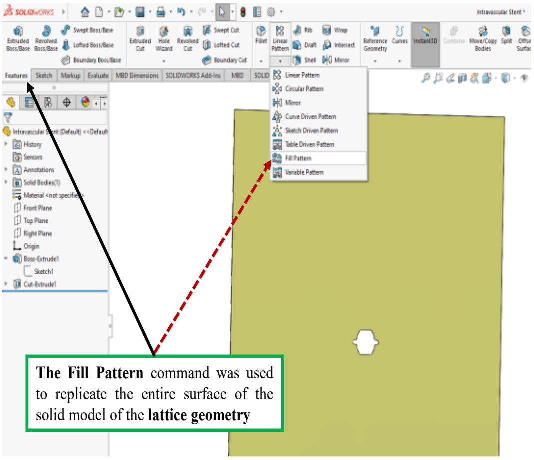


Figure 2-6. Duplication of the 3D lattice solid model of the stent

After selecting the *fill pattern* command, the general main surface was defined as the filling reference. Subsequently, the solid model of the lattice geometry to be replicated on this surface was selected and applied.

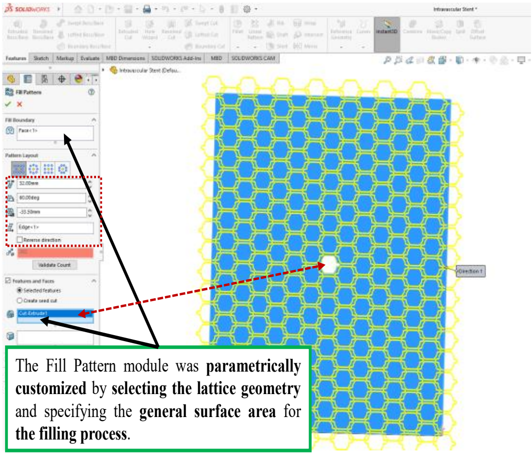


Figure 2-7. Application of the Fill Pattern on the 3D lattice solid model of the stent

During this process, the distance between two adjacent lattice geometries was set to 32 mm, and the orientation angle of the lattice on the filling surface was adjusted to 60 degrees (Figure 2-7). Following the use of the *fill pattern* module, the final form of the resulting solid model is presented in Figure 2-8.

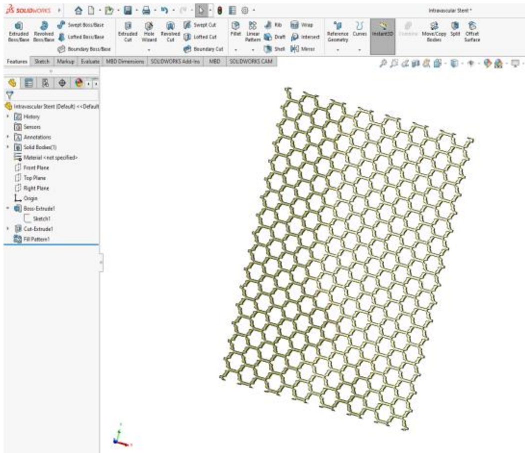


Figure 2-8. View of the 3D lattice solid model of the stent applied over the entire Surface

The next step in the solid modeling process of the stent is to transform the solid model containing the lattice geometry into a cylindrical form suitable for intravascular application. For this purpose, the *Insert*, *Features*, and *Flex* commands are sequentially selected (Figure 2-9).

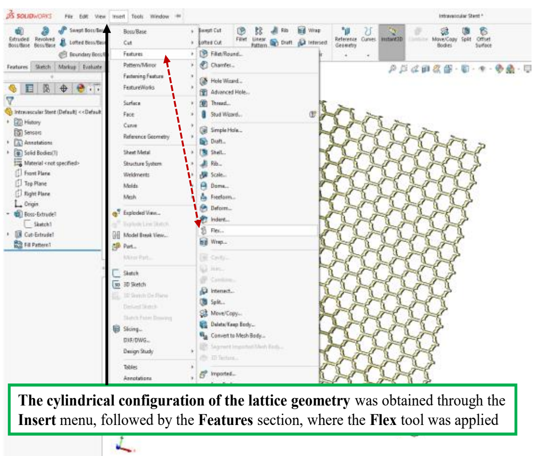


Figure 2-9. Conversion stage of the lattice geometry into a cylindrical configuration

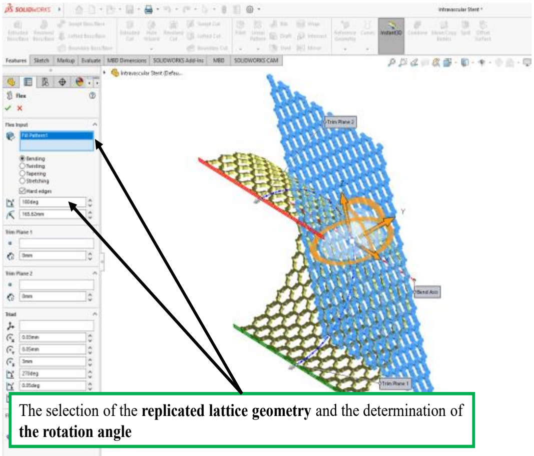


Figure 2-10. Bending of the lattice geometry into a cylindrical configuration

After activating the *Flex* command, the duplicated lattice solid model is selected, and a bending rotation is applied with an initial angle of 180° (Figure 2-10). Subsequently, the *Mirror* command is utilized, where both the mirror plane and the part to be mirrored are defined, resulting in the final solid model of the intravascular stent (Figures 2-11 and 2-12).

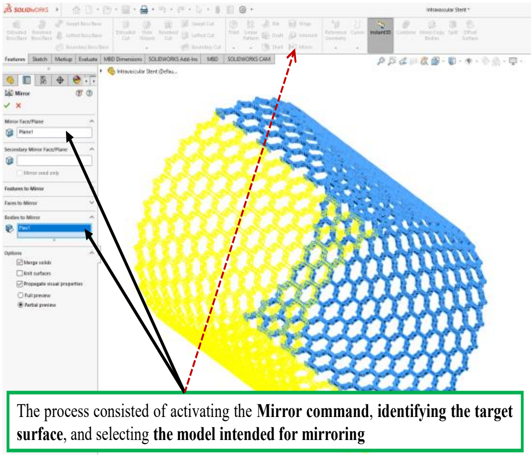


Figure 2-11. Conversion of the semi cylindrical lattice solid model into a full cylindrical configuration using the mirror command

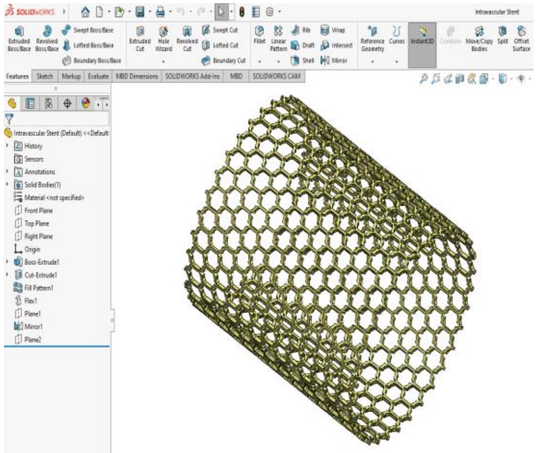


Figure 2-12. Complete representation of the intravascular stent model

3. *DETAILING OF THE PLATE IMPLANT AND DESIGN MODEL USED IN FEMORAL FRACTURES*

Fractures of the femoral shaft represent one of the most common fracture types observed in orthopedic practice [29]. Notably, approximately 30% of periprosthetic femoral fractures develop at the distal tip of the femoral stem, highlighting this region as a biomechanically critical zone. These fractures are categorized as Vancouver type B1, in which the femoral implant component retains its stability [30]. In such types of femoral shaft fractures, modern locking lateral plate osteosynthesis on the unilateral surface is commonly employed as the preferred fixation method. [30, 31]. In fact, recent studies have indicated that treatments employing single-sided plate implant applications are often insufficient; therefore, dual plating or bilateral plate osteosynthesis approaches have been recommended for distal femoral fractures [31, 32]. Accordingly, biomechanical evaluations have emphasized that double plate osteosynthesis provides superior performance compared to single lateral plating in terms of axial and torsional stiffness, fracture displacement, stress distribution, and failure resistance under loading conditions [31, 33, 34, 35]. In the biomechanical evaluation of plate implant applications used for femoral fractures, researchers have particularly employed the finite element method to investigate the effects of lateral plate implantation and dual plate implantation techniques [36, 37]. For this purpose, in studies presenting biomechanical evaluations of femoral fractures through finite element analysis, the solid model design of patient-specific plate implants constitutes the primary determinant of the study. Similarly, in the the current study, as a representative example, the detailed design of the plate implant model was developed and described based on an assumed curvature of the femur bone without using data from any specific individual.

3.1. Design and Modeling of the Plate Implant

For the planned plate implant design, a new part file is first created in the solid modeling software. At the beginning of the process, the part file is named according to the intended project; in this study, it is referred to as Plate Implant. To begin the design within the part file, the front plane is selected from the feature tree located on the left side of the modeling interface, and a sketch window is created. These steps are detailed in Figure 3-1. Subsequently, in the opened sketch window, a two-dimensional sketch defining the cross-section of the planned plate implant model is drawn (Figure 3-2). In the next stage, to design the path defining the distance that the plate implant will cover along the femur, the right plane is selected and a new sketch window is opened. In this window, the path that the plate implant will follow over the femoral surface is drawn (Figure 3-3).

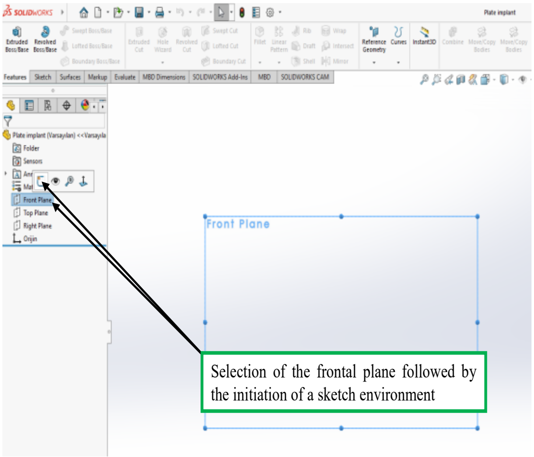


Figure 3-1. Selection of the front plane and creation of the drawing window

In the following step, for the plate implant design with the defined cross-sectional profile and path, the solid sweep command is selected from the features tab.

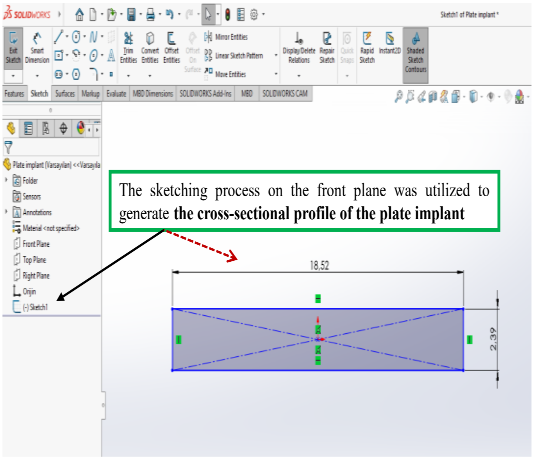


Figure 3-2. Cross-sectional 2D geometry prepared for the plate implant modeling process

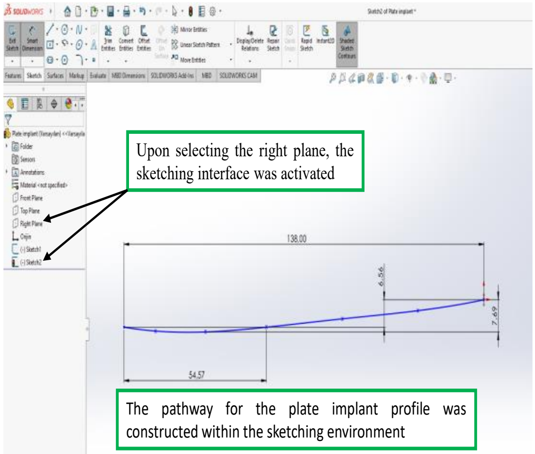


Figure 3-3. Sketch created on the right plane to define the path of the plate implant over the femur surface

Within the opened sweep window, both the cross-section geometry and the path geometry of the plate implant are individually selected, and the solid sweep operation is executed (Figure 3-4). After applying the solid sweep operation, the resulting appearance of the plate implant model is as shown in Figure 3-5.

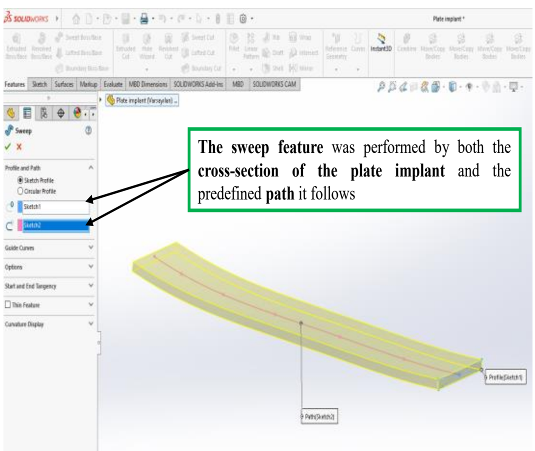


Figure 3-4. Application of the solid sweep command utilizing the cross-section and path sketches of the plate implant

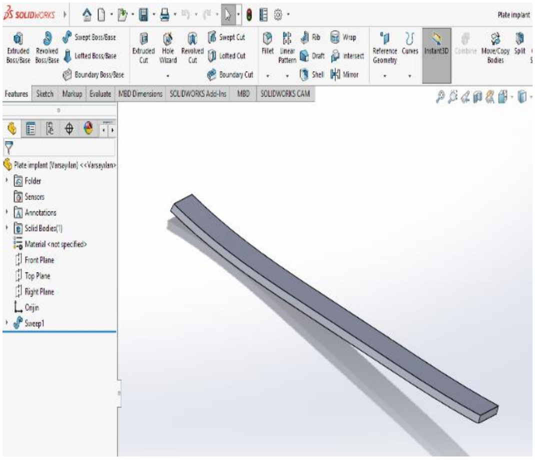


Figure 3-5. Completed solid model of the plate implant created by the solid sweep command

In the next step, to create detailed design features (such as screw holes) on the initial solid model of the plate implant, a new reference plane is generated from the features tab under reference geometry, positioned on the long side surface of the model (Figure 3-6).

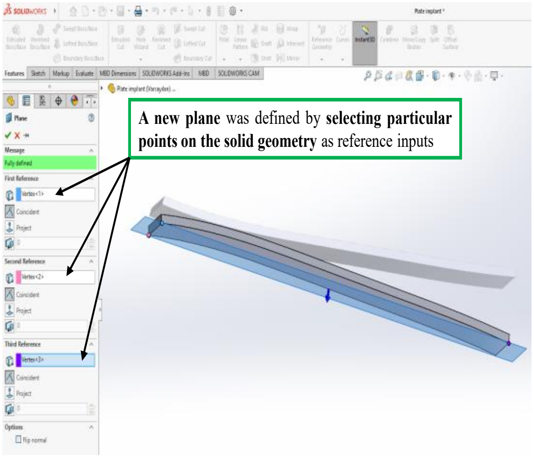


Figure 3-6. Creation of a new reference plane on the long side surface of the plate implant for detailed feature modeling

In Figure 3-7, a sketch window was opened on the newly created plane to continue the planned design of the plate implant. Two-dimensional drawings were made within this sketch. Symmetry was utilized during the drawing process; therefore, auxiliary construction lines were drawn. In several stages, the mirror command was also applied, and all sketch details were dimensioned accordingly.

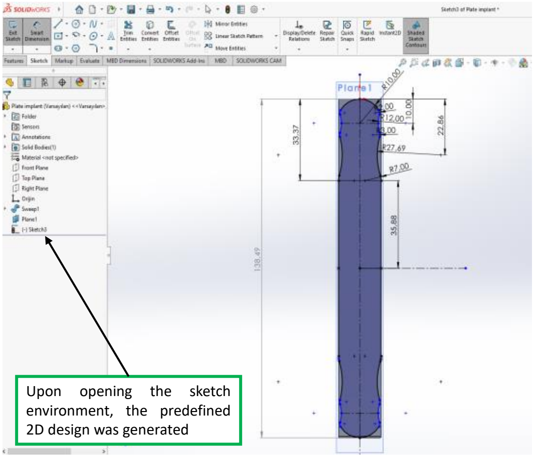


Figure 3-7. Two-dimensional detailed design of the plate implant on the newly created plane.

In Figure 3-7, the previously created 2D sketch was used to generate a solid feature by selecting the Extruded Cut command from the Features tab. For this operation, the most recently created sketch was selected as the cutting profile, and the cut direction was defined as mid-plane, extending equally in both directions. Additionally, the external region of the sketch rather than the internal one was selected for material removal during the cut operation (Figure 3-8).

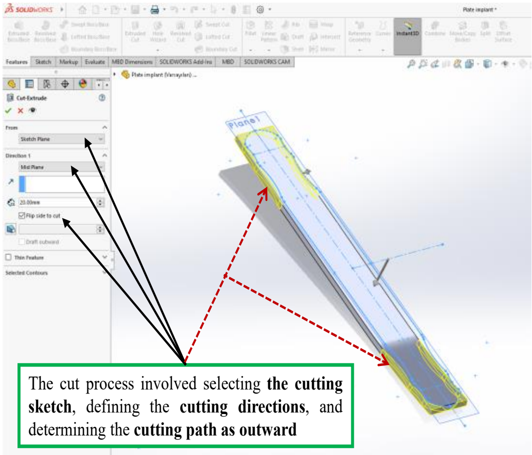


Figure 3-8. Detailed view of the cutting operation performed on the solid model of the plate implant

After the cutting operation, the next step involved opening a new sketch window on the previously generated surface to design 2D details for the creation of screw holes. These 2D sketches defined the positions and dimensions of the screw holes to be applied on the plate implant (Figure 3-9). After completing the screw hole sketches, the current sketch was selected, and the *Extruded Cut* command was applied to create the screw holes on the plate implant.

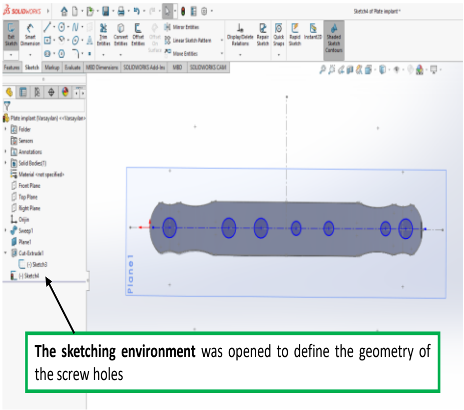


Figure 3-9. Design of screw hole positions and dimensions on the surface of the plate implant through 2D sketching.

Through this operation, the screw hole geometries were precisely formed within the solid model of the implant (Figure 3-10).

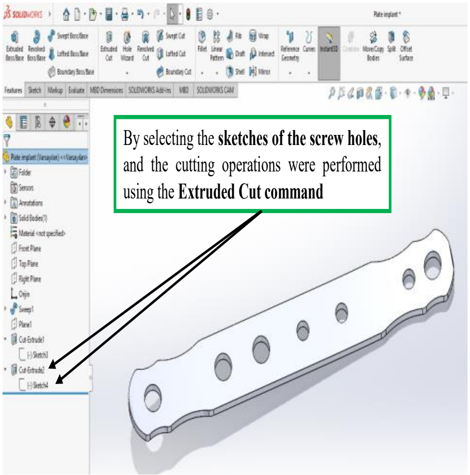


Figure 3-10. Creation of screw holes on the plate implant by cutting

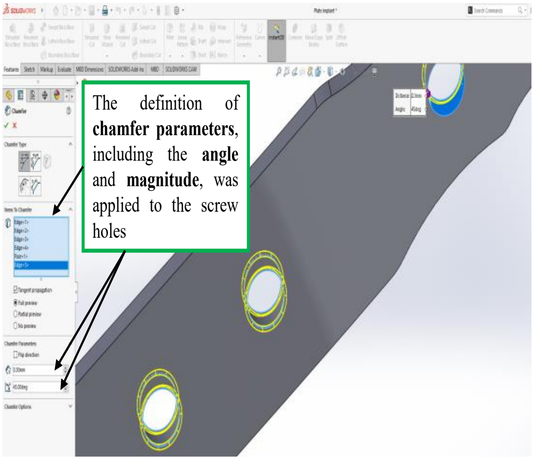


Figure 3-11. Chamfer operation for small-sized screw holes on the plate implant

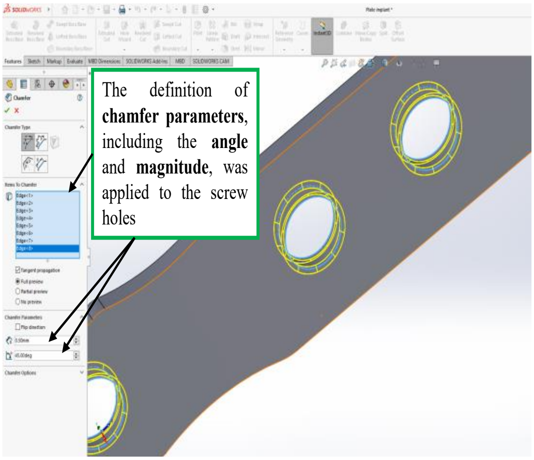


Figure 3-12. Chamfer operation for large-sized screw holes on the plate implant

After creating the screw holes on the plate implant model, chamfer surfaces were added around the screw holes to form seating areas for the screws. For this process, the Chamfer command was selected from the Features tab.

For the holes with smaller diameters, a 0.3 mm chamfer was applied, whereas a 0.5 mm chamfer was used for the larger screw holes (Figures 3-11 and 3-12). The final appearance of the plate implant after completing the design and solid modeling processes is presented in Figure 3-13.

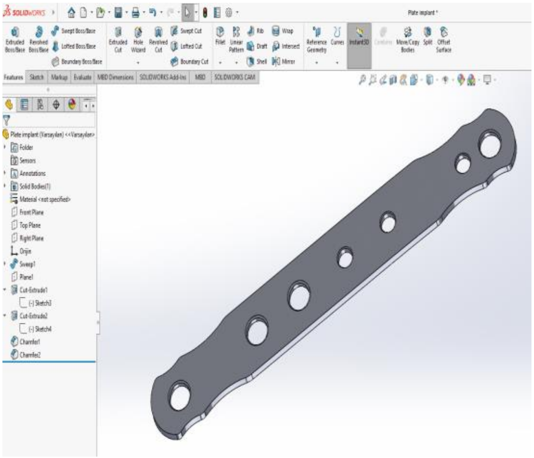


Figure 3-13. Complete view of the plate implant model

4. DETAILED DEVELOPMENT OF THE KNEE PROSTHESIS AND DESIGN MODEL

As one of the major load-bearing joints of the human body that can be rapidly exposed to sudden impacts, the knee joint possesses significant orthopedic importance owing to its extensive functional use during a lifetime and the risk of trauma-related injuries [38]. Furthermore, age-dependent joint diseases in the knee particularly osteoarthritis-induced cartilage degeneration and accompanying inflammatory processes result in restricted biomechanical mobility and have a detrimental impact on the individual's quality of life [39]. In cases of progressive joint disorders affecting the knee, total knee arthroplasty (TKA), commonly referred to as knee prosthesis implantation, is performed as the primary therapeutic intervention [40, 41]. In the application of a knee prosthesis, the axial alignment and positioning of the prosthetic components are of critical importance, as they directly influence the kinematics and biomechanical behavior of the knee joint. This precise alignment strategy plays a significant role in enhancing the longevity and functional stability of the applied prosthesis [42]. Regarding the significance of axial alignment in knee prosthesis positioning, Suh and colleagues investigated the influence of coronal alignment on joint mechanics and demonstrated that varus alignment produces greater femorotibial contact stress levels than valgus alignment [42, 43]. Koh et al. reported that different sagittal alignments of the femoral component lead to distinct changes in knee kinematics [42, 44]. Material selection for knee implant prosthesis components is determined by considering the effects of wear and the applied loads. According to the specific components of the knee prosthesis, cobalt-chromium alloy (CoCr) is used for the femoral and tibial components, polymethyl methacrylate (PMMA) for bone cement, titanium alloy (Ti6Al4V) for connecting elements, and UHMWPE, PEEK, or CFR-PEEK materials are preferred for the tibial inserts [45]. Recent studies have indicated that the use of titanium components with a porous structure has yielded improved outcomes.

The success of these porous components has been attributed to their ability to enhance the osseointegration effect of the manufactured implant parts, thereby increasing the strength of the bone–implant interface [46]. For this purpose, in the design of knee prosthesis components, porous regions have been created on specific areas of the surfaces that come into contact with bone in order to enhance the osseointegration effect. In these designs, innovative approaches have also been considered to generate the corresponding solid models.

4.1. Design Model of the Knee Prosthesis

The knee prosthesis model is comprised of three components: Femoral Component, Plastic Spacer, and Tibial Component. The Femoral Component interfaces with the femur, the Tibial Component with the tibia, and the Plastic Spacer is positioned between them to provide continuous contact and load distribution between the components. The design specifications and dimensions of the components were developed in detail.

The first component designed is the Femoral Component. For the planned Femoral Component design, a new part file was first created in the solid modeling software. At the beginning of the study, the part file was named according to the project, and this design model was designated as “Femoral Component.” To create the planned sketch in our part file, the Front Plane was first selected from the Feature Tree located on the left side of the solid modeling interface, and a new sketch environment was opened from this plane. These steps are detailed in Figure 4-1. Subsequently, the initial 2D sketch of the planned Femoral Component model was drawn in the opened sketch environment (Figure 4-2). In the next step, to convert the defined 2D sketch into a solid model, the sketch was selected, and the Extruded Boss/Base feature under the Features tab was applied to generate the 3D solid.

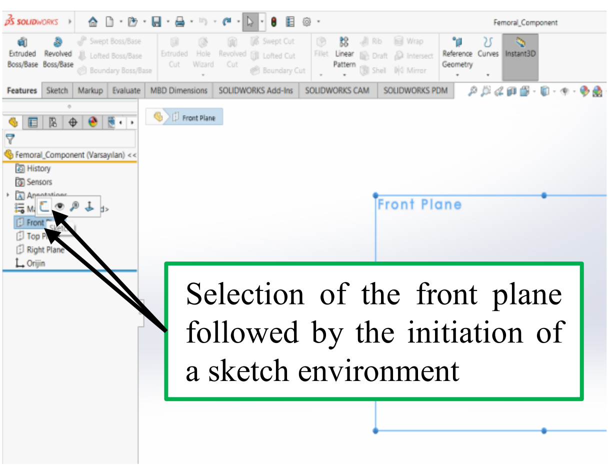


Figure 4-1. Selection of the front plane and creation of the drawing window

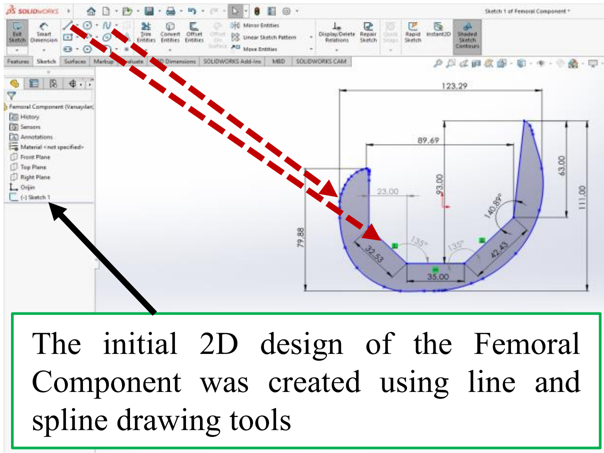


Figure 4-2. The initial stage of the 2D design of the Femoral Component model

For the current modeling, a thickness of 114 mm was applied to obtain the first solid model of the Femoral Component (Figure 4-3). In the initial stage of the solid model, certain adjustment sketches and cuts were made to enable the Femoral Component to interface with the femur. For this purpose, the Right Plane was selected, and a new sketch environment was opened on it.

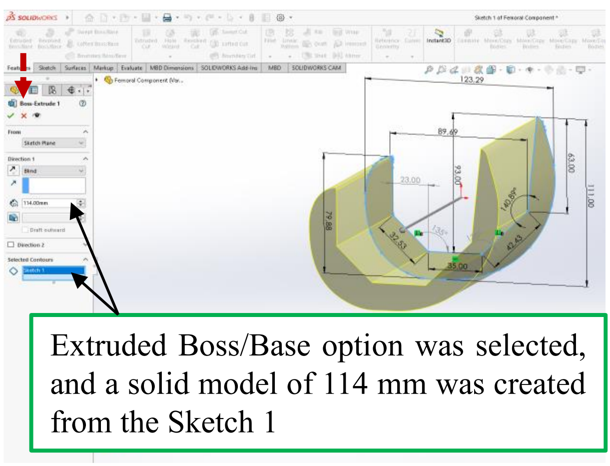


Figure 4-3. Initial development phase of the Femoral Component's 3D solid model

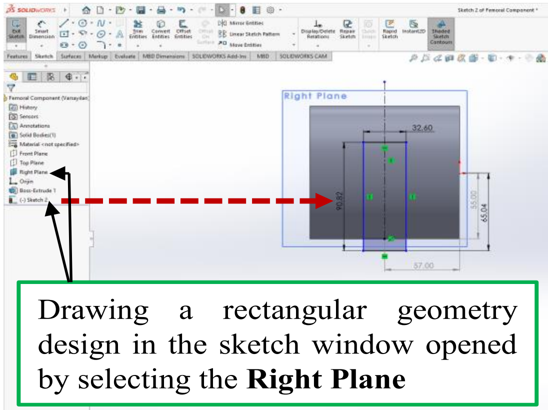


Figure 4-4. Creating a rectangular sketch on the Right Plane of the Femoral Component

The 2D sketch drawn in Figure 4-4 was used to perform a cut on the existing solid model in the next step using the Extrude Cut feature (Figure 4-5). This operation was necessary because the distal end of the Femoral Component, which interfaces with the femur, contains two protruding regions of the bone. The cut was applied in two directions as shown in Figure 4-5.

The cut was performed in two directions: a blind cut in the first direction and a cut up to the selected face in the second direction.

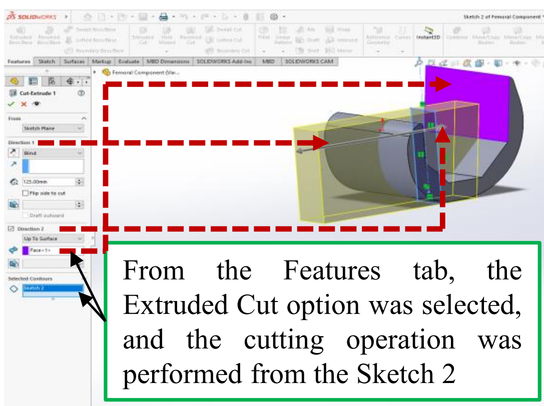


Figure 4-5. Executing a two-direction Extruded Cut on the Femoral Component based on Sketch2

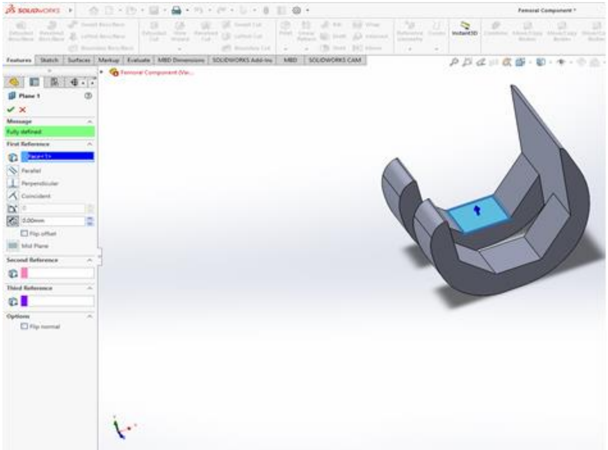


Figure 4-6. Establishment of a new plane on the chosen face of the Femoral Component

Subsequently, several additional planes were created to further detail the seating (contact) surfaces of the Femur bone on the Femoral Component. Two-dimensional sketches were created on these planes to form the corresponding bone-support regions.

For this purpose, a new plane was first created on the selected surface of the Femoral Component using the Reference Geometry option under the Features tab (Figure 4-6). Subsequently, Ø18.72 mm diameter circular geometries were sketched on two different faces covered by this plane (Figure 4-7).

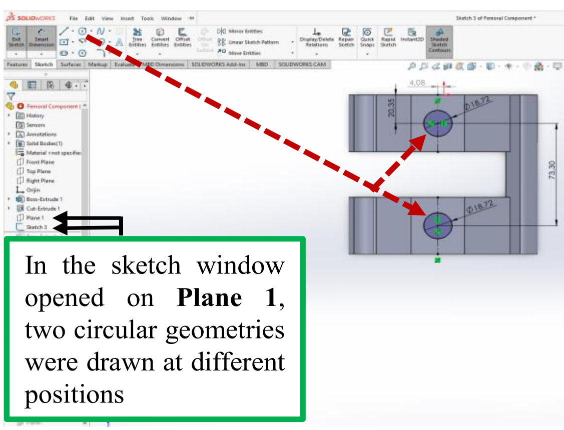


Figure 4-7. Sketching 2D circular geometries on the reference plane created on the Femoral Component

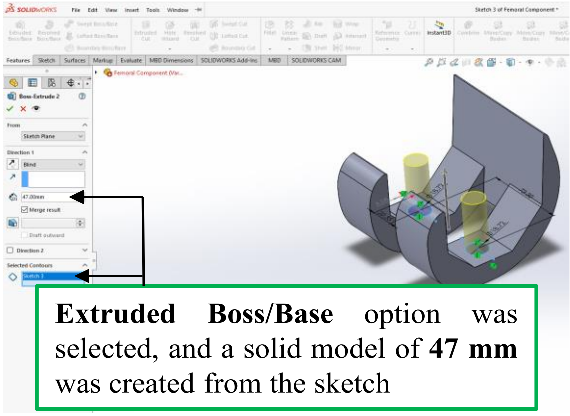


Figure 4-8. Transforming the circular sketches in Sketch 3 into a 3D solid feature

Subsequently, the sketched circular geometries were selected and converted into solid features using the Extruded Boss/Base command (Figure 4-8).

The Reference Geometry option under the Features tab was then accessed to create a new plane. By selecting the top surface of the newly formed cylindrical solid model, an additional reference plane was generated (Figure 4-9).

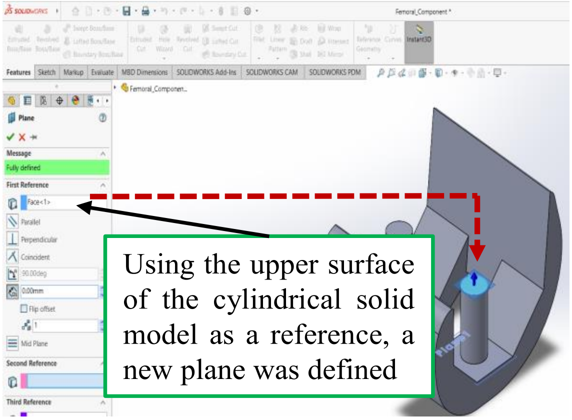


Figure 4-9. Establishment of a new plane on the top face of the cylindrical solid model on the Femoral Component

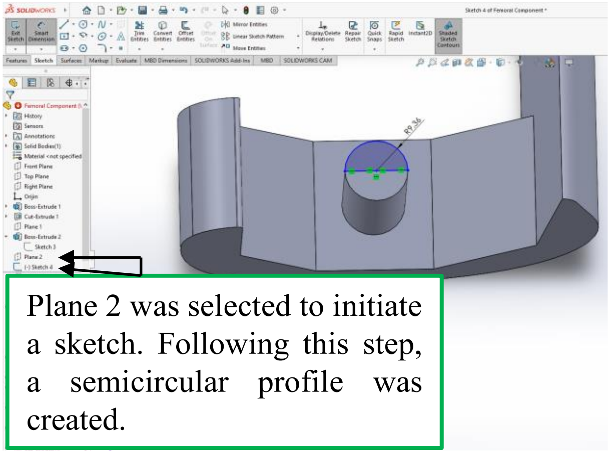


Figure 4-10. Creating a 2D semicircular sketch on the upper face of the cylindrical solid feature on the Femoral Component

Following the creation of the reference plane, a sketch containing a semicircular geometry was generated (Figure 4-10).

This sketch was revolved through 180° using the Revolved Boss/Base operation to obtain a hemispherical solid model (Figure 4-11), thereby forming the fixation feature of the Femoral Component to the femur.

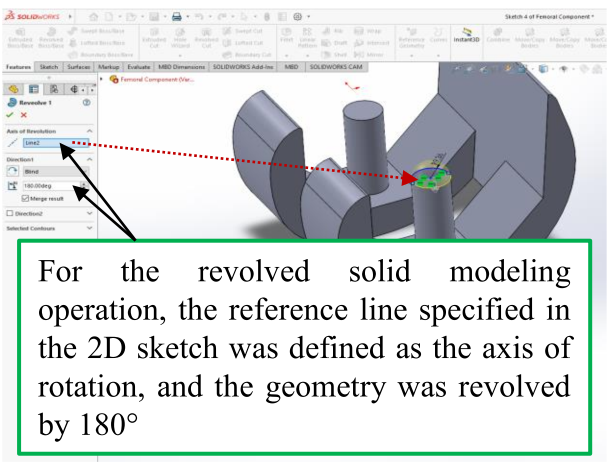


Figure 4-11. Formation of a 3D hemispherical geometry on the top surface of the cylindrical solid model

The hemispherical feature was duplicated onto the second cylindrical solid using the Mirror feature. For this purpose, an intermediate plane required for mirroring was first created using the Reference Geometry option in the Features tab. The Front Plane was selected and offset by 57 mm to generate the new mirror plane (Figure 4-12). Subsequently, the existing hemispherical feature and the mirror plane were selected, and the mirroring operation was performed using the Mirror command (Figure 4-13).

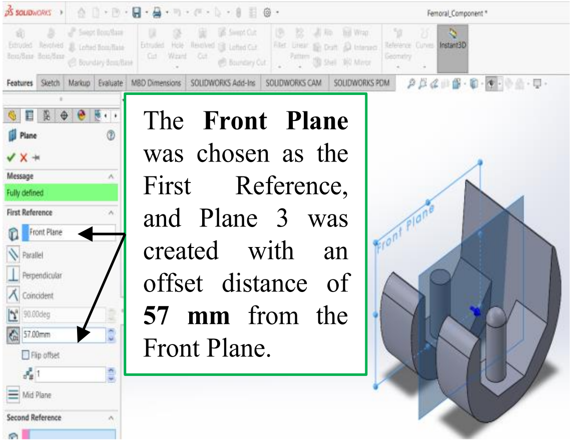


Figure 4-12. Establishment of the mirror plane on the Femoral Component solid model

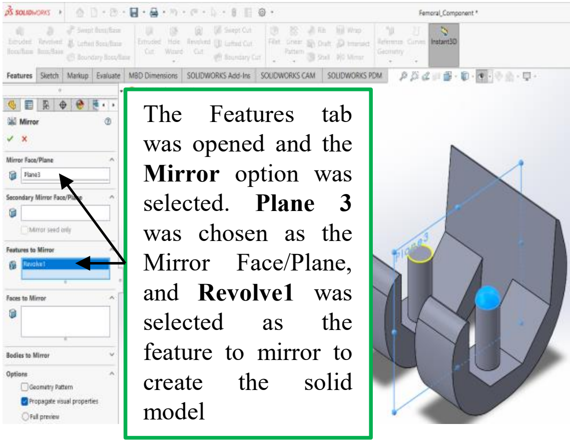


Figure 4-13. Mirroring of the hemispherical solid feature on the cylindrical model of the Femoral Component

In the next stage, based on the biological design of the knee region (cartilage seating cavity), a 2D design sketch was created on the Femoral Component (Figure 4-14). The regions defined by this sketch were then removed from the Femoral Component using the Extruded Cut operation (Figure 4-15).

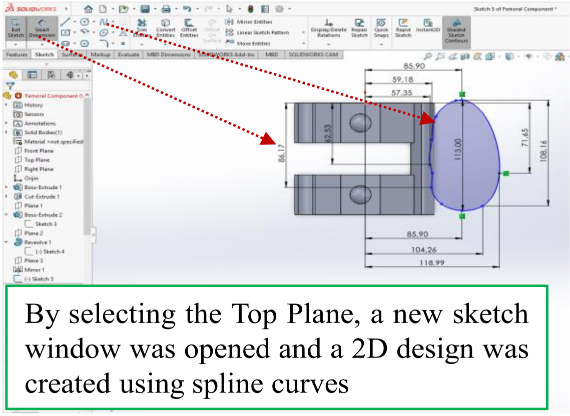


Figure 4-14. Creation of a 2D sketch on the top surface of the Femoral Component to define the cartilage cavity

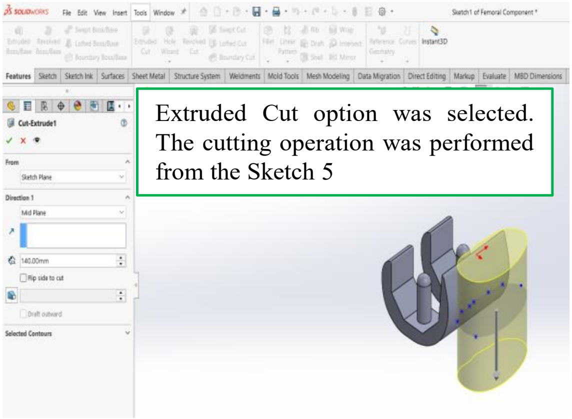


Figure 4-15. Performing an Extruded Cut based on the 2D sketch defined in Sketch 5

Following the cutting operation, a fillet with a radius of 15 mm was applied to the sharp edges to obtain a smoother and more continuous geometry (Figure 4-16).

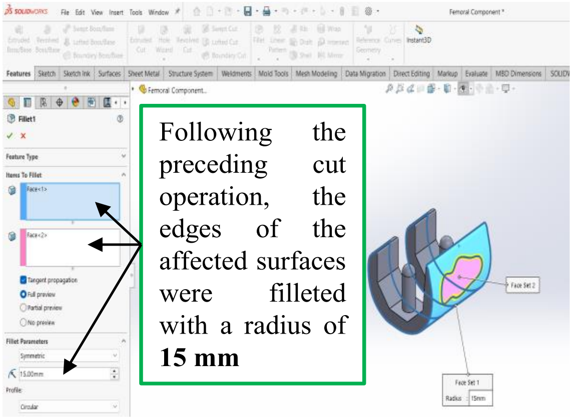


Figure 4-16. Application of a 15 mm radius fillet to the sharp corners on the cut surface of the Femoral Component

In the following step, a fillet with a radius of 1 mm was applied to the selected edges of the Femoral Component to further refine the geometry (Figure 4-17).

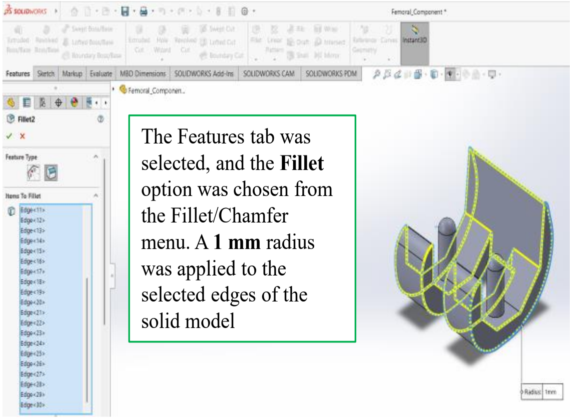


Figure 4-17. Application of a 1 mm radius fillet to the selected edges of the Femoral Component

In order to provide full conformity between the femur bone and the Femoral Component, individual reference planes were defined on the selected surfaces via the Reference Geometry feature under the Features tab (Figure 4-18).

Subsequently, a new sketch environment was opened on Plane 4, and a 2D sketch was created. This sketch was then used to perform an Extruded Cut operation with a depth of 4 mm (Figure 4-19). The purpose of this sketch design was to form a seating cavity that enables the femur bone to fit precisely onto the Femoral Component.

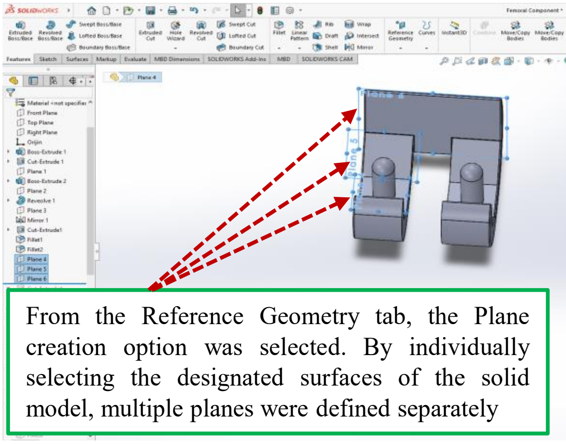


Figure 4-18. Establishment of new planes on the selected surfaces of the Femoral Component

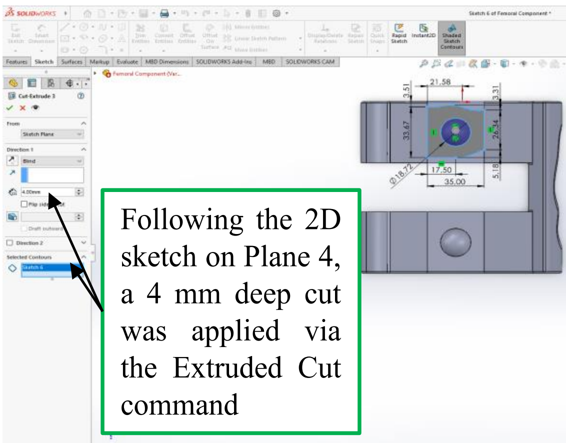


Figure 4-19. Creation of a 2D sketch on Plane 4 of the Femoral Component and application of the Extruded Cut feature

Another surface of the Femoral Component was selected, and a new reference plane was defined on this surface. Subsequently, a new sketch environment was opened on Plane 5, and after completing the 2D design, the Sketch 7 profile was cut using the Extruded Cut feature with a depth of 4 mm (Figure 4-20).

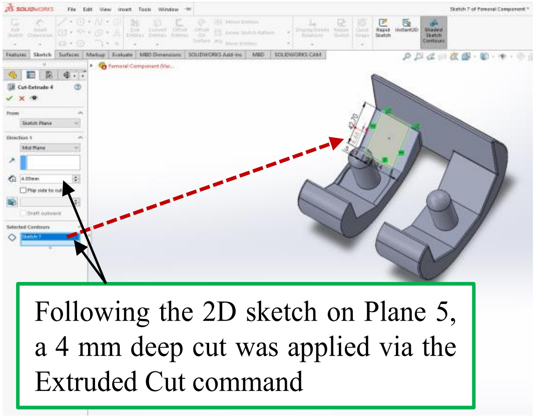


Figure 4-20. Creation of a 2D sketch on a newly defined reference plane generated on the selected surface of the femoral component, followed by geometry cut using the Extruded Cut command

Another surface of the Femoral Component was selected, and a new reference plane was defined on this surface. Subsequently, a new sketch environment was opened on Plane 6, and after completing the 2D sketch, Sketch 8 was cut to a depth of 4 mm using the Extruded Cut command (Figure 4-21). Similarly, an additional surface of the Femoral Component was selected, and another reference plane was defined on this surface. A new sketch environment was then created on Plane 7, and following the completion of the 2D sketch, Sketch 9 was cut to a depth of 4 mm using the Extruded Cut command (Figure 4-22).

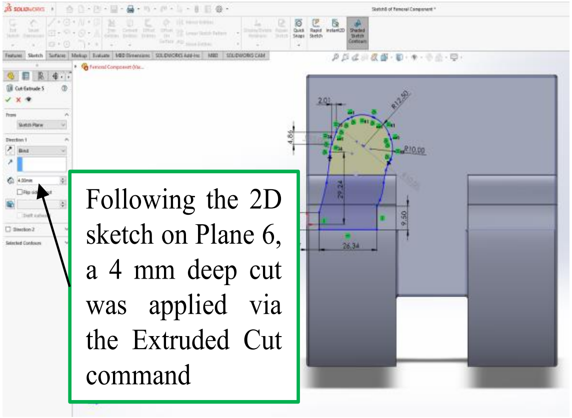


Figure 4-21. Creation of a 2D sketch on Plane 6, which was defined on the selected surface of the Femoral Component, followed by cutting using the Extruded Cut command

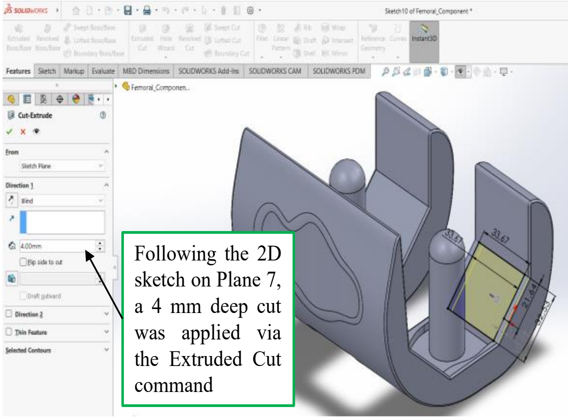


Figure 4-22. Creation of a 2D sketch on Plane 7, which was defined on the selected surface of the Femoral Component, followed by cutting using the Extruded Cut command.

Another surface of the Femoral Component was selected, and a new reference plane was defined on this surface. Subsequently, a new sketch environment was opened on Plane 8, and after completing the 2D sketch, Sketch 10 was cut to a depth of 4 mm using the Extruded Cut command (Figure 4-23).

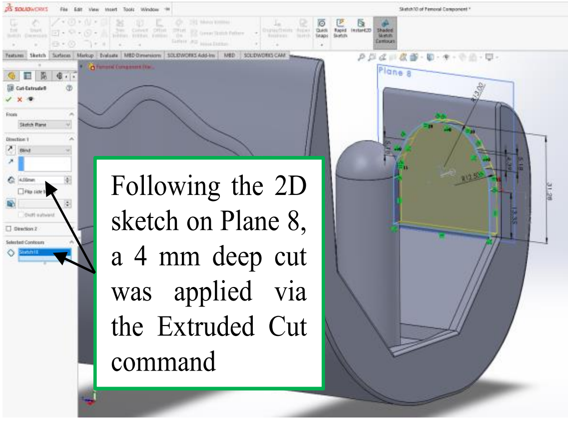


Figure 4-23. Creation of a 2D sketch on Plane 8, which was defined on the selected surface of the Femoral Component, followed by cutting using the Extruded Cut command

A lateral surface within the newly created housing seat region of the Femoral Component was selected, and a new reference plane was defined on this surface (Figure 4-24).

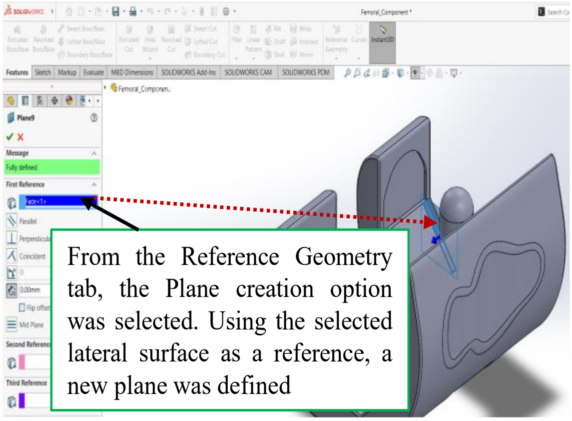


Figure 4-24. Creation of a new reference plane on the selected surface within the housing seat region of the Femoral Component

Subsequently, a new sketch environment was opened on the defined plane, and a triangular 2D sketch was created to refine the connection regions (Figure 4-25). The resulting 2D sketch was then cut up to the selected face using the Extruded Cut command (Figure 4-26).

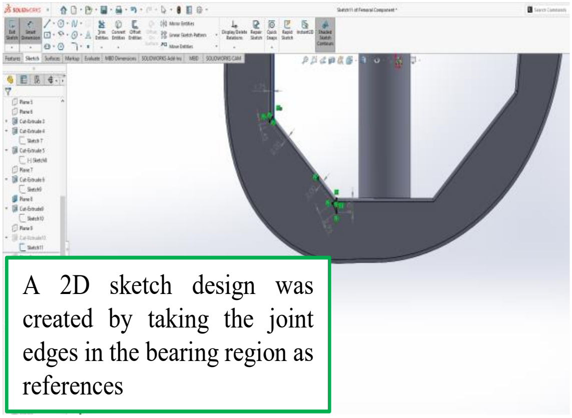


Figure 4-25. Opening a new sketch environment on Plane 9 of the Femoral Component and creating a 2D sketch at the transition regions of the housing seat areas

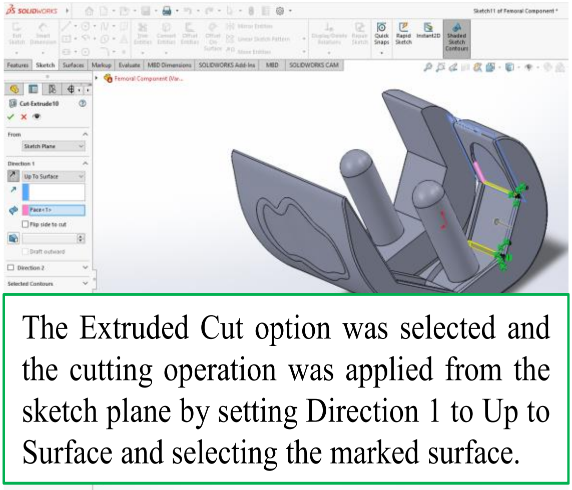


Figure 4-26. Cutting the 2D sketch created at the transition regions of the housing seat areas of the Femoral Component using the Extruded Cut command

Another lateral surface within the newly created housing seat region of the Femoral Component was selected, and a new reference plane was again defined on this surface (Figure 4-27). Subsequently, a new sketch environment was opened on the defined plane, and a triangular 2D sketch was created to refine the connection regions (Figure 4-28). The resulting 2D sketch was then cut up to the selected face using the Extruded Cut command (Figure 4-29).

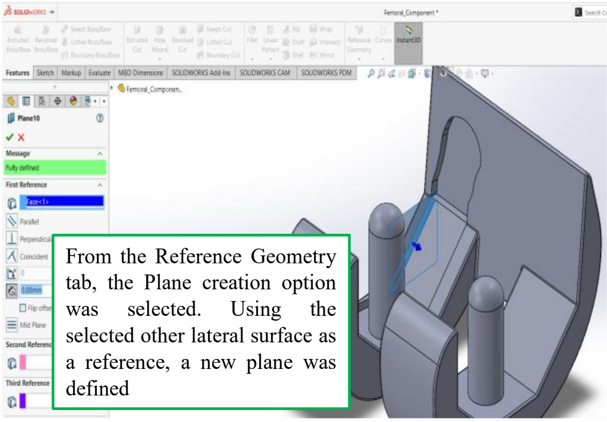


Figure 4-27. Creation of a new reference plane on the other selected surface within the seat region of the Femoral Component

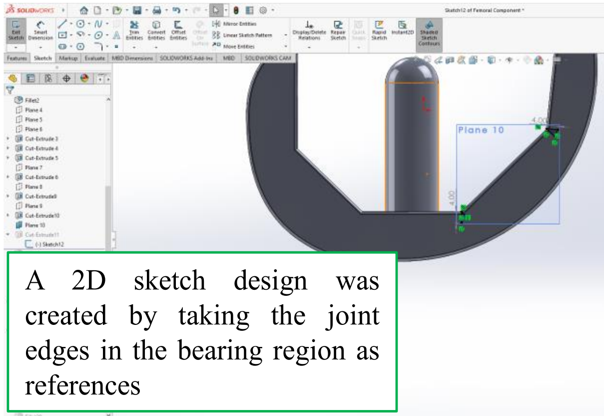


Figure 4-28. Opening a new sketch environment on Plane 10 of the Femoral Component and creating a 2D sketch at the transition regions of the socket bearing areas

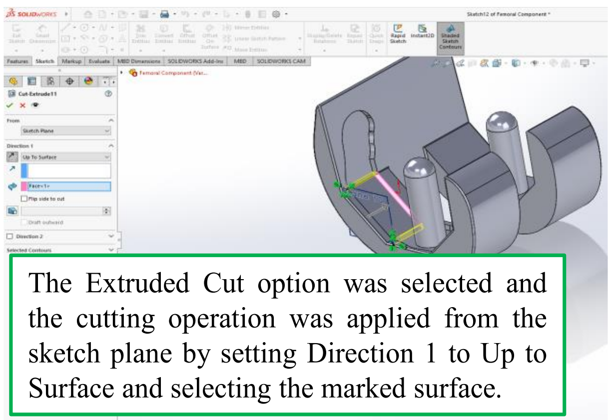


Figure 4-29. Cutting the 2D sketch created at the transition regions of the socket bearing areas of the Femoral Component using the Extruded Cut command

Subsequently, to provide a smooth transition at the edges formed in the cut transition regions of the socket bearing areas, a 3 mm radius was applied to the selected edges using the Fillet feature located in the Features tab (Figure 4-30).

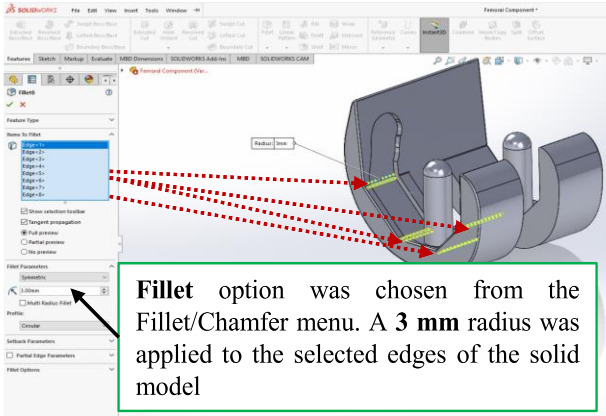


Figure 4-30. Applying a 3 mm radius to the selected edges at the transition regions of the socket bearing areas of the Femoral Component

In the subsequent stage, it was planned to create a new reference plane for a single section of the fork region of the Femoral Component that engages the femur bone from the posterior side. For this purpose, the Reference Geometry feature in the Features tab was selected to initiate plane creation, and a new plane was defined based on the specified reference points (Figure 4-31).

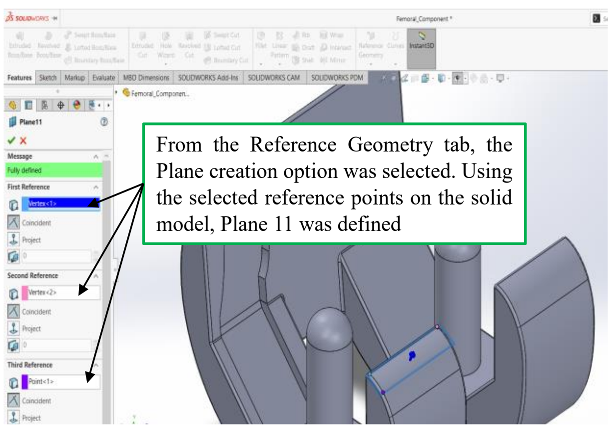


Figure 4-31. Creation of a new reference plane on the fork region of the Femoral Component based on the selected reference points

Subsequently, a new sketch environment was opened on the created plane, with the objective of increasing the contact surface between the Femoral Component and the femur bone in this region. For this purpose, the 2D sketch was cut using the Extruded Cut operation (Figure 4-32).

After the cutting operation, sharp corners were observed in the modified region. Therefore, a 3 mm radius was applied to the selected edges using the Fillet command (Figure 4-33).

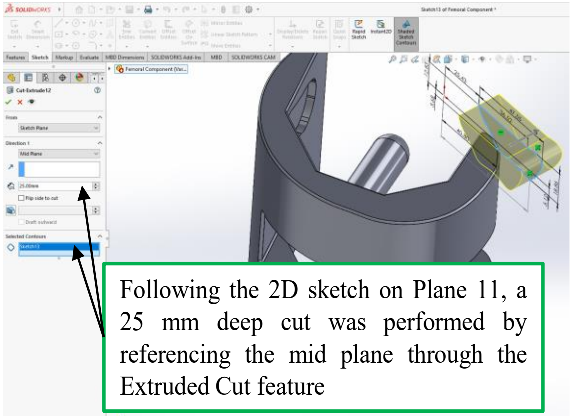


Figure 4-32. Creating a 2D sketch on the fork region of the Femoral Component and performing a cut using the Extruded Cut command

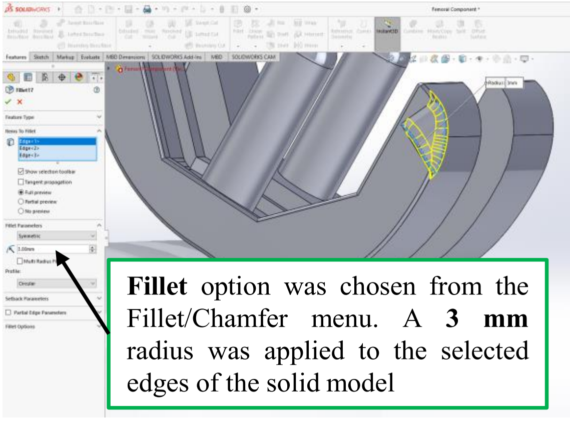


Figure 4-33. Applying a 3 mm radius to the sharp edges in the fork region of the Femoral Component after the cutting operation using the Fillet command

Subsequently, a 3 mm radius was also applied to both the cut region and the specified edges of the existing solid model (Figure 4-34).

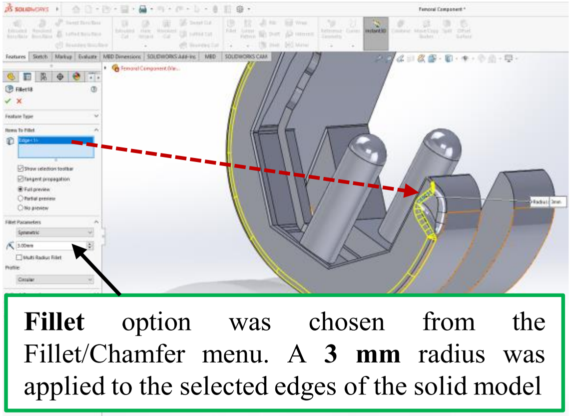


Figure 4-34. Applying a 3 mm radius to the selected sharp edges in the fork region of the Femoral Component using the Fillet command

In the next stage, a new sketch environment was opened for the other section of the fork region of the Femoral Component, and the 2D sketch created in this area was cut using the Extruded Cut operation (Figure 4-35).

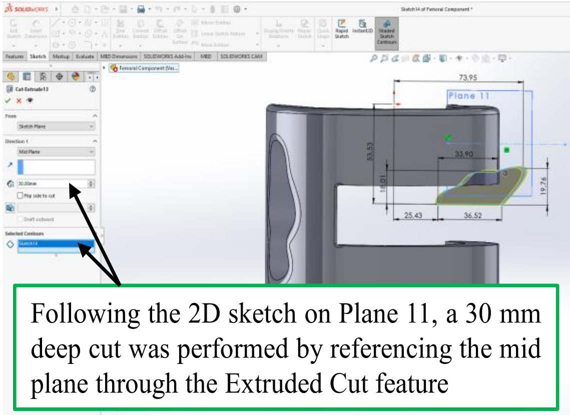


Figure 4-35. Creating a 2D sketch on the other end of the fork region of the Femoral Component and performing a cut using the Extruded Cut command

After the cutting operation, sharp corners were observed in the modified region. Therefore, a 3 mm radius was applied to the selected edges using the Fillet command (Figure 4-36). In the previous operation, applying a radius was not possible for some edges due to geometric interference; this was addressed as shown in Figure 4-37.

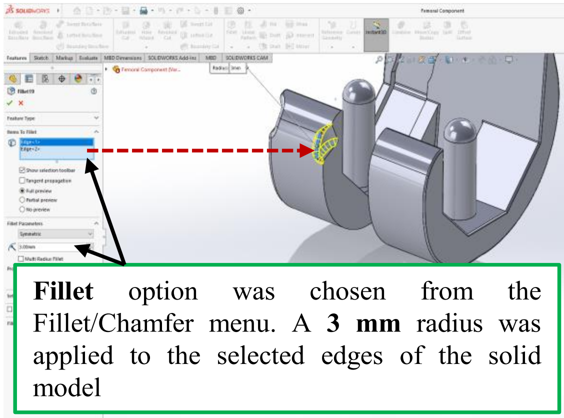


Figure 4-36. Applying a 3 mm radius to the selected sharp edges on the other end of the fork region of the Femoral Component using the Fillet command

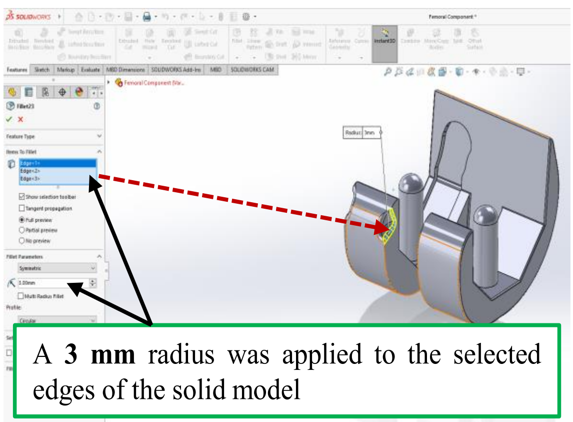


Figure 4-37. Applying a 3 mm radius to the selected sharp edges on the other end of the fork region of the Femoral Component

In the next step, a 3 mm radius was applied to the selected edges following the internal channel region of the Femoral Component using the Fillet command located in the Features tab (Figure 4-38).

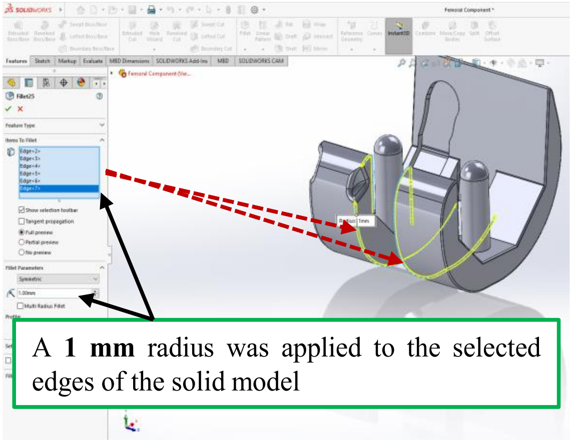


Figure 4-38. Applying a 3 mm radius to the selected edges in the internal channel region of the Femoral Component using the Fillet command

In the next step, since the Femoral Component has two separate fork regions, it was planned to replicate the designs created on the processed fork region. For this purpose, the Mirror command was used to duplicate the operations performed on the first fork region, using Plane 3 as the mirror reference (Figure 4-39). Subsequently, it was planned to apply fillets to the sharp edges in the design of the mirrored second fork region of the Femoral Component. For this purpose, a 3 mm radius was applied to the selected edges using the Fillet command (Figure 4-40).

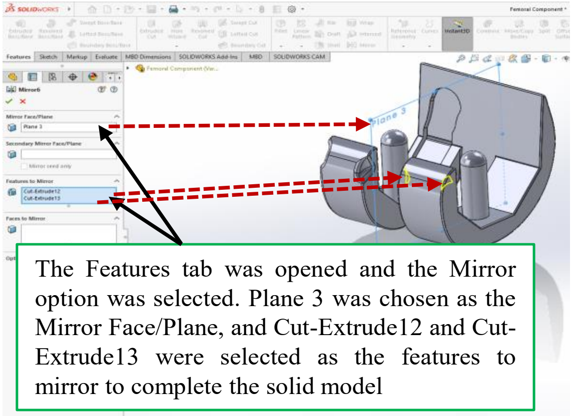


Figure 4-39. Mirroring the designs applied to the previous fork region to create the other fork region of the Femoral Component

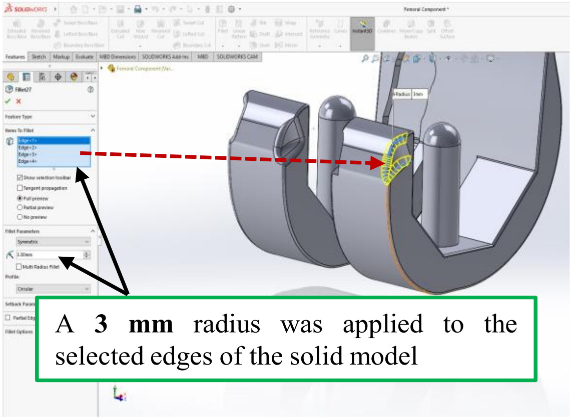


Figure 4-40. Applying a 3 mm radius to the selected sharp edges at the tip of the mirrored fork region of the Femoral Component

In the previous operation, applying a radius to the selected edge was not possible due to geometric interference; this was addressed as shown in Figure 4-41. As a result, the outer lateral edge of the existing solid model was also assigned a 3 mm radius (Figure 4-41).

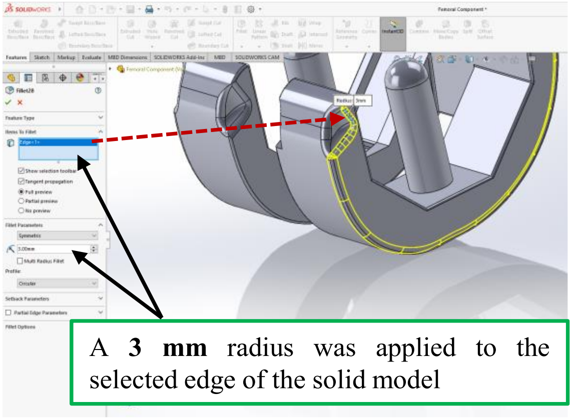


Figure 4-41. Applying a 3 mm radius to the previously unfilletable sharp edge at the tip of the mirrored fork region of the Femoral Component

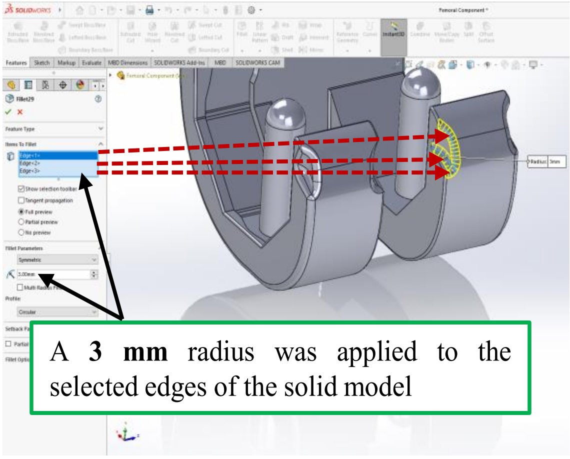


Figure 4-42. Applying a 3 mm radius to the selected sharp edges at the other end of the mirrored fork region of the Femoral Component

After the cutting operation, sharp corners were observed at the other end of the mirrored fork region of the Femoral Component. Therefore, a 3 mm radius was applied to the selected edges using the Fillet command (Figure 4-42). In the previous operation, applying a radius to certain edges was not possible due to geometric interference; this was addressed as shown in Figure 4-43.

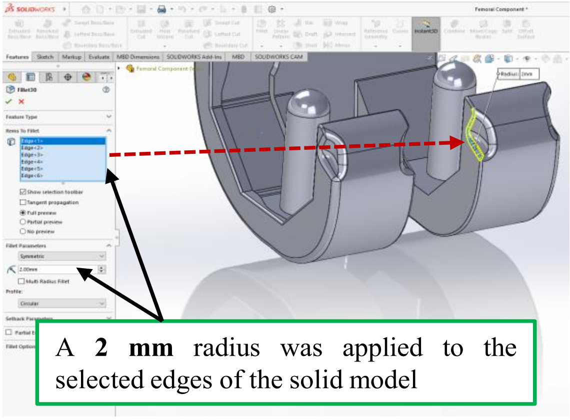


Figure 4-43. Applying a 2 mm radius to the selected sharp edges at the tip of the mirrored fork region of the Femoral Component

In the next step, a new sketch environment was opened on the previously created Plane 6 of the Femoral Component, and after completing the 2D sketch, Sketch 17 was cut to a depth of 4 mm using the Extruded Cut command (Figure 4-44).

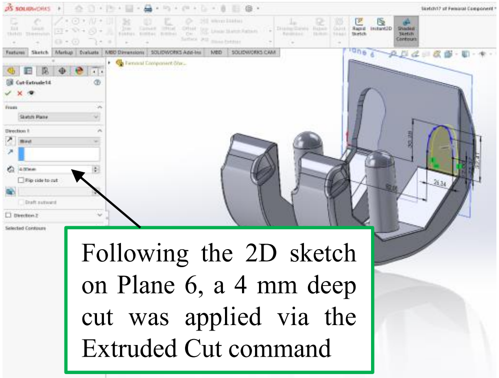


Figure 4-44. Creating a 2D sketch on Plane 6, defined on the selected surface of the Femoral Component, and performing a cut using the Extruded Cut command

In the next step, the socket bearing designs created on the first fork region of the Femoral Component, which engage the femur bone, were mirrored to the other fork region using the Mirror command (Figure 4-45). This included all modeling operations performed with Extruded Cut and Fillet. During the mirroring process, the Mirror command in the Features tab was used, with Plane 3 serving as the mirror reference.

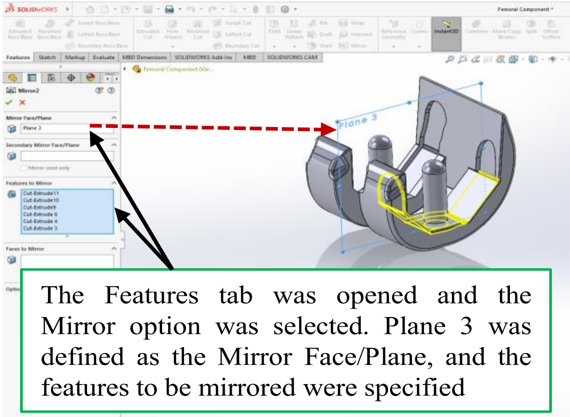


Figure 4-45. Mirroring the socket bearing designs applied to the previous fork region to create similar designs on the other fork region of the Femoral Component

In the next step, one of the lateral surfaces in the channel region of the Femoral Component was selected, and a new reference plane was created on this surface (Figure 4-46). Subsequently, a sketch environment was opened on the newly created Plane 12, and the first 2D sketch of the intermediate solid model connecting the two fork regions was created (Figure 4-47).

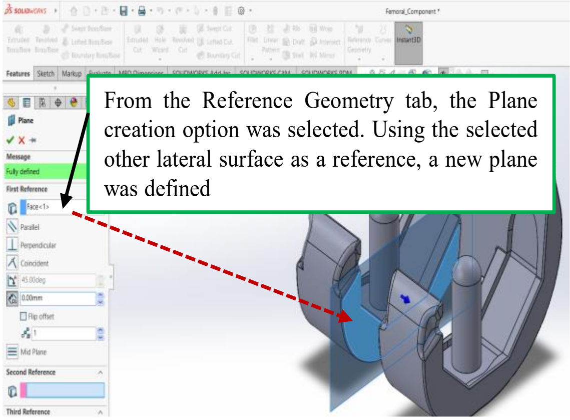


Figure 4-46. Creation of a new reference plane on the selected lateral surface in the channel region of the Femoral Component

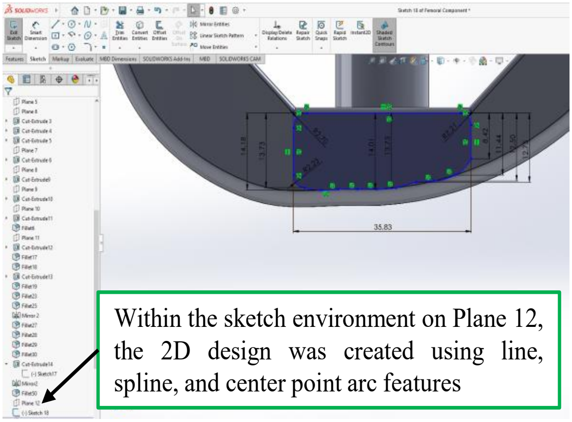


Figure 4-47. Creating the 2D sketch for the solid model that connects the fork sections in the channel region of the Femoral Component

Subsequently, the 2D sketch was selected, and a solid model was created up to the lateral surface of the other fork region using the Extruded Boss/Base command (Figure 4-48).

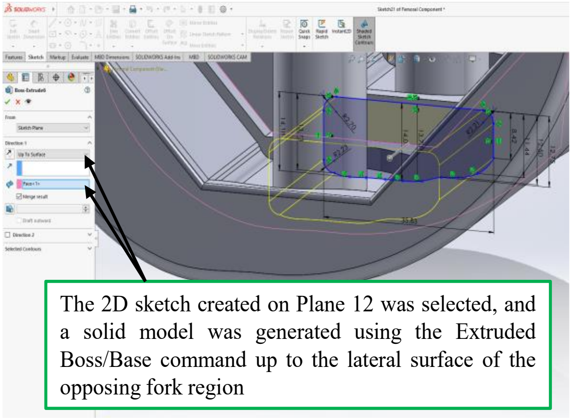


Figure 4-48. Converting the 2D sketch into a 3D solid model to connect the fork sections in the channel region of the Femoral Component

In the next step, design modifications were planned on the solid model connecting the fork sections in the channel region of the **Femoral Component** to accommodate the contact areas of the femur bone. For this purpose, the surface of the connecting solid model was selected, and a reference plane was defined (Figure 4-49).

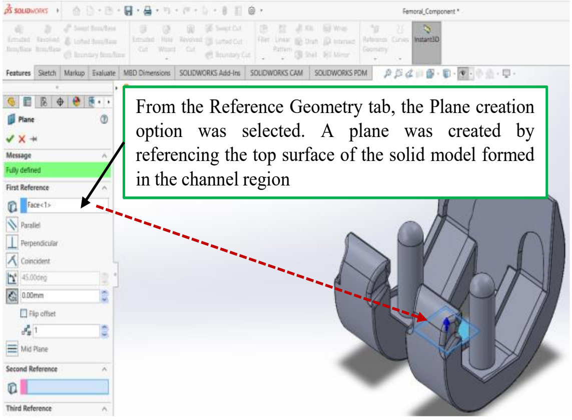


Figure 4-49. Creating a reference plane on the top surface of the solid model in the channel region of the Femoral Component

Subsequently, a new sketch environment was opened on the created plane, and a 2D sketch was drawn according to the femur bone contact design. The 2D sketch created on the connecting solid region in the channel area was then selected, and a complete cut in both directions was performed using the Extruded Cut command (Figure 4-50).

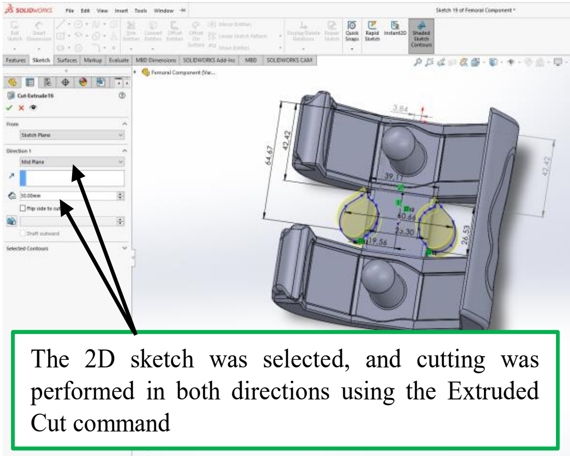


Figure 4-50. Applying the Extruded Cut operation to the 2D sketch created on Plane 13

Subsequently, Plane 12 was selected, and a new sketch environment was opened. On this plane, a 2D sketch was created to define the contact surface between the Femoral Component and the Plastic Spacer. The 2D sketch representing the contact region was then selected, and a through cut was performed using the Extruded Cut command (Figure 4-51).

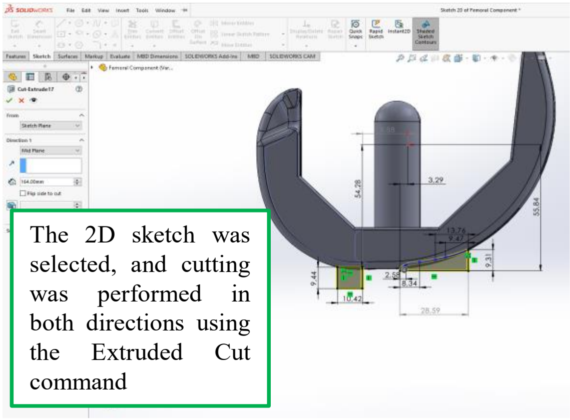


Figure 4-52. Applying the Extruded Cut operation to the 2D sketch defining the contact surface region between the Femoral Component and the Plastic Spacer

In the next step, a 2.5 mm radius was applied to the sharp edges formed after cutting the connecting solid region in the intermediate channel area of the Femoral Component using the Fillet command in the Features tab. This operation smoothed the sharp edges in this region (Figure 4-53).

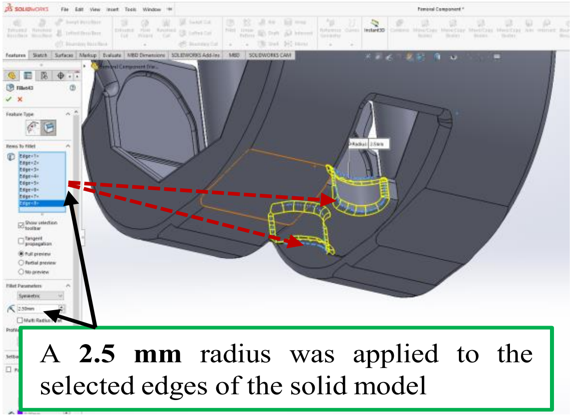


Figure 4-53. Applying a 2.5 mm radius to the existing sharp edges in the solid region of the channel area of the Femoral Component using the Fillet command

After completing the described operations, the solid modeling process of the Femoral Component was finalized. The final appearance of the Femoral Component is presented in Figure 4-54.

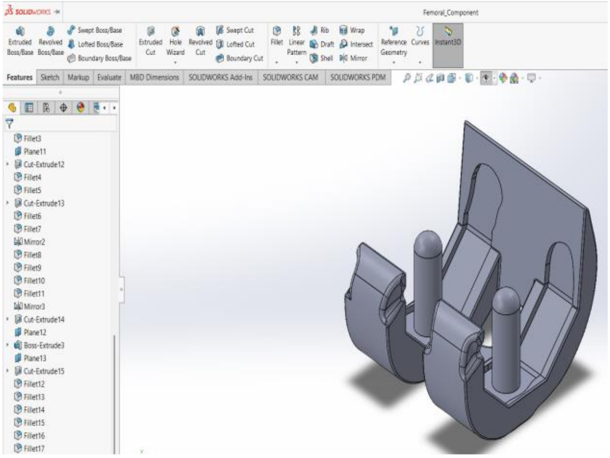


Figure 4-54. Final appearance of the completed Femoral Component

In the knee prosthesis, our second design model is the Plastic Spacer component. For the planned Plastic Spacer design, a new part file was first opened in the solid modeling software. At the beginning of the study, the part file was named according to the planned project, and it was labeled as “Plastic Spacer”. To create the planned sketch in this part file, the Top Plane was first selected from the feature tree located on the left side of the solid modeling interface, and a new sketch environment was opened from this plane. These steps are illustrated in detail in Figure 4-55.

Subsequently, in the opened sketch environment, the initial 2D sketch of the planned Plastic Spacer model was created according to the projection of the Femoral Component (Figure 4-56).

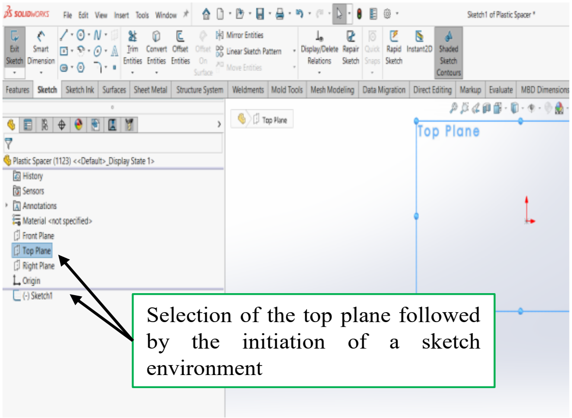


Figure 4-55. Selection of the front plane and creation of the drawing window for Plastic Spacer Design

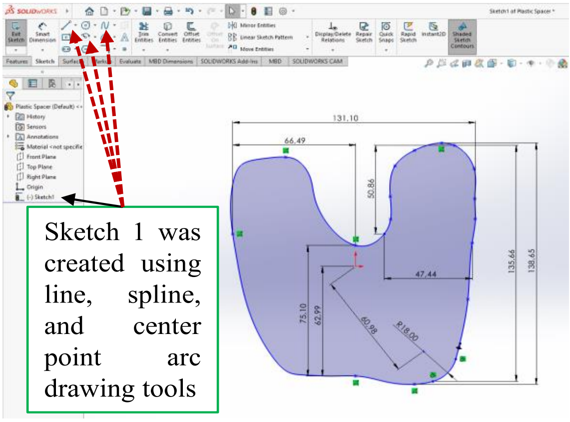


Figure 4-56. The initial stage of the 2D design of the Plastic Spacer model

In the next step, to convert the 2D sketch lines into a solid model, the sketch file was selected, and the Extruded Boss/Base command in the Features tab was used to create the solid. For the current modeling, a thickness of 32.60 mm was applied to obtain the initial solid model design (Figure 4-57).

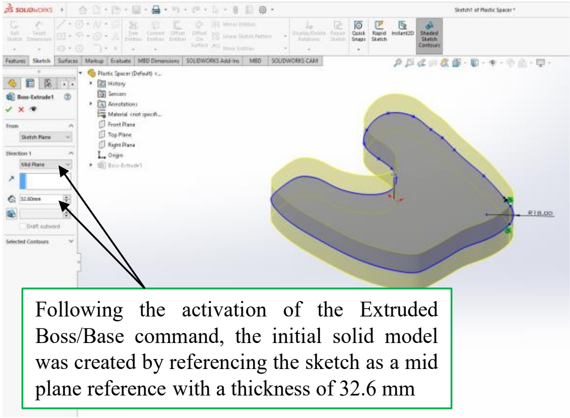


Figure 4-57. Initial development phase of the Plastic Spacer's 3D solid model

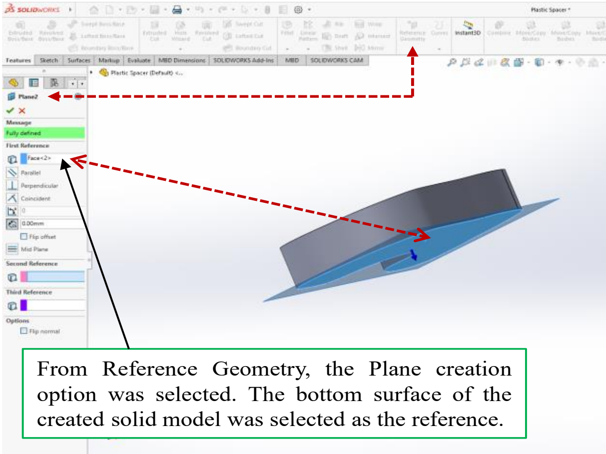


Figure 4-58. Creating a new reference plane on the bottom surface of the Plastic Spacer component

The bottom surface of the created solid model was then selected, and a new reference plane was defined. This operation was performed using the Reference Geometry feature in the Features tab (Figure 4-58). Subsequently, a sketch environment was opened on the created Plane 1.

In this sketch, the design from Sketch 1 was copied. Additionally, by offsetting inward by 4 mm, two nested sketches were created (Figure 4-59). The new sketch created in Sketch

2 was then selected, and a 4 mm cut was performed using the Extruded Cut command (Figure 4-60).

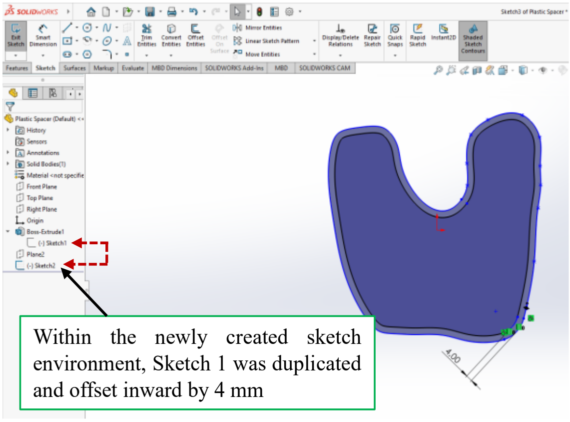


Figure 4-59. Creating a 2D sketch on the reference plane at the bottom surface of the Plastic Spacer component

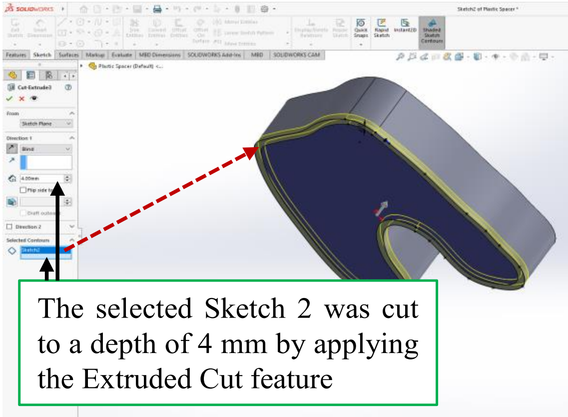


Figure 4-60. Performing a 4 mm cut on Sketch 2 of the Plastic Spacer component using the Extruded Cut command

In the next step, a new reference plane was created on the bottom surface of the Plastic Spacer to define the region that will mate with the Tibial Component (Figure 4-61).

A sketch environment was then opened on this plane, and 2D sketches were created (Figure 4-62).

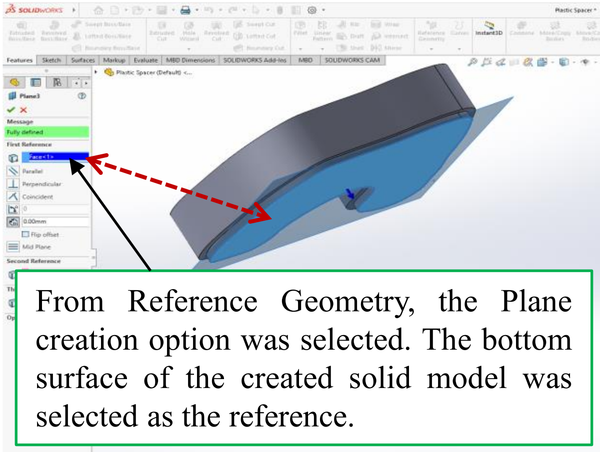


Figure 4-61. Creating a new reference plane on the bottom surface of the Plastic Spacer component

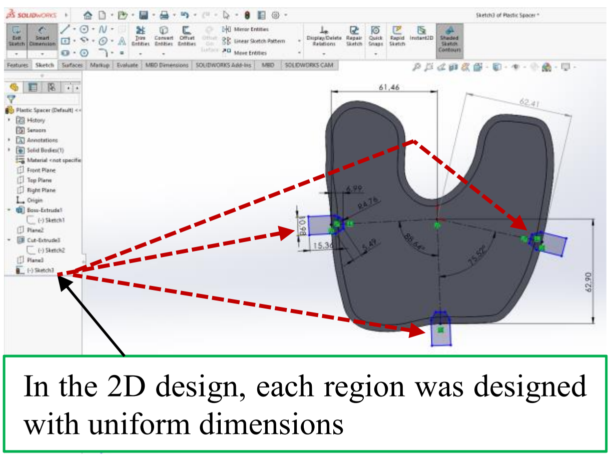


Figure 4-62. Creating the 2D sketch for the mating region with the Tibial Component on the Plastic Spacer component

The created 2D sketches were subsequently selected, and a cut with a depth of 8.4 mm was performed using the Extruded Cut command (Figure 4-63).

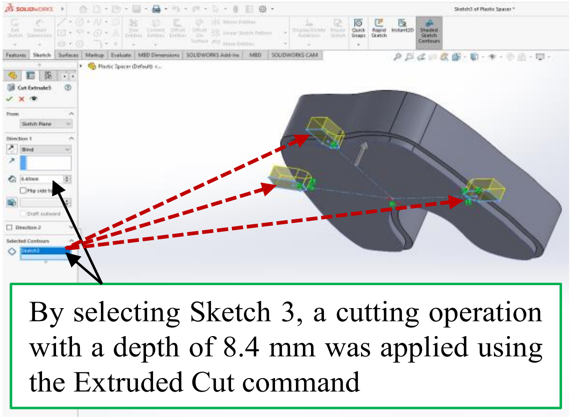


Figure 4-63. Performing an 8.4 mm deep cut on Sketch 3 of the Plastic Spacer component using the Extruded Cut command.

To soften the sharp edges on the top surface of the **Plastic Spacer** component, the **Fillet** feature from the **Features** tab was used. One edge on the top surface was selected, and a **2.1 mm radius** was applied. After selecting a single edge, the **Tangent** option was checked to apply the same radius to all connected edges on the top surface of the **Plastic Spacer** (Figure 4-64).

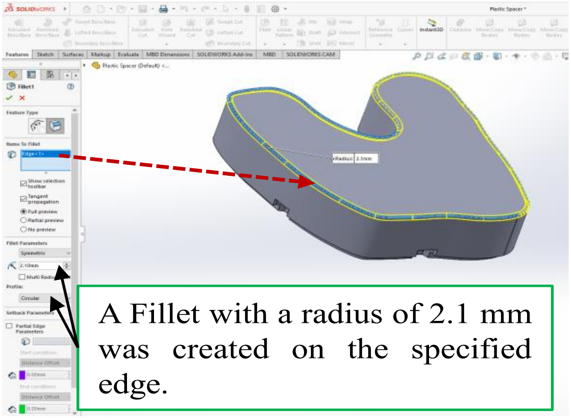


Figure 4-64. Applying a 2.1 mm radius to the sharp edges on the top surface of the Plastic Spacer component using the Fillet command

Subsequently, a new reference plane was created on the top surface of the Plastic Spacer component, using the top surface and selected reference points on the Plastic Spacer as references (Figure 4-65).

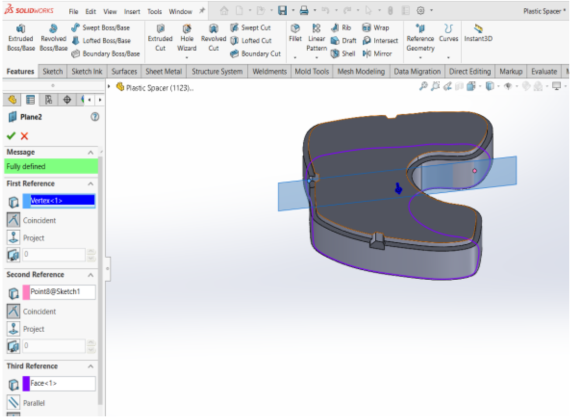


Figure 4-65. Creating a new reference plane on the Plastic Spacer component using the selected reference points and reference Surface

In the next step, considering the biological design details of the knee region, it was planned to cut a specific section of the Plastic Spacer component. For this purpose, a sketch environment was opened on the newly created Plane 2, and a 2D sketch was created (Figure 4-66). The sketch in Sketch 4 was then selected, and a through cut was performed using the Extruded Cut command (Figure 4-67).

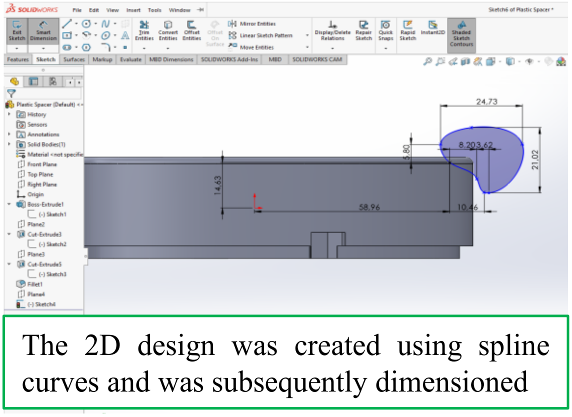


Figure 4-66. Creating a 2D sketch on the Plastic Spacer component for modification purposes

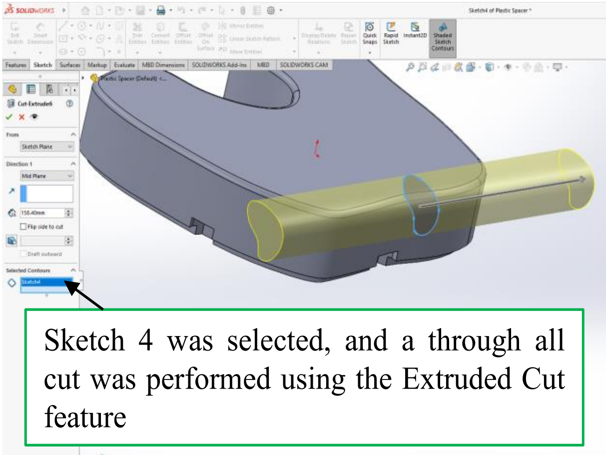


Figure 4-67. Applying the Extruded Cut operation to Sketch 4 of the Plastic Spacer component

To achieve a smoother design on the edges formed in the cut region, a 5 mm radius was applied using the Fillet command (Figure 4-68).

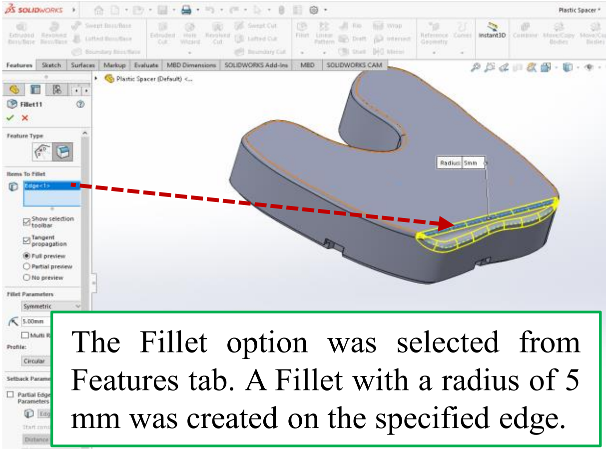


Figure 4-68. Applying a 5 mm radius to the sharp edges formed after the third Extruded Cut operation on the Plastic Spacer component using the Fillet command

In the next step, a new reference plane was created on the top surface of the Plastic Spacer component (Figure 4-69). A sketch environment was then opened on this plane, and two circular geometries, identical to the design in the Femoral Component, were drawn (Figure 4-70). The purpose of this step is to ensure that, in the subsequent process when the Femoral Component is inserted into this part file, the contact seating surface on the top of the Plastic Spacer aligns correctly. The drawn circular geometries will provide a centering relationship with the cylindrical solid features in the Femoral Component.

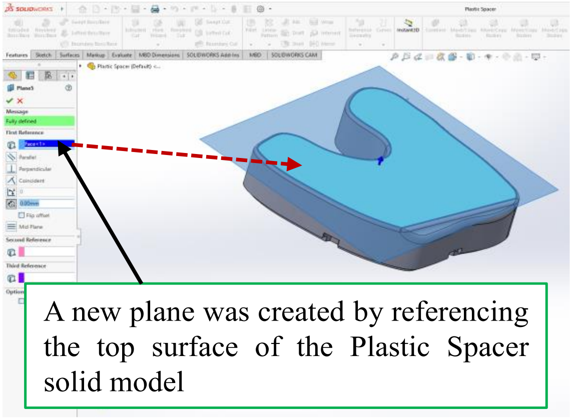


Figure 4-69. Establishment of a new plane on the top surface of the Plastic Spacer

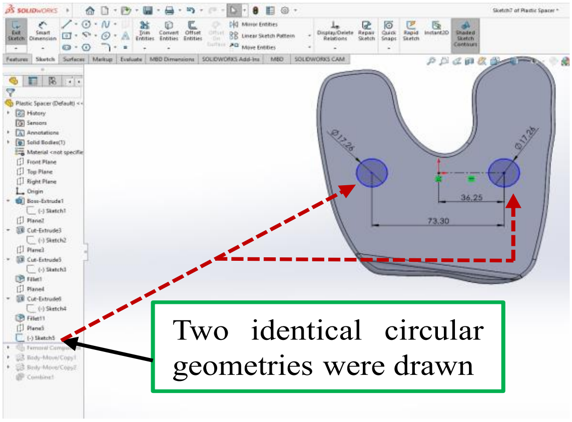


Figure 4-70. Sketching of two circular profiles on Plane 3 of the Plastic Spacer

The Femoral Component was imported into the Plastic Spacer part file to define its seating contact region. Using the Move/Copy Bodies feature, the lateral cylindrical surfaces and corresponding circular geometries were selected to establish proper centering between the parts (Figure 4-71). The same feature was then applied to offset the Femoral Component 3.4 mm into the Plastic Spacer (Figure 4-72).

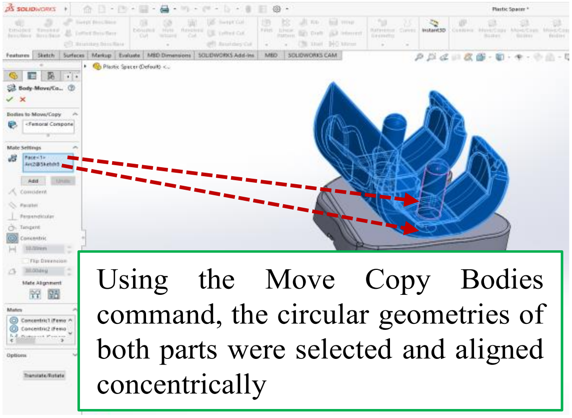


Figure 4-71. Application of the Move/Copy Bodies command to center the Femoral Component with respect to the Plastic Spacer

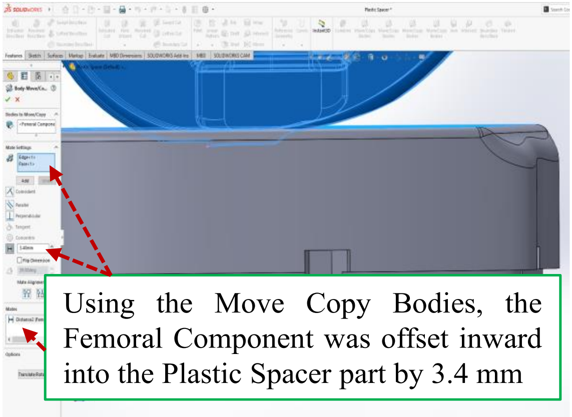


Figure 4-72. Applying the Move/Copy Bodies command to translate the Femoral Component 3.4 mm within the Plastic Spacer

The Femoral Component, previously aligned and centered with respect to the Plastic Spacer, was subtracted using the Combine feature, available in the Features tab (SolidWorks 2022) or the Mesh Modeling tab (SolidWorks 2025) (Figure 4-73). During the subtraction process, the Combine command was activated, the Plastic Spacer was designated as the main body, and the Femoral Component was specified as the body to be removed.

Consequently, a seating contact surface for the Femoral Component was established on the Plastic Spacer, and the final completed solid model of the Plastic Spacer was obtained (Figure 4-74).

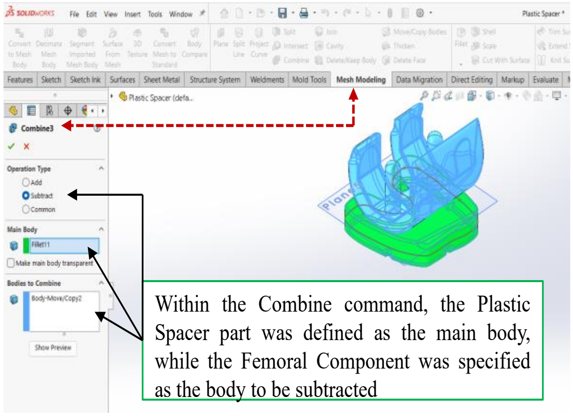


Figure 4-73. Application of the Combine feature to subtract the Femoral Component from the Plastic Spacer

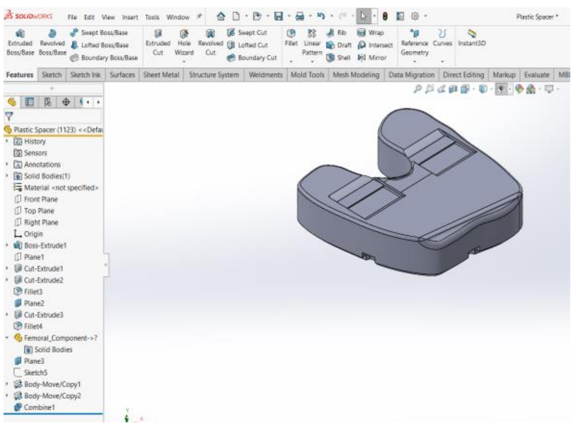


Figure 4-74. Completed view of the Plastic Spacer showing the seating contact region for the Femoral Component

The third design model in this study pertains to the Tibial Component of the knee prosthesis. Initially, a part file was created in the solid modeling software and named “Tibial Component” following the project designation.

To develop the planned sketch, the top plane was selected from the Feature Tree, and a new sketch environment was initiated (Figure 4-75). Using the projection of the Plastic Spacer, the initial 2D sketch of the Tibial Component was generated (Figure 4-76).

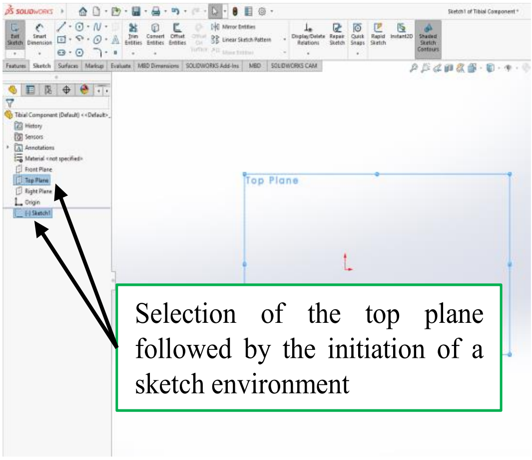


Figure 4-75. Selection of the front plane and creation of the drawing window for Tibial Component Design

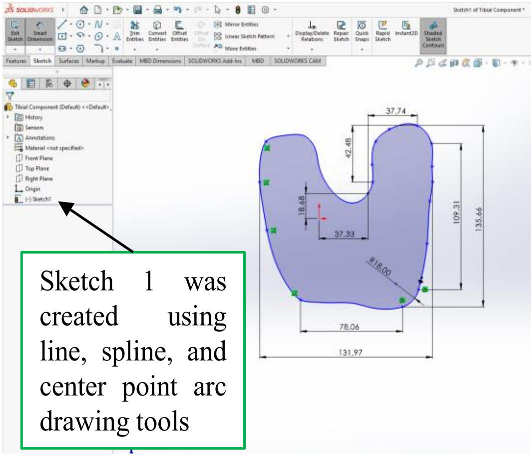


Figure 4-76. The initial stage of the 2D design of the Tibial Component model

The 2D sketch was then converted into a solid model by selecting the sketch and applying the Extruded Boss/Base command with a thickness of 33.85 mm, resulting in the first solid model of the component (Figure 4-77).

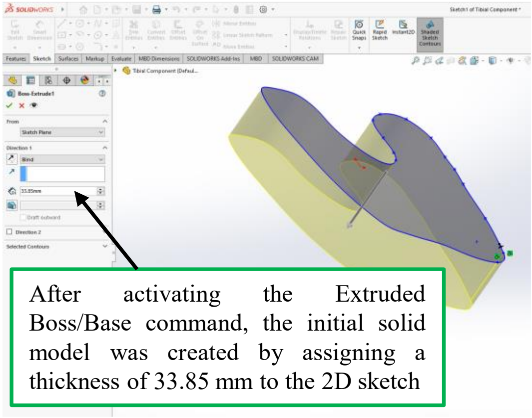


Figure 4-77. Initial development phase of the Tibial Component's 3D solid model

In the next step, the contact regions between the Tibial Component and the Plastic Spacer were planned to be established. For this purpose, the Plastic Spacer part was referenced in the existing model, and it was aligned and positioned to create proper contact with the Tibial Component (Figure 4-78).

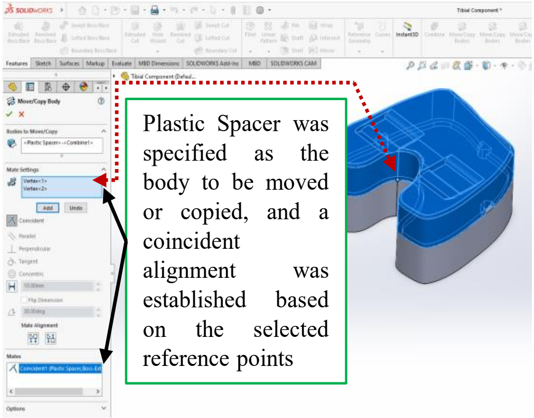


Figure 4-78. Contact relationships between the Tibial Component and the Plastic Spacer

Subsequently, in order to create the seating cavity for the Plastic Spacer on the Tibial Component, the Combine feature in the Features tab was used to subtract the Plastic Spacer from the Tibial Component (Figure 4-79).

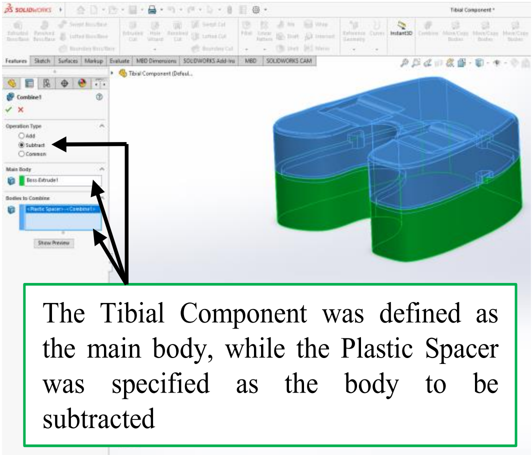


Figure 4-79. Application of the Combine–Subtract command to remove the Tibial Component body from the Plastic Spacer body

In the next step, a new plane was created using the Reference Geometry tool located in the Features tab. To this end, the bottom surface of the Tibial Component was selected as the reference (Figure 4-80).

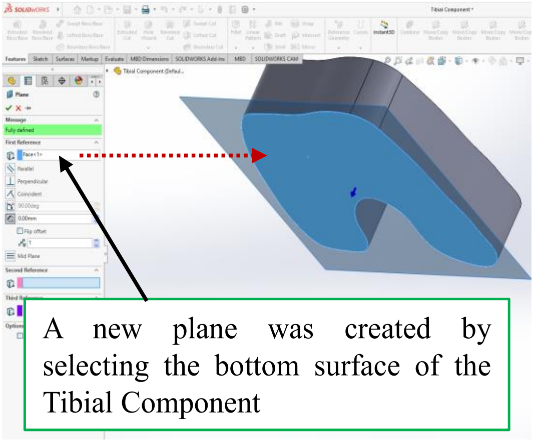


Figure 4-80. Creation of a new reference plane on the bottom surface of the Tibial Component part.

Thereafter, a new plane was defined 48.76 mm away from Plane 1, taken as the reference (Figure 4-81).

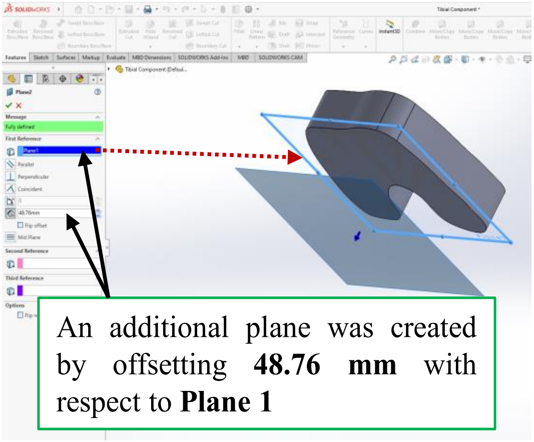


Figure 4-81. Creation of a new reference plane offset 48.76 mm from Plane 1

A pentagonal geometry was created on the sketch plane associated with Plane 1 (Figure 4-82).

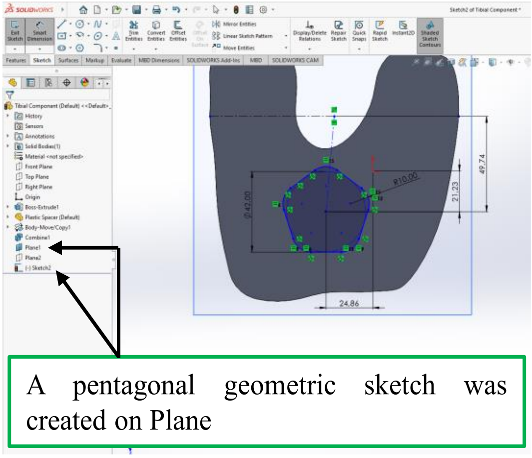


Figure 4-82. Creation of a pentagonal sketch on Plane 1

A circular sketch was created on Plane 2 (Figure 4-83). These procedures were carried out using the geometric sketch features located in the Sketch tab.

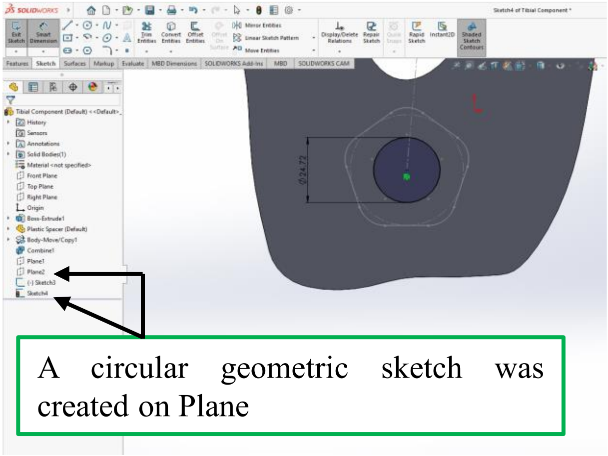


Figure 4-83. Creation of a circular sketch on Plane 2

Subsequently, a solid body was created by applying the Loft feature to the sketches defined on two distinct planes (Figure 4-84).

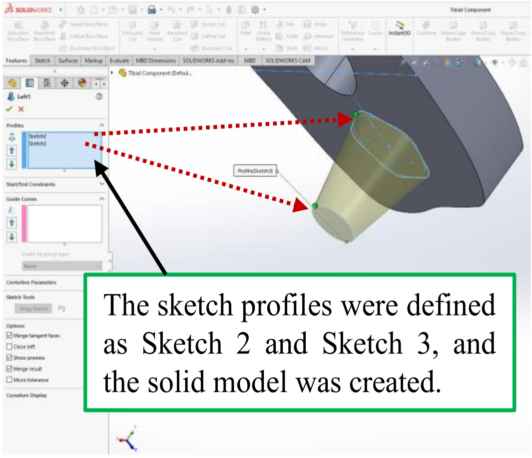


Figure 4-84. Creation of a solid model using the Loft feature from sketches on two different planes

In order to improve the mechanical integrity of the lofted part, reinforcement regions were then designed. Accordingly, the sketch plane on Plane 1 was activated to create the first support profile, which was subsequently revolved using the Revolved Boss/Base feature to form a solid reinforcement (Figure 4-85).

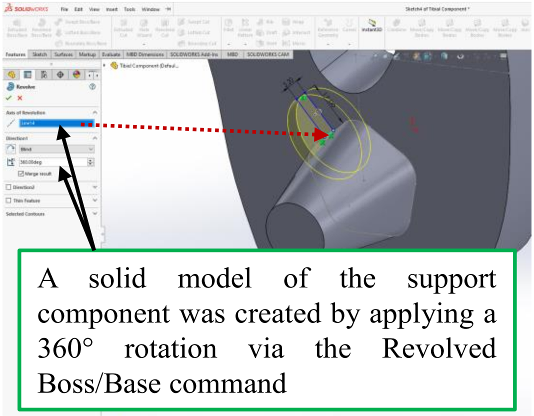


Figure 4-85. Creation of a solid support feature by revolving a rectangular sketch using the Revolved Boss/Base command

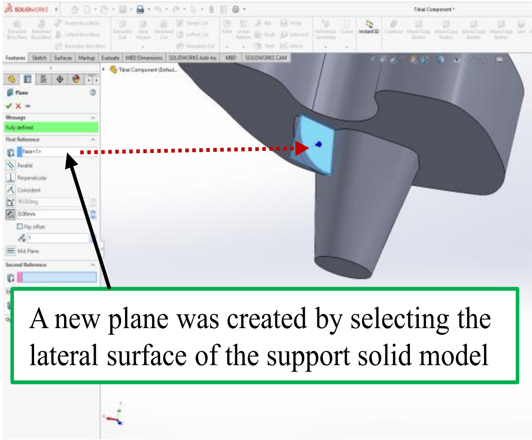


Figure 4-86. Definition of a new plane on the lateral face of the solid support model

While performing the revolved solid modeling operation, one of the short edges of the rectangular sketch was specified as the axis of revolution. Thereafter, a new plane was generated by selecting the lateral face of the solid support feature (Figure 4-86). To convert the support solid model into a triangular form, a semicircular sketch was created covering the specified regions. The cutting operation was then carried out by selecting Sketch 5 and applying the Extruded Cut feature (Figure 4-87).

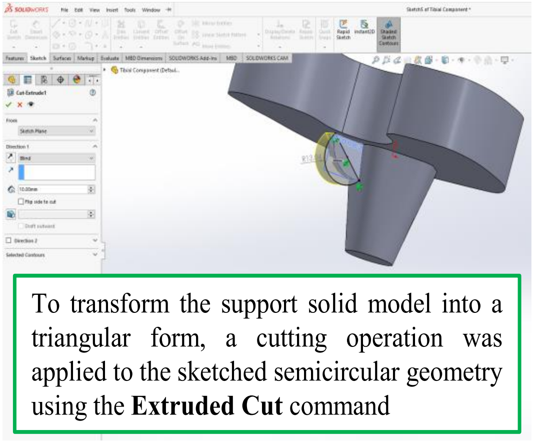


Figure 4-87. Application of the Extruded Cut feature to remove the semicircular geometry from the support region

In the next step, fillet radii of 1 mm and 2 mm were assigned to the sharp edges of the support solid feature using the Fillet tool (Figure 4-88). The support solid was then replicated four times around the lofted solid model by applying a circular pattern with 72° angular spacing in two different directions (Figure 4-89).

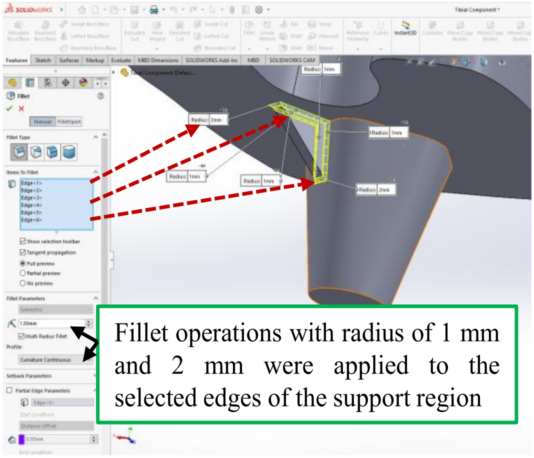


Figure 4-88. Rounding of sharp edges of the support solid feature by applying fillet Radius

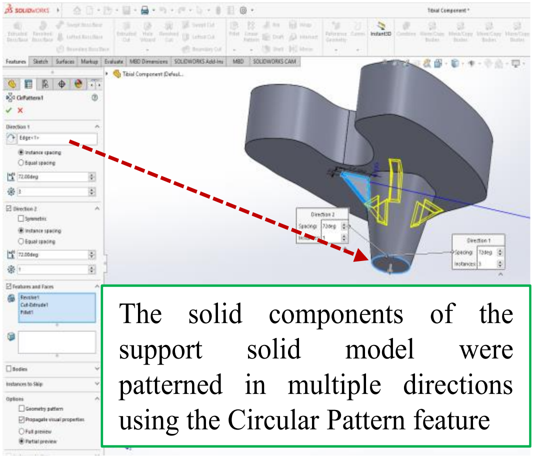


Figure 4-89. Creation of four support solid features using the Circular Pattern operation

In the next step, to promote improved osseointegration between the tibial component and the surrounding bone, a new sketch was created on Plane 1, in which a semicircular geometry was defined to increase the surface contact area. Subsequently, the Sketch 6 profile was selected and material removal was carried out using the Revolved Cut feature in the Features tab (Figure 4-90).

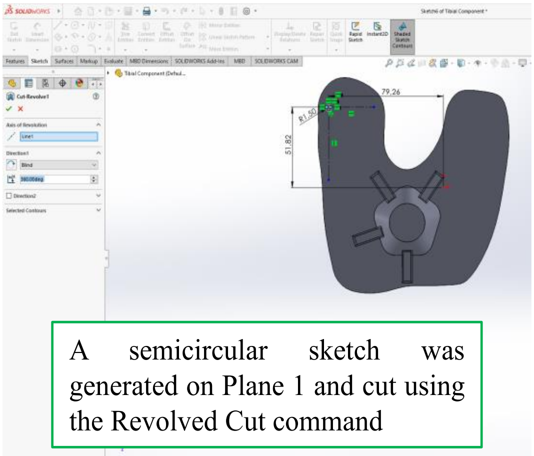


Figure 4-90. Application of the Revolved Cut operation to remove semicircular profiles from the left side of the Tibial Component

After the cutting process, the resulting hemispherical indentations were duplicated by means of a linear pattern along the defined axes, using configurations of 4 mm with 9 instances and 5 mm with 7 instances (Figure 4-91). Subsequently, analogous operations were performed on the right region of the Tibial Component. The sketch environment on Plane 1 was reopened, a semicircular geometry was defined, and the Revolved Cut operation was applied (Figure 4-92).

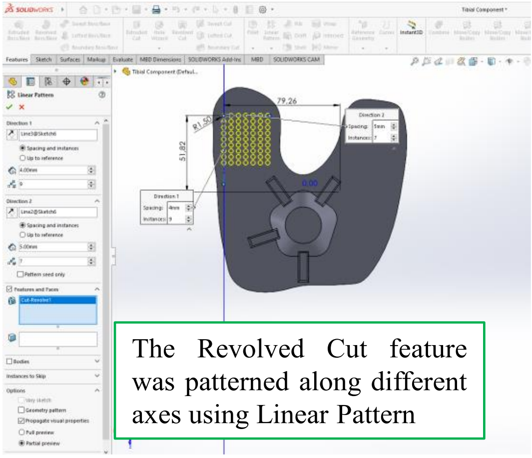


Figure 4-91. Linear duplication of hemispherical cavities along the specified X and Y axes on the left region of the Tibial Component

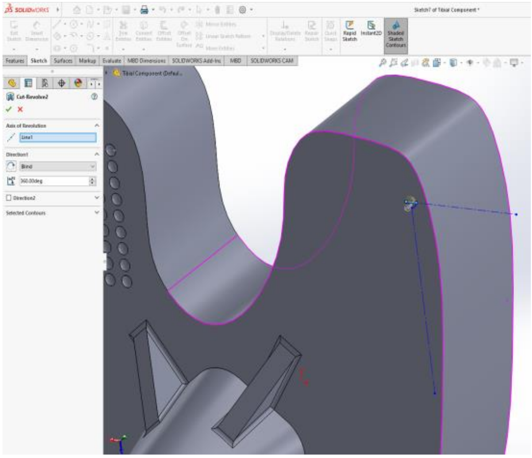


Figure 4-92. Application of the Revolved Cut operation to remove semicircular profiles from the right region of the Tibial Component at a different Plane 1 location

Subsequently, the hemispherical cavities were duplicated using the Linear Pattern feature along specified axes, with spacings of 4 mm × 9 instances and 5 mm × 7 instances (Figure 4-93). The completed solid model of the Tibial Component is shown in Figure 4-94.

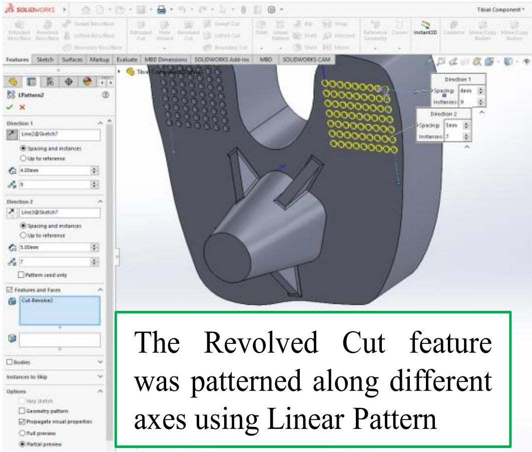


Figure 4-93. Linear duplication of hemispherical cavities along the specified X and Y axes on the right region of the Tibial Component

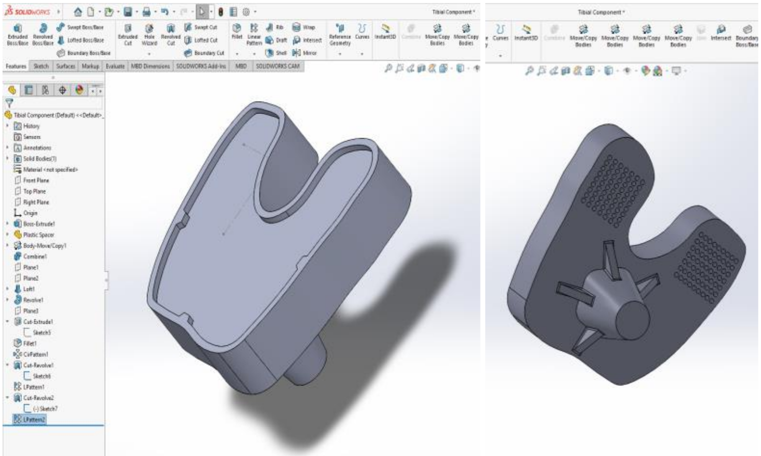


Figure 4-94. Completed solid model of the Tibial Component

The assembly procedure for the complete knee prosthesis was carried out as follows: first, the Plastic Spacer part was opened, then the Femoral Component was inserted and aligned by applying a coincident mate between their surfaces (Figure 4-95).

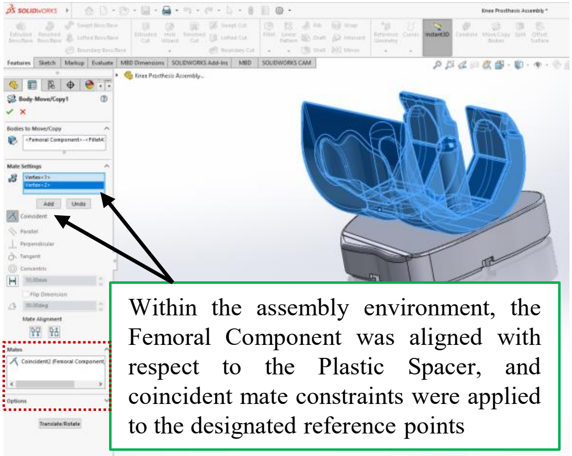


Figure 4-95. Establishing the assembly relationship between the Femoral Component and the Plastic Spacer

Next, the Tibial Component was inserted, and coincident mates were applied to its contact surfaces with the Plastic Spacer (Figure 4-96). The completed assembly of the knee prosthesis is shown in Figure 4-97.

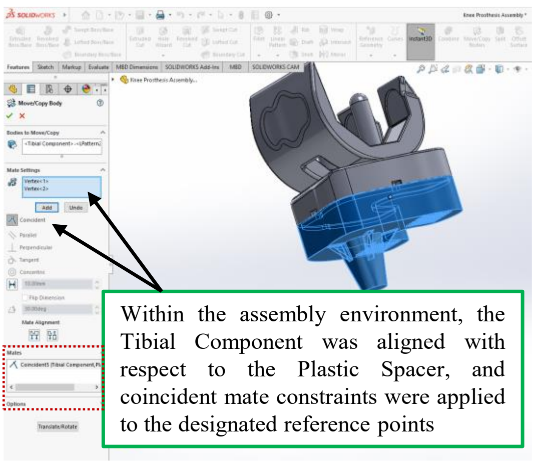


Figure 4-96. Establishing the assembly relationship between the Tibial Component and the Plastic Spacer

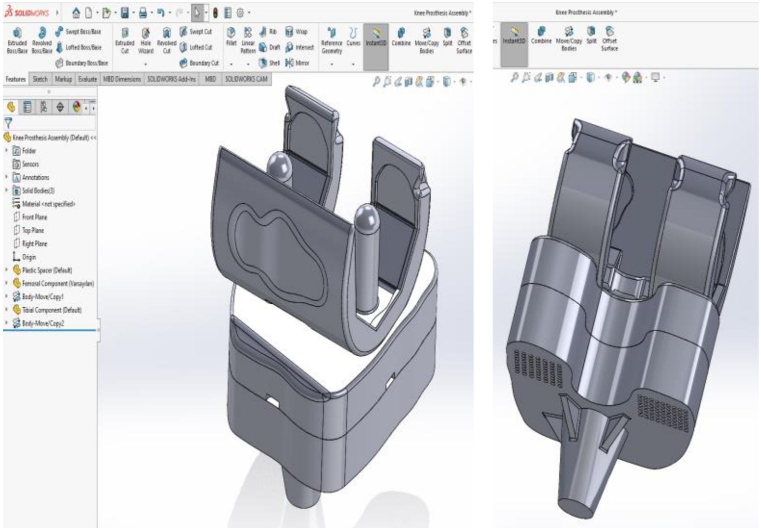


Figure 4-97. Overall appearance of the fully assembled knee prosthesis

5. DETAILED OF THE FEMORAL HEAD HIP PROSTHESIS AND DESIGN MODEL

Before the application of total hip prostheses to the femoral head, symptoms such as pain, loss of function, and decreased quality of life, commonly associated with hip osteoarthritis, usually manifest in the hip region. In addition to these, femoral head collapse (aseptic necrosis), inflammatory arthritis, and femoral neck fractures are among the most common and significant reasons for performing hip prosthesis implantation on the femoral head [47, 48]. Hip prostheses are generally categorized into two types: cemented and cementless designs [49]. In cemented prosthesis systems, fixation is achieved by integrating bone cement between the bone and the implant [49], whereas in cementless types, stabilization is accomplished through a press-fit mechanism that ensures tight contact with the bone [50, 51]. Anatomically, the hip joint is a ball-and-socket (spheroidal) type of joint, in which the femoral head is largely positioned within the acetabulum of the pelvis [52]. Accordingly, total hip prostheses are designed to replicate this anatomical equilibrium. Typically, a total hip prosthesis consists of three main components—femoral stem, femoral head, and acetabular cup or socket. Regarding material selection, titanium alloys are preferred for the femoral stem, while ceramic or metallic materials are used for the femoral head. The acetabular component, on the other hand, is manufactured either from a polymer-based material (for cemented prostheses) or from a metallic outer shell combined with a liner (for cementless prostheses) [47; 53]. From a biomechanical perspective, the forces acting on the hip joint primarily consist of the total compressive loads. These include the force generated by body weight, the tensile force within the abductor muscles, and the impact forces transmitted upward from the foot during daily activities, all of which together contribute to the overall compressive load on the joint [53].

In modern clinical practice, where femoral head pathologies are frequently encountered, both new-generation total hip prostheses and advanced manufacturing techniques have become increasingly prominent. Therefore, whether designed as patient-specific or in standard dimensions, the pre-manufacturing planning and modeling of total hip prosthesis components represent a critical stage in ensuring successful clinical and mechanical outcomes. For this purpose, in the present study, exemplary and representative total hip prosthesis components were designed, and each stage of the design process was described in detail.

5.1. Design Model of the Femoral Head Hip Prosthesis

In hip prosthesis design, the components are sequentially composed of the Acetabular Cup, which is fixed to the hip bone with implant screws. First, the design of this component will be described in detail. Subsequently, the Liner part and the Femoral Head Bearing components, which are assembled onto this part, will be designed. Following this, the Femoral Head and Femoral Stem implant components, which are mounted onto the femur bone, will be designed with detailed modeling and dimensioning. Within this scope, our first detailed design model is the Acetabular Cup component. For the planned Acetabular Cup design, a new part file was first opened in the solid modeling software. At the beginning of the study, the part file was named according to the planned project, and it was labeled "Acetabular Cup". To create the planned sketch in this part file, the Front Plane was first selected from the feature tree on the left side of the solid modeling interface, and a new sketch environment was opened from this plane. These steps are illustrated in detail in Figure 5-1. A 2D sketch of the planned Acetabular Cup model was then started in the opened sketch environment (Figure 5-2). Since the solid model will be created by revolving the sketch, only half of the geometry was drawn in the 2D sketch. In the next step, to convert the defined 2D sketch lines into a solid model, the sketch file was selected, and the Revolve Boss/Base command in the Features tab was used to generate the solid model.

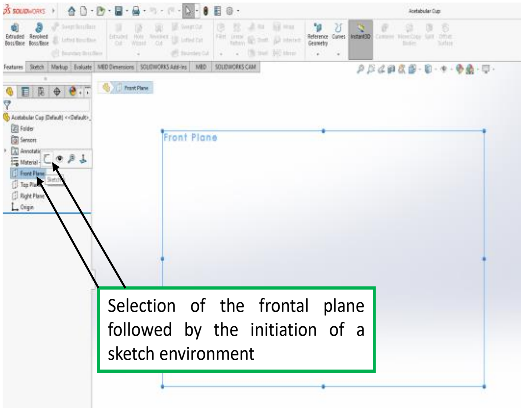


Figure 5-1. Selection of the front plane and creation of the drawing window

At this stage, the revolve axis is required. The axis line defined during the sketch was selected as the revolve axis, and the first solid model of the Acetabular Cup was created (Figure 5-3).

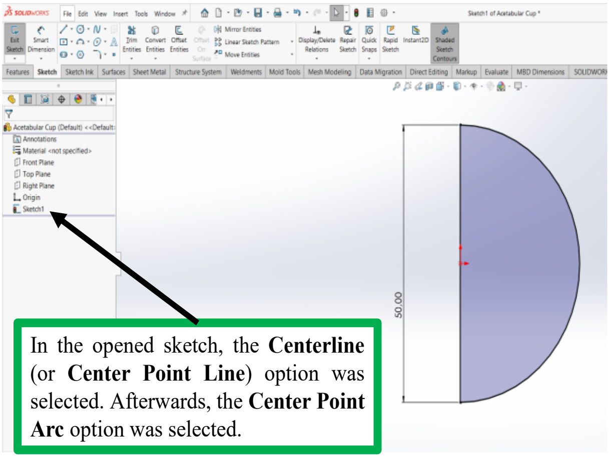


Figure 5-2. The initial stage of the 2D design of the Acetabular Cup model

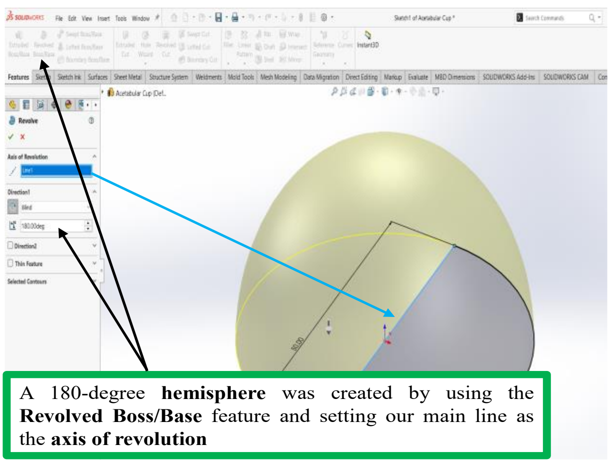


Figure 5-3. Initial development phase of the Acetabular Cup's 3D solid model

A sketch environment was opened on the base of the created hemispherical solid, and a 2D semicircular sketch was drawn (Figure 5-4). This 2D sketch was then used to perform a circular cut, creating the recess where the Liner part will be assembled (Figure 5-5). This operation was carried out using the Cut Revolve command in the Features tab.

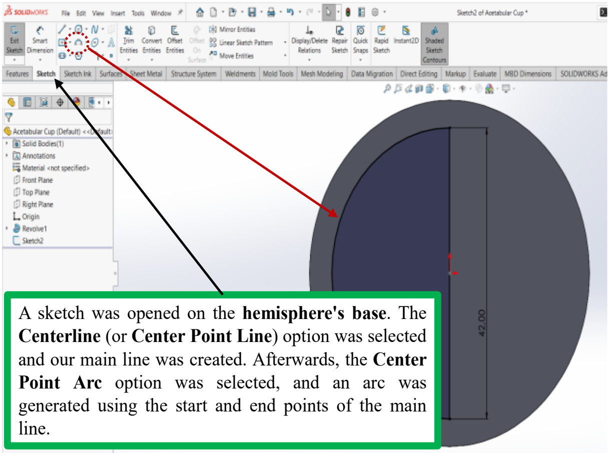


Figure 5-4. Creating the 2D semicircular sketch on the base of the Acetabular Cup solid model

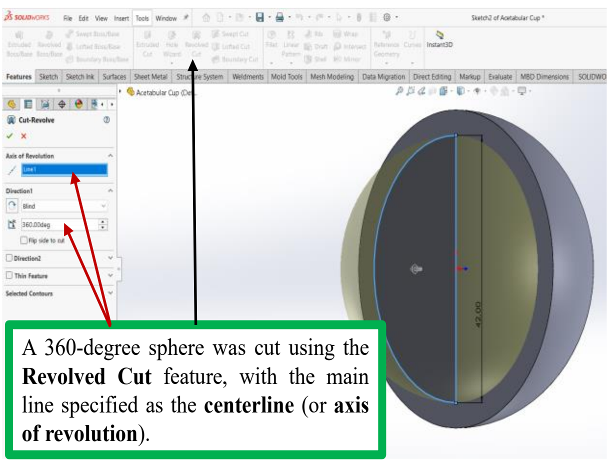


Figure 5-5. Creating the hemispherical cavity on the base of the Acetabular Cup solid model by applying a revolve cut

The state of the solid model after the second stage of operations is shown in Figure 5-6. Subsequently, to create the fixation screw holes for the hip bone, a new reference plane was created on the Acetabular Cup solid model by selecting Reference Geometry from the features tab, as illustrated in Figure 5-7.

In Figure 5-8, the centers of the holes for positioning the screws to fix the Acetabular Cup to the hip are to be created. For this purpose, the Front Plane was first selected, and a new sketch environment was opened on this plane to draw the initial hole centers.

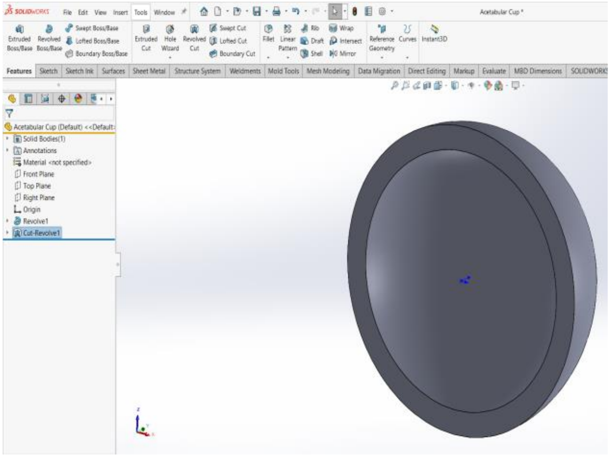


Figure 5-6. The solid model of the Acetabular Cup after performing the revolve cut operation

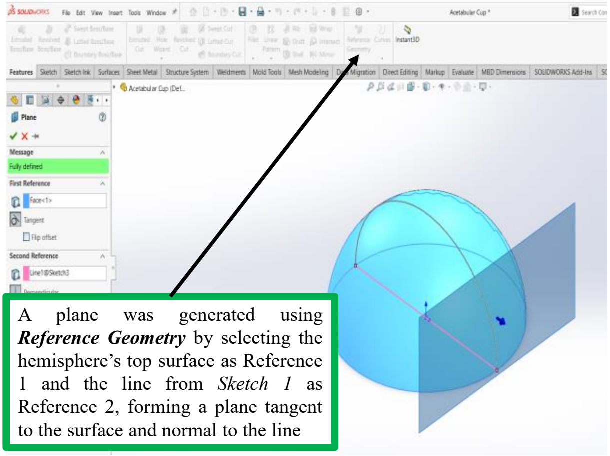


Figure 5-7. Establishing a new plane on the Acetabular Cup solid model via the Reference Geometry feature

In the opened sketch environment, a centerline was drawn from the origin as a reference. This centerline was positioned at an angle of 23.55° relative to the bottom base of the hemispherical part. Next, for the screw holes, a new plane was assigned on the surface of the hemispherical part (Figure 5-9). The new plane was created by selecting Reference

Geometry from the Features tab. Then, the centerline drawn in Figure 5-8 and the outer surfaces of the hemispherical part were selected as references, and a plane tangent to the surface of the hemispherical part was created (Figure 5-9).

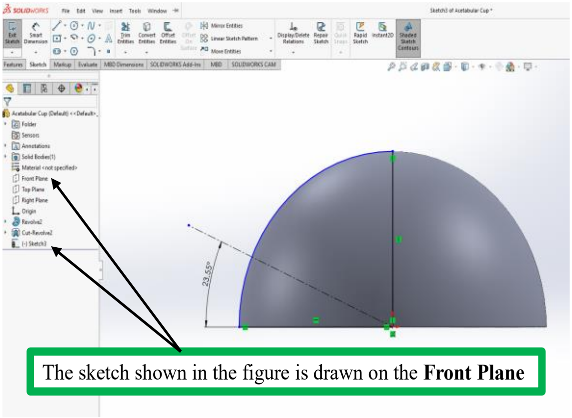


Figure 5-8. Creating the screw hole centers on the Acetabular Cup solid model

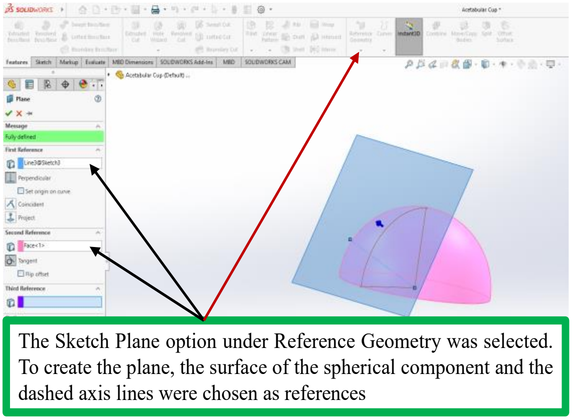


Figure 5-9. Creating a plane tangent to the spherical surface of the Acetabular Cup solid model.

Using the centering points created in Figure 5-8 for the hip fixation screw holes, a new sketch environment was opened on the plane created in Figure 5-9.

Subsequently, a circular geometry was drawn in the opened sketch environment to define the hole cut (Figure 5-10).

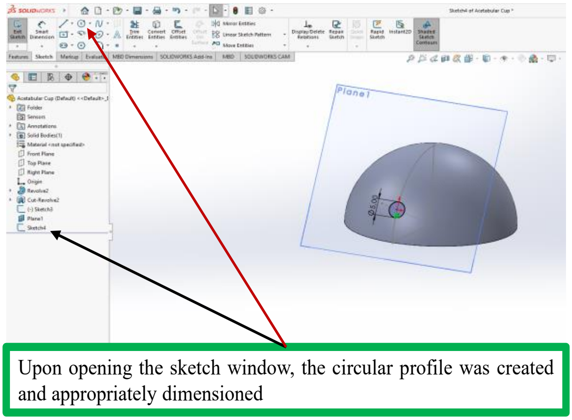


Figure 5-10. Creating 2D sketches of the screw holes on the surface of the Acetabular Cup solid model

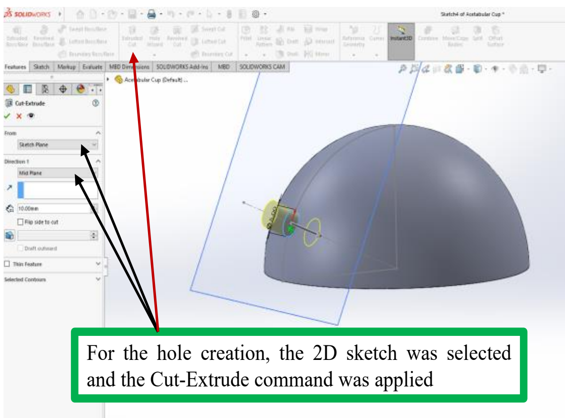


Figure 5-11. Creating 3D holes on the surface of the Acetabular Cup solid model

For the hole solid model, the 2D sketch was first selected. Then, the Cut Extrude command was applied to create the hole (Figure 5-11).

After creating the initial screw hole for hip fixation, the hole was replicated to form additional screw positions. For this purpose, the Circular Pattern operation under the Pattern section in the Features tab was used to create a total of three holes, each positioned 25° apart. During the circular pattern operation, the direction vector was set by selecting the line perpendicular to the bottom base in the sketch shown in Figure 5-8 (Figure 5-12).

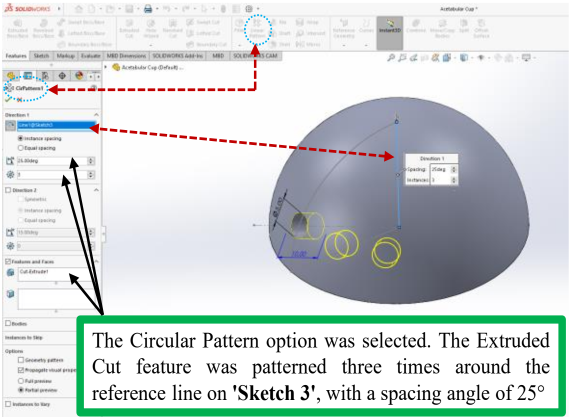


Figure 5-12. Circular patterning of the hole on the surface of the Acetabular Cup solid model.

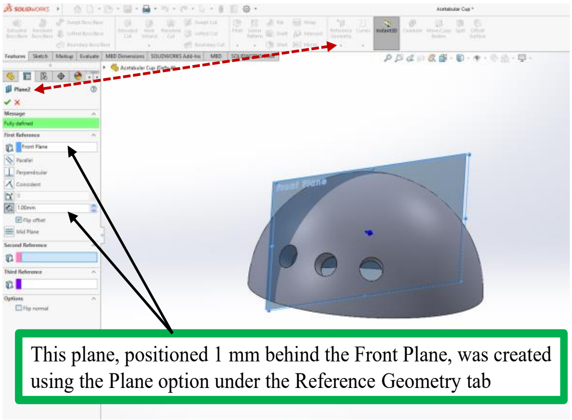


Figure 5-13. Creating a new plane on the Acetabular Cup solid model

The next step involved creating additional positions for the hip fixation screw holes. For this purpose, a new plane was defined using the Front Plane as a reference through the Reference Geometry feature (Figure 5-13). To create additional screw holes for fixing the Acetabular Cup to the hip bone from a different position, a new centerline was drawn at a 45° angle in the sketch environment opened on the plane created in Figure 5-13 (Figure 5-14).

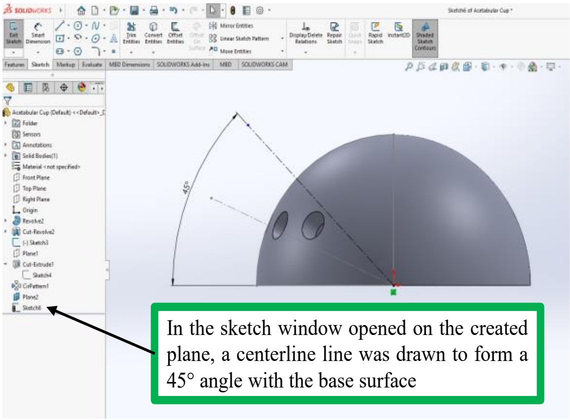


Figure 5-14. Creating the screw hole center at a new position on the Acetabular Cup solid model

Then, to create new screw holes on the Acetabular Cup spherical part, a new plane was created on the surface of the spherical part using the new center as a reference (Figure 5-15).

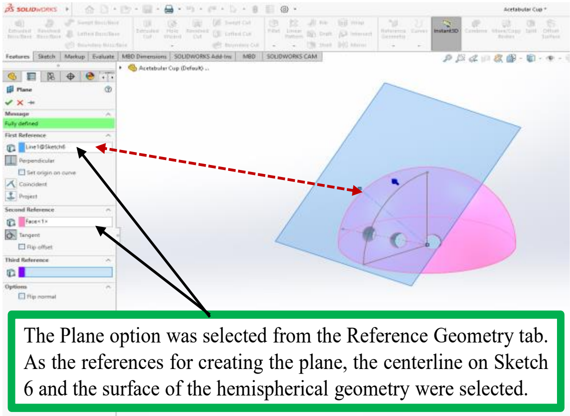


Figure 5-15. Creating a new tangent plane at a different position on the spherical surface of the Acetabular Cup solid model

A sketch was opened on the newly created plane in Figure 5-15, and circular geometry was drawn using the defined centerline as a reference for the new screw hole positions (Figure 5-16).

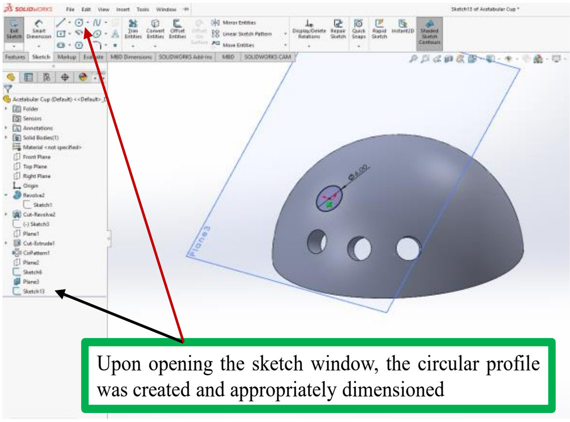


Figure 5-16. Creating a 2D sketch of the screw holes on the spherical surface of the Acetabular Cup solid model.

For the hole solid model, the 2D sketch was first selected. Then, the Cut Extrude command was applied to create the hole (Figure 5-17).

Subsequently, the screw holes were replicated in the existing region. For this purpose, the Circular Pattern operation under the Linear Pattern section in the Features tab was used, resulting in a total of three holes, each spaced at 35°. During the circular pattern operation, the direction vector was set by selecting the line perpendicular to the bottom base in the sketch shown in Figure 5-8 (Figure 5-18).

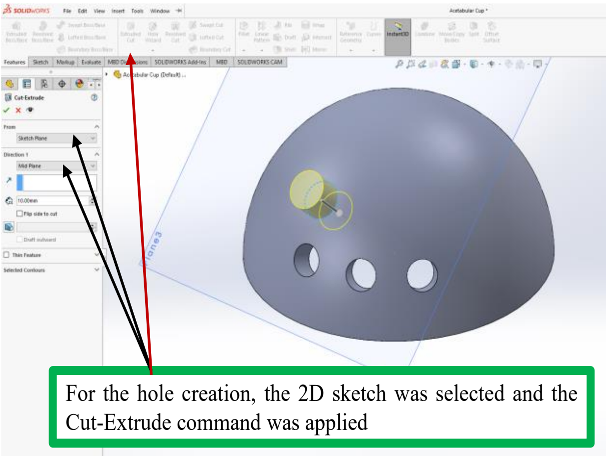


Figure 5-17. Creating 3D holes on the newly positioned surface of the Acetabular Cup solid model

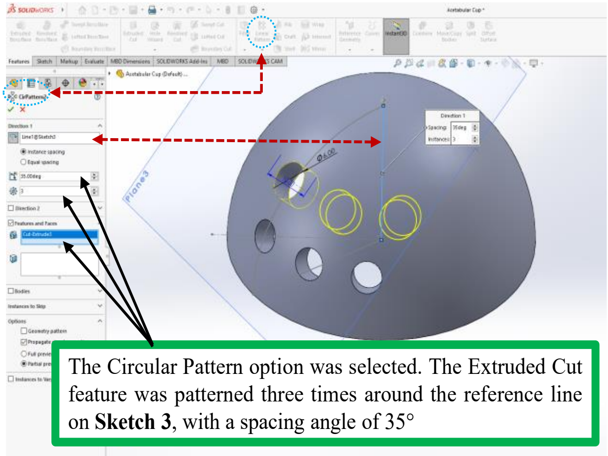


Figure 5-18. Circular patterning of the hole on the newly positioned surface of the Acetabular Cup solid model

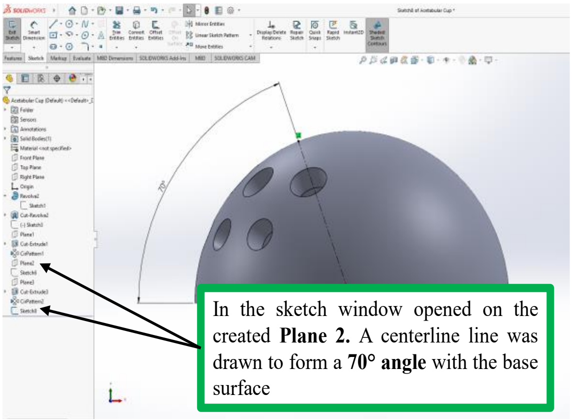


Figure 5-19. Creating the screw hole center at a new position on the Acetabular Cup solid model

To create additional screw holes for fixing the Acetabular Cup to the hip bone from a different position, a new centerline was drawn at a 70° angle in the sketch opened on the plane created in Figure 5-13 (Figure 5-19).

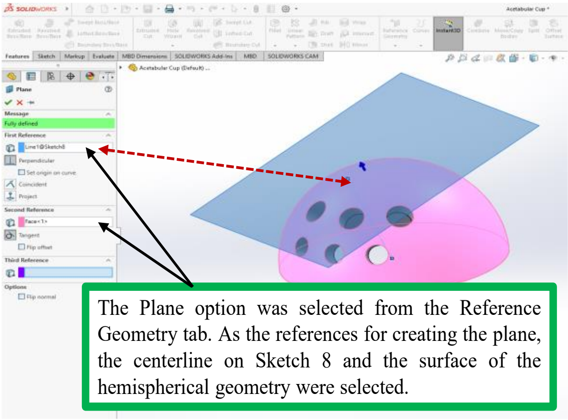


Figure 5-20. Creating a new tangent plane at a different position on the spherical surface of the Acetabular Cup solid model

Then, to create new screw holes on the Acetabular Cup spherical part, a new plane was generated on the spherical surface using the newly defined center as a reference (Figure 5-20).

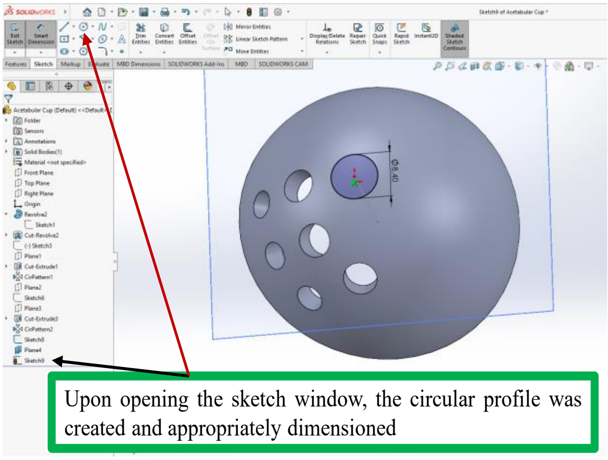


Figure 5-21. Creating a 2D sketch of the screw holes on the spherical surface of the Acetabular Cup solid model

A sketching window was opened on the newly created plane, and a circular geometry was drawn using the defined centerline as a reference for the new screw hole positions (Figure 5-21). Next, the 2D sketch for the hole solid model was selected, and the Cut Extrude command was applied to create the hole (Figure 5-22). Subsequently, the newly created hole was replicated in the existing region. For this purpose, the Curve Driven Pattern operation under the Linear Pattern section in the Features tab was used, creating two holes spaced 40 mm apart. The direction vector was defined by selecting the circular edge of the bottom base (Figure 5-23).

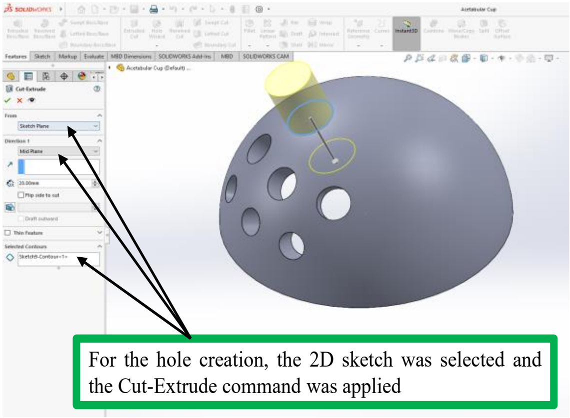


Figure 5-22. Creating a 3D hole on the newly positioned surface of the Acetabular Cup solid model

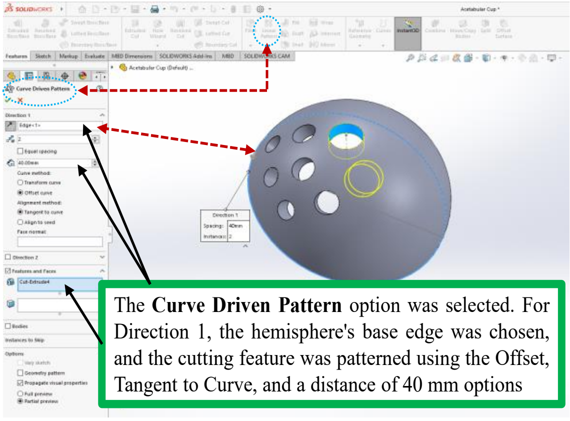


Figure 5-23. Curve-driven pattern replication of the hole on the newly positioned surface of the Acetabular Cup solid model

For the Acetabular Cup model with created screw hole sections, the inner edges of the holes were selected and fillet operations were applied to smooth sharp edges. Accordingly, a 0.5 mm radius was applied to the Ø 8.4 mm holes (Figure 5-24), a 0.3 mm radius to the Ø 6 mm holes (Figure 5-25), and a 0.2 mm radius to the Ø 5 mm holes (Figure 5-26).

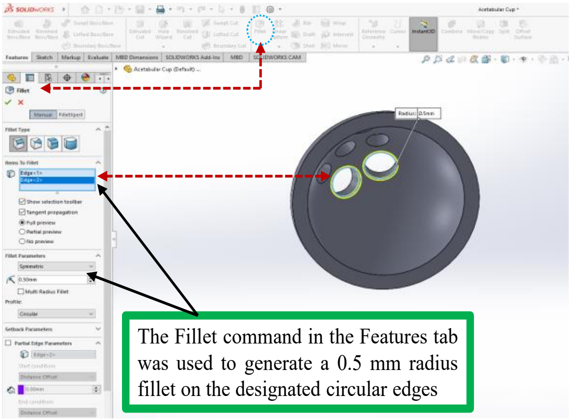


Figure 5-24. Application of a 0.5 mm fillet to the Ø 8.4 mm holes in the Acetabular Cup model

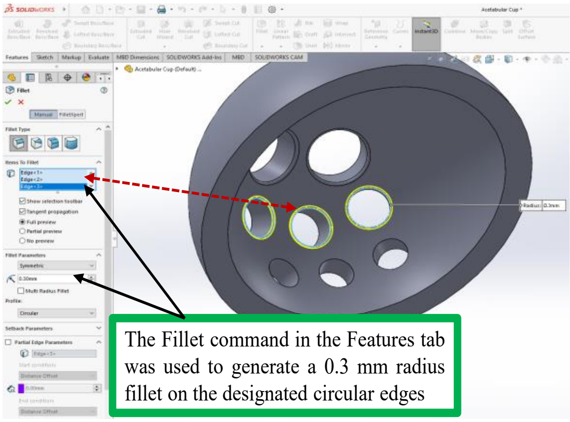


Figure 5-25. Application of a 0.3 mm fillet to the Ø 6 mm holes in the Acetabular Cup model

In the next step, the base of the Acetabular Cup model was selected to open the sketching window. Subsequently, two circular geometries of different diameters were drawn within this sketching window (Figure 5.27).

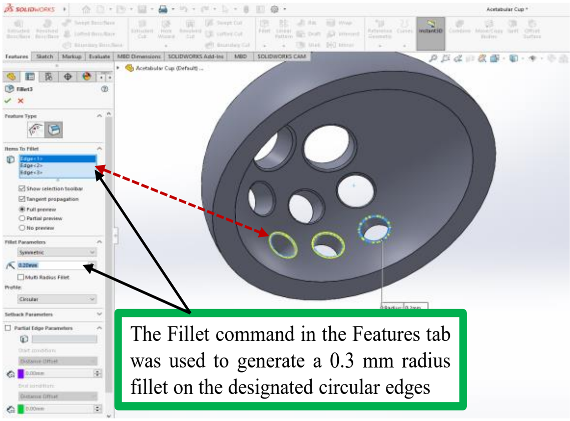


Figure 5-26. Application of a 0.2 mm fillet to the Ø 5 mm holes on the Acetabular Cup model

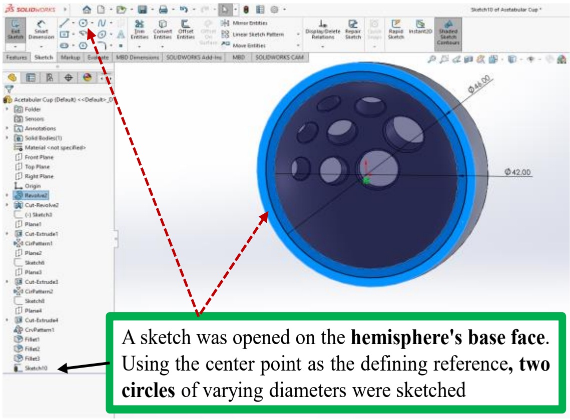


Figure 5-27. Application of a 0.2 mm fillet to the Ø 5 mm holes on the Acetabular Cup model

In the next step, the sketch drawn in Figure 5-27 was converted into a solid model by applying a 2 mm thickness using the Extruded Boss/Base command (Figure 5-28). In the solid model created with the 2 mm thickness shown in Figure 5-28, fillet operations were applied to smooth the edges of the circular geometry: a 0.25 mm fillet on the outer edges (Figure 5-29) and a 0.45 mm fillet on the inner edges (Figure 5-30).

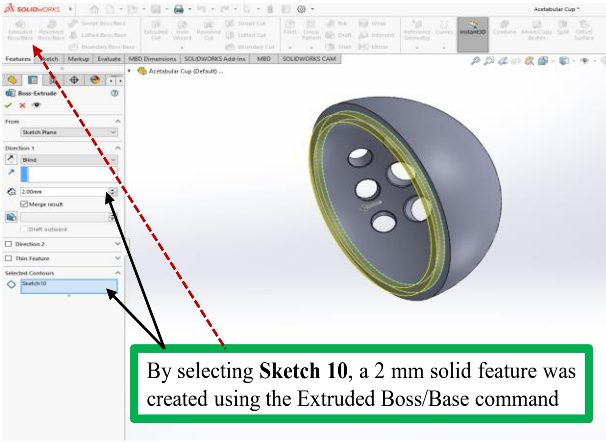


Figure 5-28. Application of a 2 mm thickness to the sketch on the base of the Acetabular Cup model

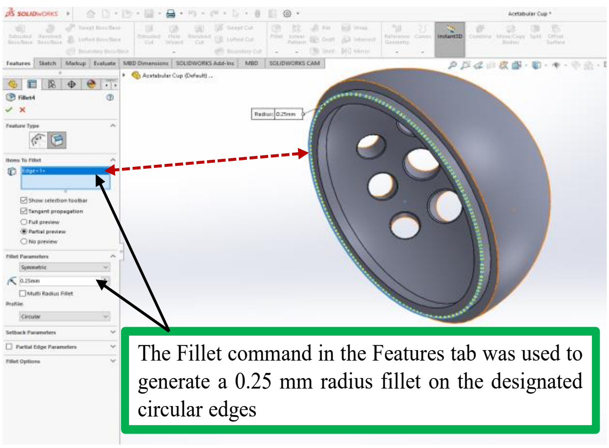


Figure 5-29. Application of a 0.25 mm fillet to the outer edge of the solid model at the base of the Acetabular Cup model

Subsequently, a new plane was assigned to the bottom surface of the newly created solid model on the base of the Acetabular Cup. This was performed by selecting Reference Geometry from the Features tab and then choosing Plane (Figure 5-31).

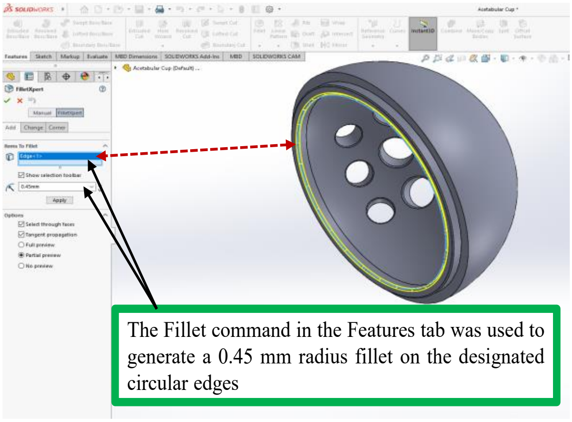


Figure 5-30. Application of a 0.45 mm fillet to the inner edge of the solid model at the base of the Acetabular Cup model

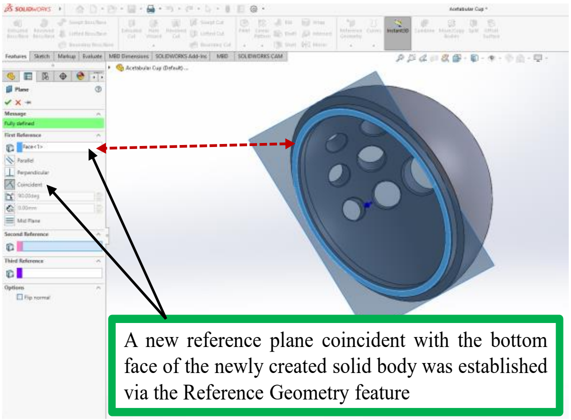


Figure 5-31. Creation of a plane on the bottom surface of the newly created solid model of the Acetabular Cup

In the next step, the sketching window was opened on the newly created plane to create the connection regions for the other model components, and a rectangular geometry was drawn (Figure 5-32). The 2D rectangular geometry was then used with the Cut-Extrude command on the existing solid model to cut down to the first base surface of the hemisphere (Figure 5-33). In this way, the connection sections of the Acetabular Cup model with the other component parts were established.

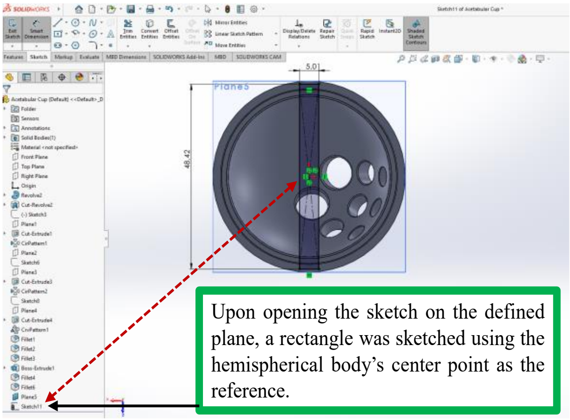


Figure 5-32. Opening the sketching window on the new plane at the base surface of the Acetabular Cup model and drawing the rectangular geometry

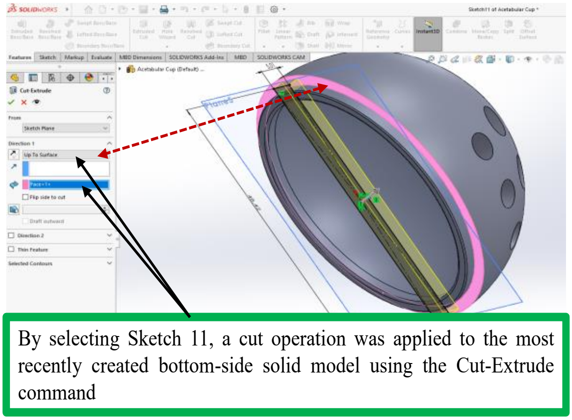


Figure 5-33. Selection of the rectangular geometry on the base surface of the Acetabular Cup model and application of the cut operation to the underlying solid model

To replicate these connection sections around a circular path, the Circular Pattern feature was used. In the first step, four connection regions were created at 60° intervals around the circle (Figure 5-34). With this initial replication, four connection points of the Acetabular Cup model with other components were established.

In the next step, the Circular Pattern command was applied again to increase the number of connection regions to six (Figure 5-35). The difference compared to the previous modeling was that the direction of the pattern replication was reversed.

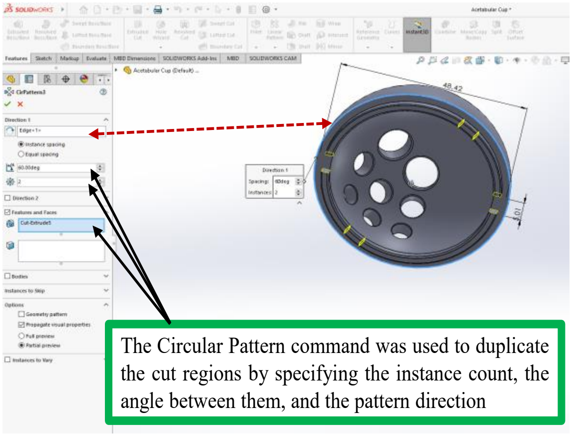


Figure 5-34. First stage of replicating the connection regions on the base surface of the Acetabular Cup model using the Circular Pattern command

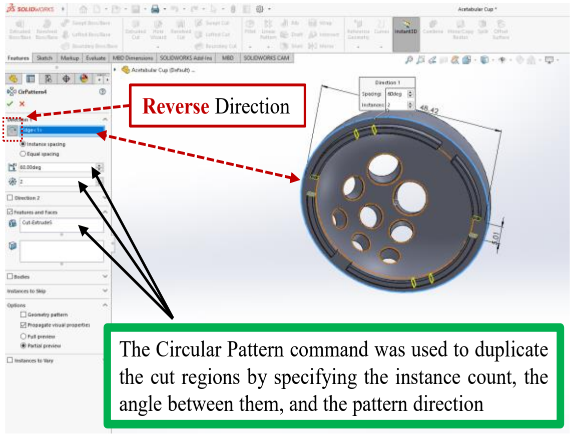


Figure 5-35. Second stage of replicating the connection regions on the base surface of the Acetabular Cup model using the Circular Pattern command

In the next step, chamfering with a 0.8 mm dimension was applied to the connection regions. For this purpose, the Chamfer option within the Fillet command under the Features tab was selected. The chamfer was then applied with an angle of 60° and a distance of 0.8 mm (Figure 5-36).

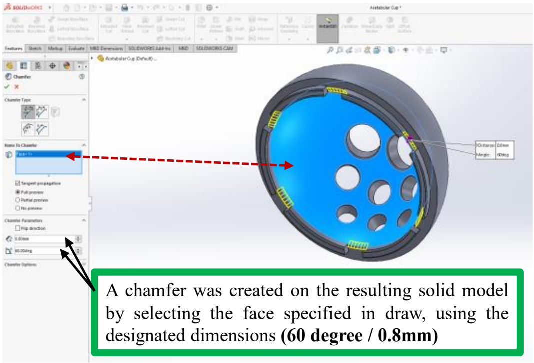


Figure 5-36. Application of chamfering between the connection regions and the inner surface on the base of the Acetabular Cup model

To increase osseointegration in the seating area of the Acetabular Cup component on the hip bone, the surface area was targeted to be enlarged. For this purpose, lateral volumes with circular geometry were added to the portion of the spherical surface without screw holes to expand the surface area. First, the Front Plane was used as a reference to open the sketching window. Then, using the origin of the spherical part as a reference, two centerline sketches were created on the base surface at a 15° angle (Figure 5-37). To coincide with the centerlines, the Reference Geometry → Plane command was used to create a new plane. The selected surface was offset by 6.47 mm to generate the new plane (Figure 5-38).

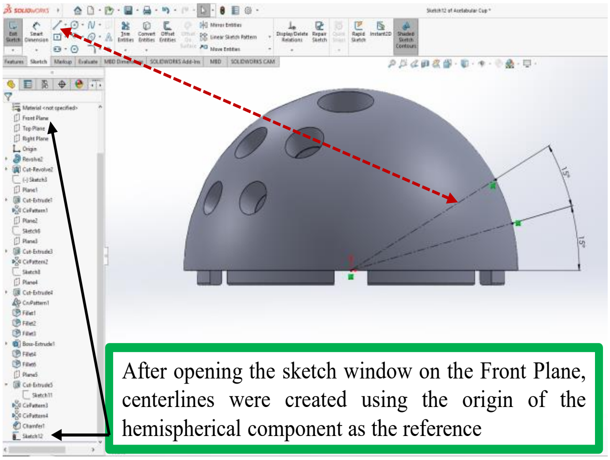


Figure 5-37. Drawing two centerlines at a 15° angle on the sketching window opened on the Front Plane of the Acetabular Cup model

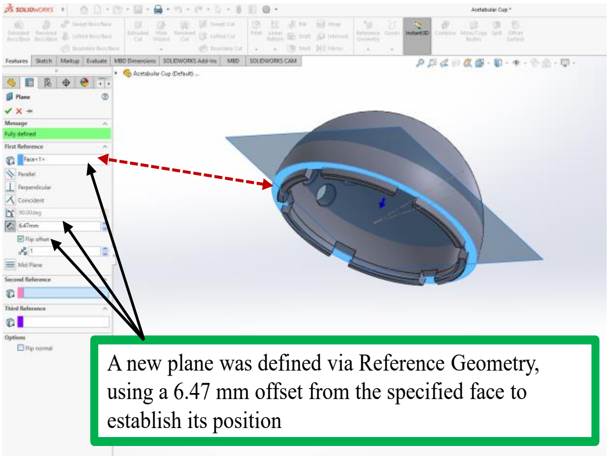


Figure 5-38. Creation of a plane in the opposite direction on the selected surface of the Acetabular Cup model

A new sketching window was opened on the plane created in Figure 5-38 (Figure 5-39). The next step aimed to create the path for the solid sweep model. An arc was drawn at the specified angle and radius as indicated in the illustration (Figure 5-39).

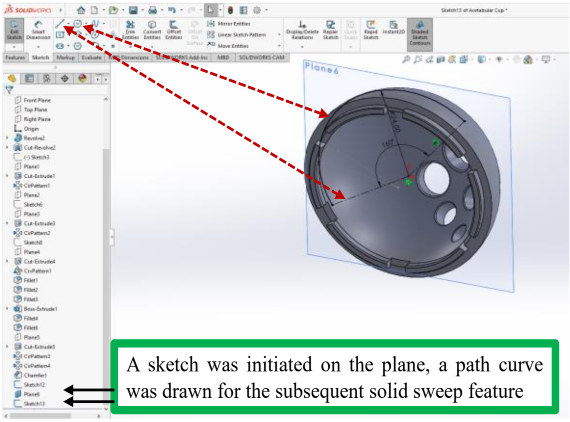


Figure 5-39. Sketching the path on the spherical surface of the Acetabular Cup model in preparation for the solid sweep model

The positions between the drawn centerlines were set by entering the specified angular value (Figure 5-39). Then, using the endpoints of the 15° centerlines from Figure 5-37 and the arc drawn in Figure 5-39 as references, a new plane was created through the **Reference Geometry** option under the **Features** tab (Figure 5-40).

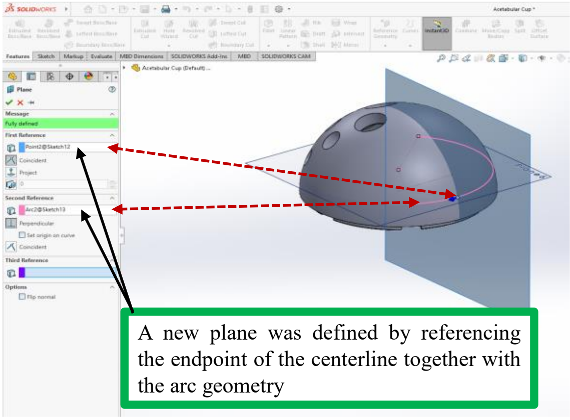


Figure 5-40. Creation of a new plane on the spherical surface of the Acetabular Cup model

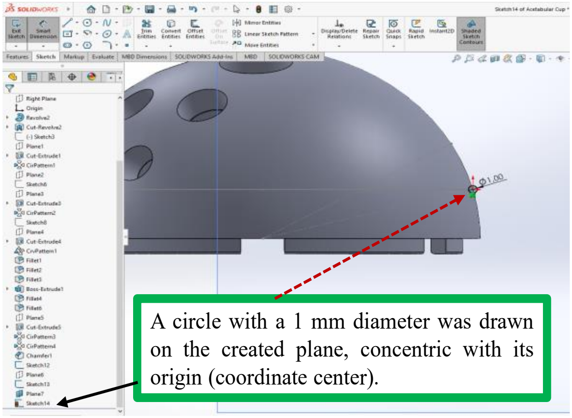


Figure 5-41. Section sketch on the spherical surface of the Acetabular Cup model for creating the solid sweep model

On the created plane, the section sketch for the solid sweep model, aimed at increasing the targeted surface area for osseointegration, was drawn (Figure 5-41). In the subsequent step, the circular geometry drawn in Figure 5-41 and the arc profiles from Figure 5-39 were used with the Swept Boss/Base command under the Features tab to generate the solid sweep model (Figure 5-42).

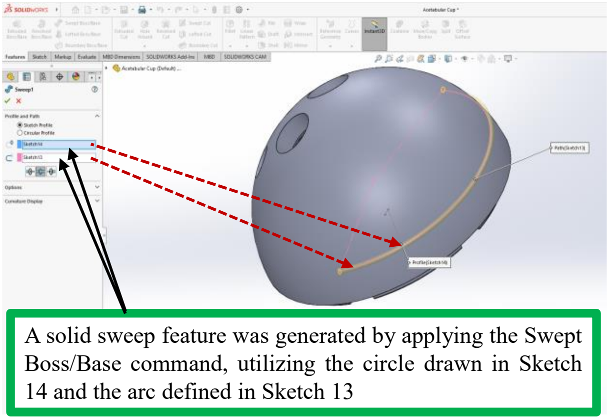


Figure 5-42. Creation of the solid sweep model on the spherical surface of the Acetabular Cup model

In the next step, chamfers were applied to the edges at the ends of the solid model created in Figure 5-42 using the Chamfer option within the Fillet command under the Features tab (Figure 5-43).

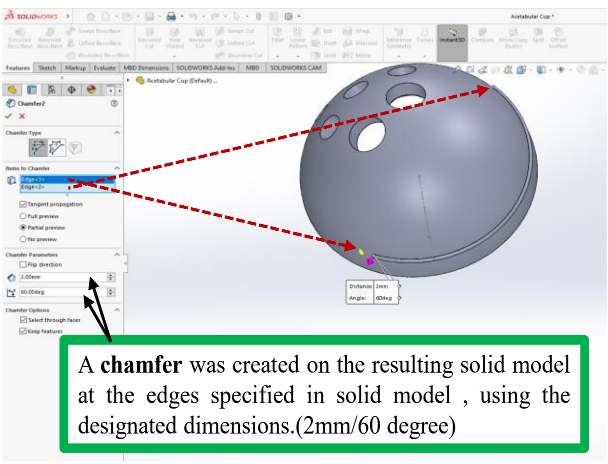


Figure 5-43. Application of chamfering to the solid sweep model on the spherical surface of the Acetabular Cup model

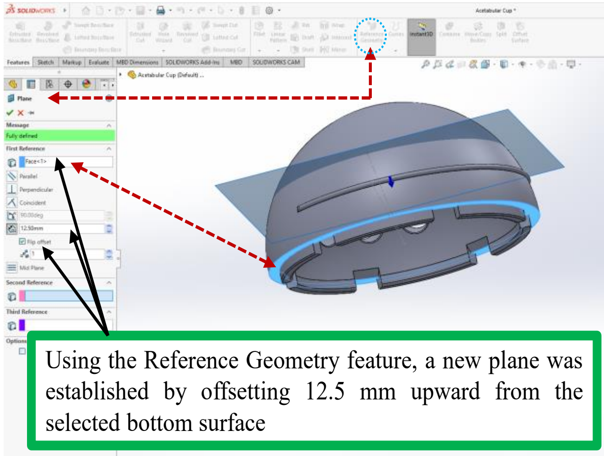


Figure 5-44. Creation of a new plane in the opposite direction on the selected surface of the Acetabular Cup model

A new plane was created from the Reference Geometry feature to coincide with the center of the centerline sketches. The selected surface was offset by 12.5 mm to generate the new plane (Figure 5-44). After opening a new sketching window on this plane, the path for the subsequent solid sweep model was planned. An arc was drawn at the specified angle and radius as indicated in the illustration (Figure 5-45).

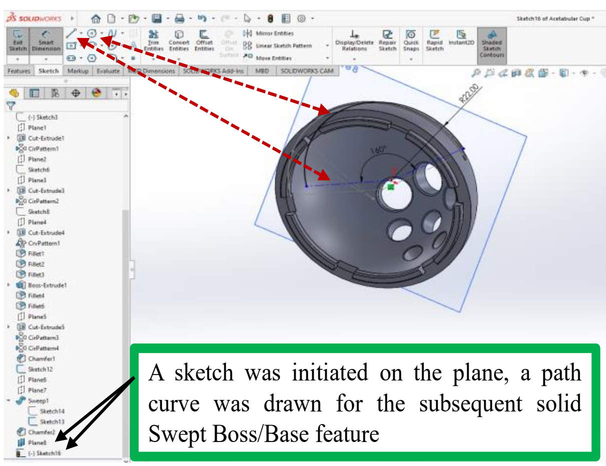


Figure 5-45. Sketching the path for the second-stage solid sweep model on the spherical surface of the Acetabular Cup model

Subsequently, using the endpoints of the 15° centerlines from Figure 5-37 and the arc drawn in Figure 5-45 as references, a new plane was created through the Reference Geometry option under the Features tab (Figure 5-46). On this newly created plane, a section sketch for the solid sweep model was drawn on the base surface of the hemispherical part, positioned at a 30° angle, aimed at increasing the targeted surface area for osseointegration (Figure 5-47).

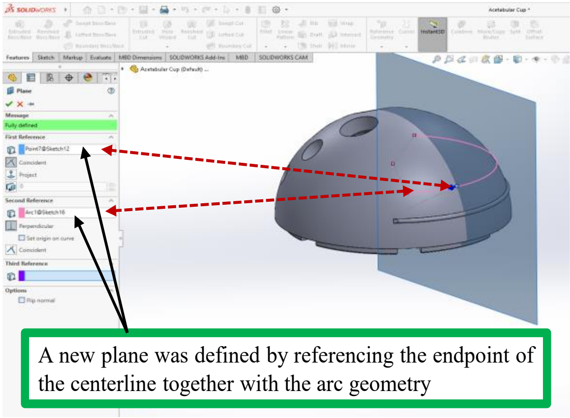


Figure 5-46. Creation of a new plane at the cut point on the spherical surface of the Acetabular Cup model, positioned at a 30° angle relative to the base

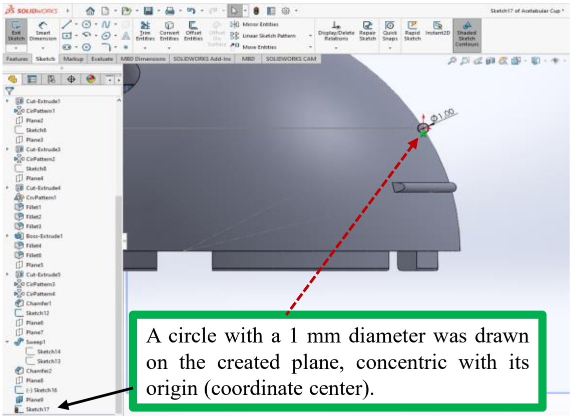


Figure 5-47. Section sketch for the solid sweep model at a 30° position relative to the base on the spherical surface of the Acetabular Cup model

In the next step, using the circular geometry drawn in Figure 5-47 and the arc profiles from Figure 5-46, the Swept Boss/Base command under the Features tab was applied to create the solid sweep model on the hemispherical part, positioned at a 30° angle (Figure 5-48).

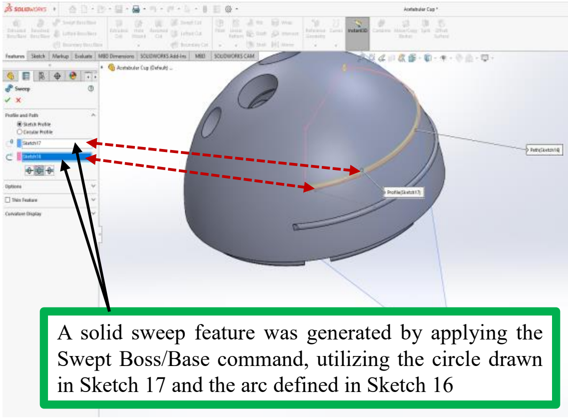


Figure 5-48. Creation of the solid sweep model at a 30° position relative to the base on the spherical surface of the Acetabular Cup model

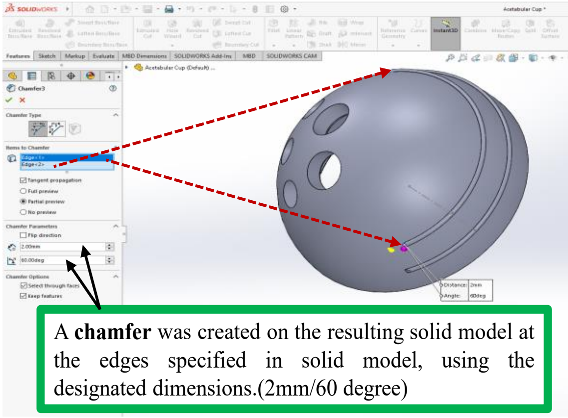


Figure 5-49. Application of chamfering to the solid sweep model at a 30° position relative to the base on the spherical surface of the Acetabular Cup model

In the next step, chamfers (2 mm at a 60° angle) were applied to the edges at the ends of the solid model created in Figure 5-48 using the Chamfer option within the Fillet command under the Features tab (Figure 5-49). The view of the fully completed solid model of the Acetabular Cup component is shown in Figure 5-50.

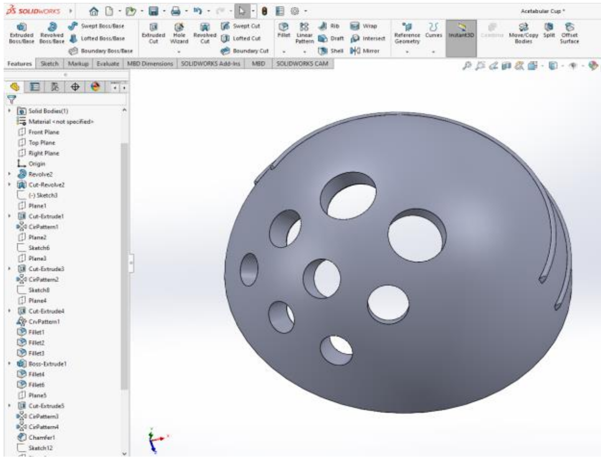


Figure 5-50. Final view of the completed Acetabular Cup solid model

The second component of the hip prosthesis design is the solid modeling of the liner part. For the planned Liner Part (Polyethylene Insert) design, a new part file is first created in the solid modeling software. This file is then named according to the project plan; in this study, it is designated as ‘Liner Part (Polyethylene Insert). To construct the planned geometry, the Front Plane is selected from the Feature Tree located on the left panel of the software, and a new sketch window is initiated. These steps are illustrated in detail in Figure 5-51. Once the sketch environment is opened, the initial 2D profile of the liner part is drawn as the starting point of the design (Figure 5-52). Since the solid model will be generated using a revolve operation, only half of the profile is sketched. In the subsequent stage, the 2D geometry is converted into a solid model. With the sketch selected, the Features tab is accessed and the ‘Revolved Boss/Base’ command is activated. In the revolve setup, the axis selected is the centerline defined during the sketching stage. Using this axis, the first version of the solid model is generated (Figure 5-53).

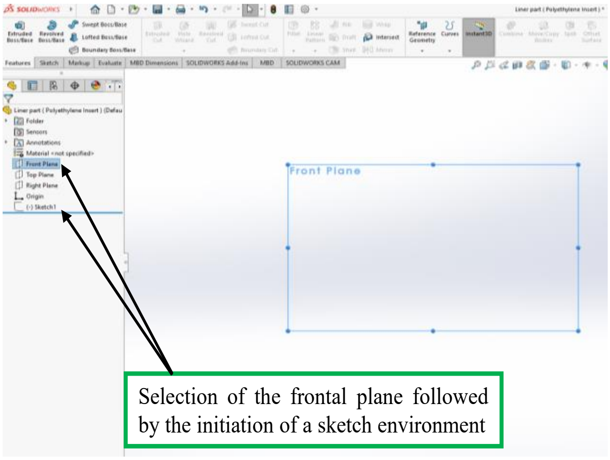


Figure 5-51. Selection of the front plane and creation of the drawing window

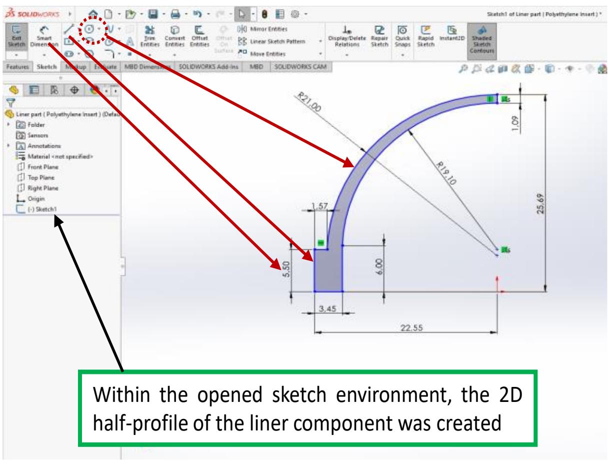


Figure 5-52. The initial stage of the 2D design of the Liner part model

In the first stage of creating the solid model, the connection interfaces of the liner part and the acetabular cup must be identical to ensure proper seating of the hemispherical liner on the acetabular cup. For this purpose, without generating an additional sketch, the Acetabular Cup component was imported into the existing part file.

This was achieved by dragging and dropping the component into the current part document where the solid model had been created. Subsequently, to enable contact alignment between the two components, adjustments were made using the Body Move/Copy command located in the Features tab. In the initial step, the circular edges of the components were used as reference entities to perform center alignment (Figure 5-54).

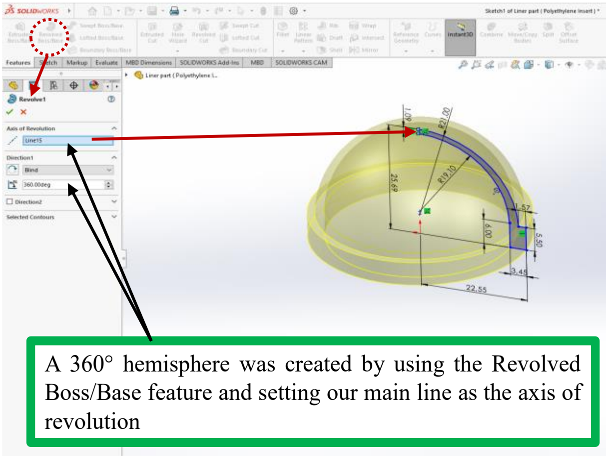


Figure 5-53. Initial development phase of the Liner Part’s 3D solid model

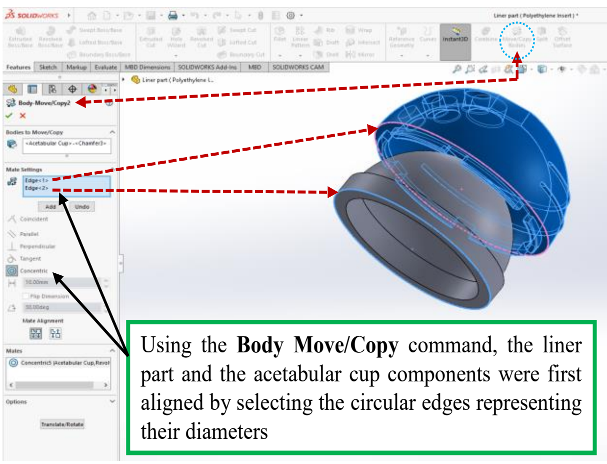


Figure 5-54. Alignment of the centers of the liner part and the acetabular cup components using the Body Move/Copy command

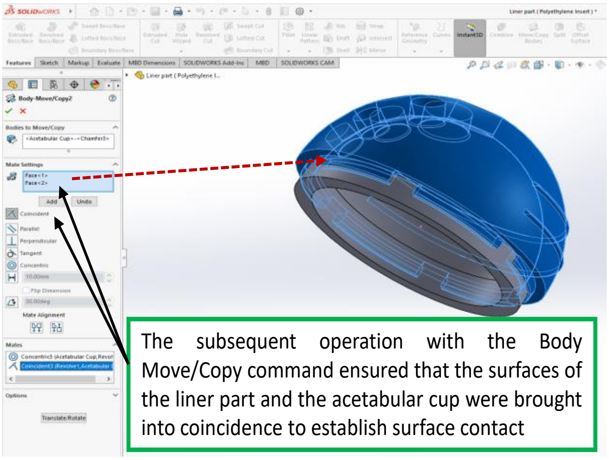


Figure 5-55. Coincident surface alignment of the acetabular cup component with the solid-modeled liner part

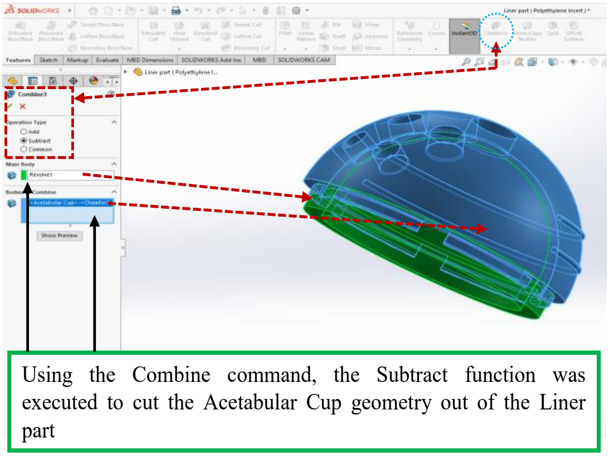


Figure 5-56. Application of the Combine–Subtract command to remove the Acetabular Cup body from the Liner part body

Subsequently, the inner surface of the Acetabular Cup component and the outer surface of the solid-modelled liner part were brought into coincident contact (Figure 5-55).

After achieving full surface alignment, the two components were separated, and the Combine command was used to generate the detailed connection interface of the liner part corresponding to the acetabular cup geometry (Figure 5-56). After activating the Combine command, the Subtract operation was applied to remove the Acetabular Cup component from the Liner part. In the subsequent step, the Fillet command was used to apply a 1 mm radius to the outer edge of the bottom surface of the Liner part (Figure 57), and a 0.4 mm radius to the inner edge located on the same surface (Figure 58).

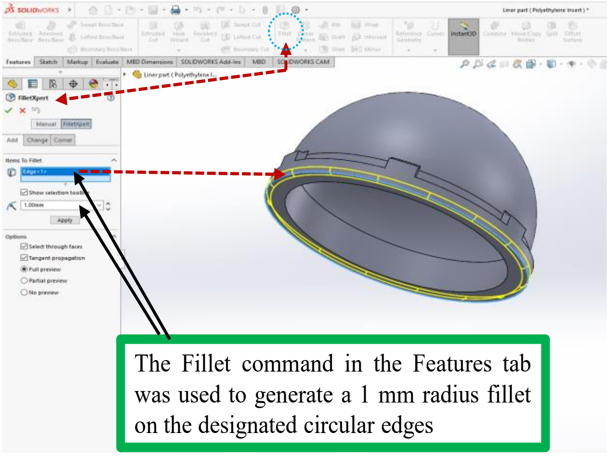


Figure 5-57. Applying a 1 mm-radius fillet to the lower outer edge of the liner component

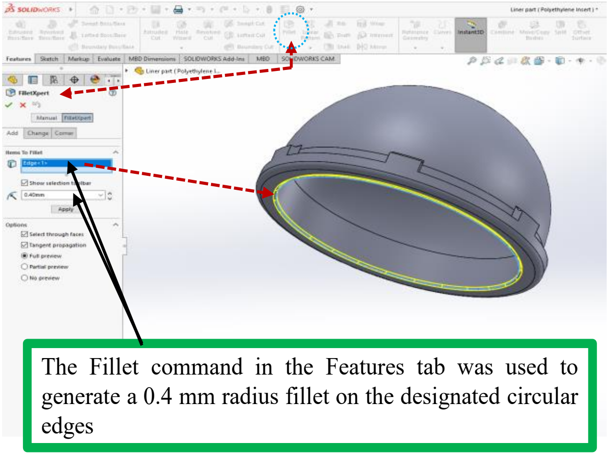


Figure 5-58. Applying a 0.4 mm-radius fillet to the lower outer edge of the liner component

Following the completion of all design features for the liner part component, its final configuration is shown in Figure 59.

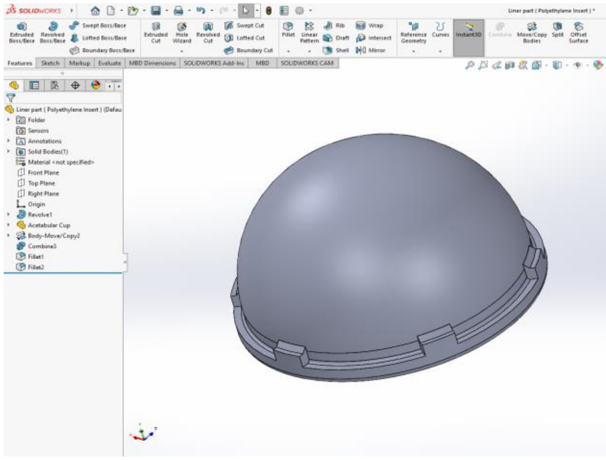


Figure 5-59. The completed liner part in its final form

The third model component in the hip prosthesis design is the Femoral Head Bearing. For the planned Femoral Head Bearing design, a new part file is first opened in the CAD software. The part file is then named according to the planned project; for this study, it will be named "Femoral Head Bearing". To create the planned sketch in the part file, the Top Plane is selected from the Feature Tree on the left-hand side of the CAD interface, and a sketching window is opened from this plane. These steps are detailed in Figure 5-60. Next, the initial 2D sketch for the planned Femoral Head Bearing model is created in the opened sketching window (Figure 5-61). After selecting the circle geometry in the 2D sketch, the upper and lower quadrants of the circle are used to determine the midpoint, and a centerline is drawn from the center of the circle to divide it into two halves. One half of the divided circle is then removed using the Trim command. In the subsequent step, the Revolved Boss/Base feature is selected to create the 3D solid model. Since the solid model will be generated by revolving, only half of the circle is sketched in the 2D drawing.

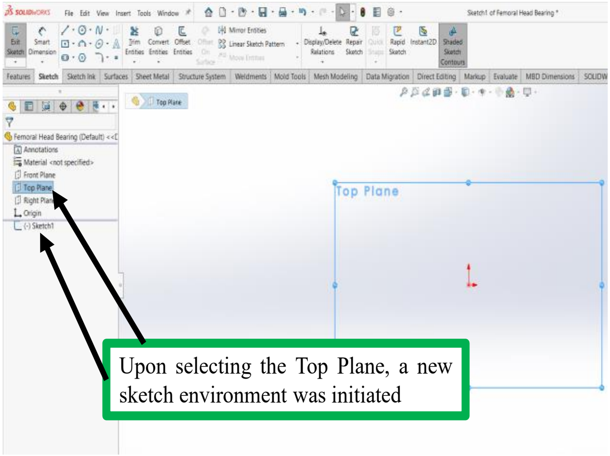


Figure 5-60. Selection of the top plane and creation of the drawing window

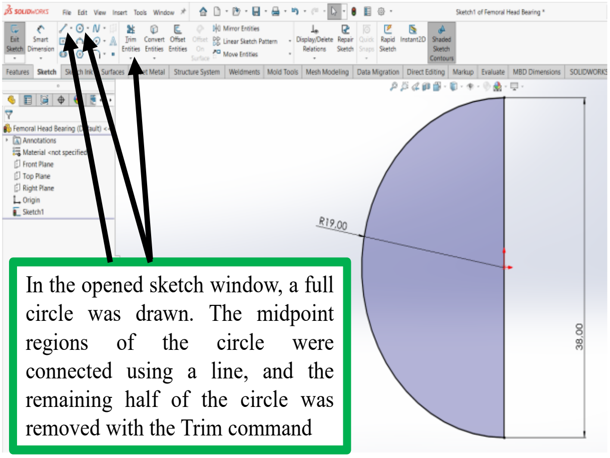


Figure 5-61. The initial stage of the 2D design of the Femoral Head Bearing model

In the next step, to convert the planned 2D sketch into a 3D solid model, the sketch file was selected, and the Features tab was accessed to choose the Revolved Boss/Base option. At this stage, the revolving axis appears, and the axis line defined during the sketching process was selected as the revolving axis. Using this axis, the first 3D solid model of the design was created (Figure 5-62).

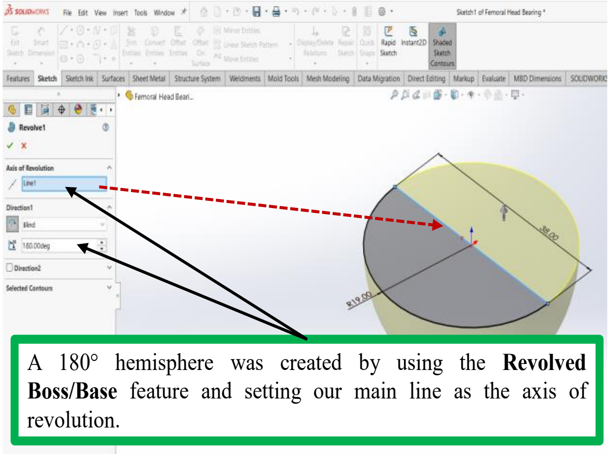


Figure 5-62. Initial development phase of the Femoral Head Bearing's 3D solid model

After creating the solid model, a new plane was generated on the bottom surface of the hemispherical Femoral Head Bearing component using the Reference Geometry option under the Features tab (Figure 5-63). A sketching window was then opened on this plane, and a circular geometry corresponding to the bottom surface of the hemisphere was drawn (Figure 5-64).

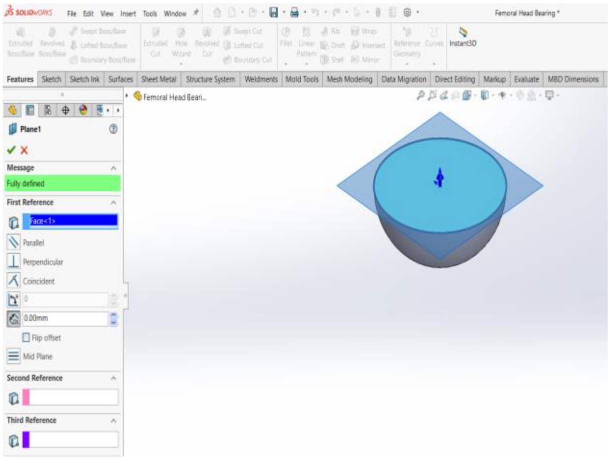


Figure 5-63. Creating a new plane on the bottom surface of the Femoral Head Bearing solid model

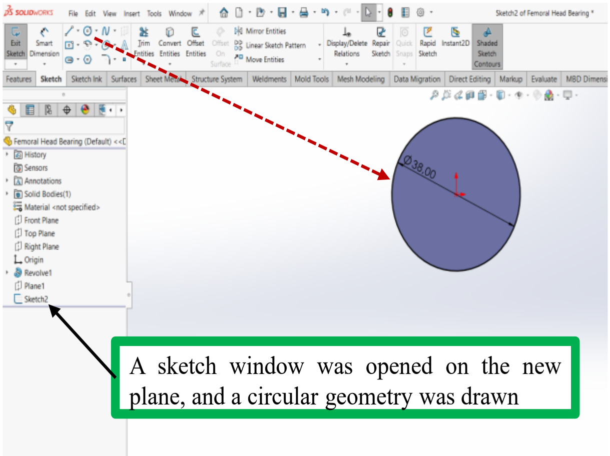


Figure 5-64. Initiation of a sketch on the bottom face of the Femoral Head Bearing model followed by the creation of a circular profile

To convert the 2D circular geometry into a solid model, the Extruded Boss/Base command under the Features tab was used to create a solid with a thickness of 6 mm (Figure 5-65). The newly created solid model was merged with the previously generated solid model to ensure a single, unified component.

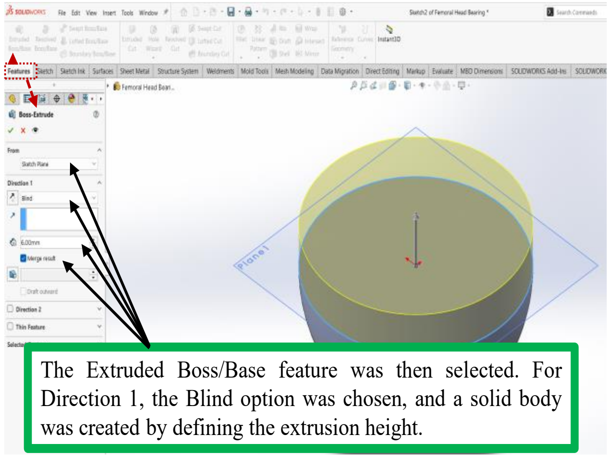


Figure 5-65. Generating a second solid model merged into the Femoral Head Bearing

In Figure 5-65 a new plane was created on the surface of the newly added solid model to plan the formation of a socket for assembling the Femoral Head component. For this purpose a new plane was generated through the Reference Geometry Plane option under the Features tab (Figure 5-66). The newly created plane was then selected and a sketching window was opened on it. A semicircular geometry was drawn in the opened sketching window (Figure 5-67).

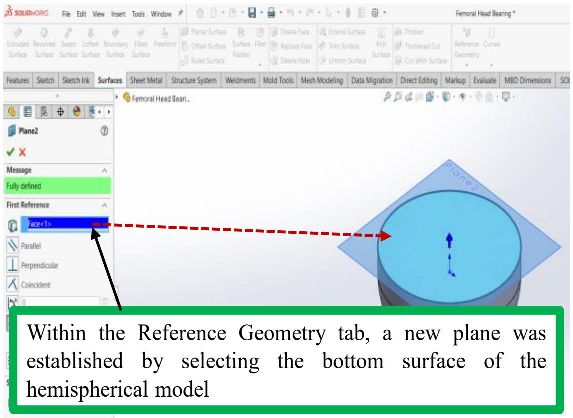


Figure 5-66. Creating a new plane on the bottom surface of the Femoral Head Bearing solid model

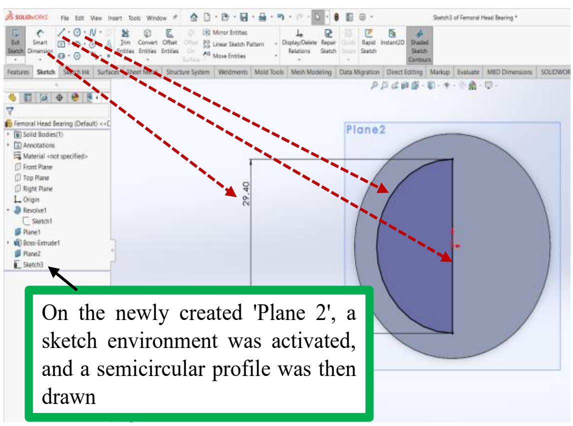


Figure 5-67. Creating a semicircular profile within the sketch initiated on the new plane at the bottom surface of the Femoral Head Bearing

The semicircular sketch was then used as a reference to create the targeted spherical socket using the Revolved Cut command under the Features tab (Figure 5-68). In the next step, a fillet was applied to the outer circular edge on the bottom surface of the Femoral Head Bearing solid model to soften the sharp geometry. This operation was performed using the Fillet command under the Features tab, with a fillet radius of 1 mm (Figure 5-69).

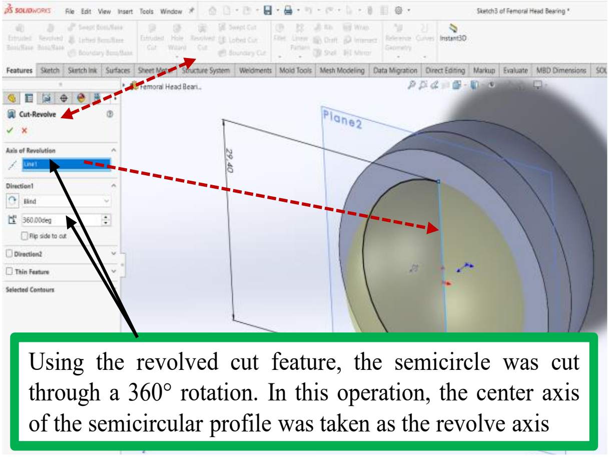


Figure 5-68. Creating the hemispherical cavity on the base of the Femoral Head Bearing solid model by applying a revolve cut

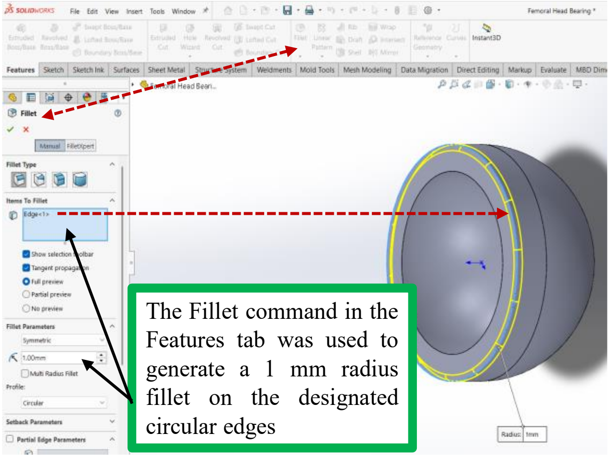


Figure 5-69. Applying a 1 mm radius fillet to the outer edge of the solid model at the bottom of the Femoral Head Bearing model

The final view of the completed solid model of the Femoral Head Bearing component is shown in Figure 5-70. The fourth model component in the hip prosthesis design is the Femoral Head. For the planned Femoral Head design, a new part file is first opened in the CAD software.

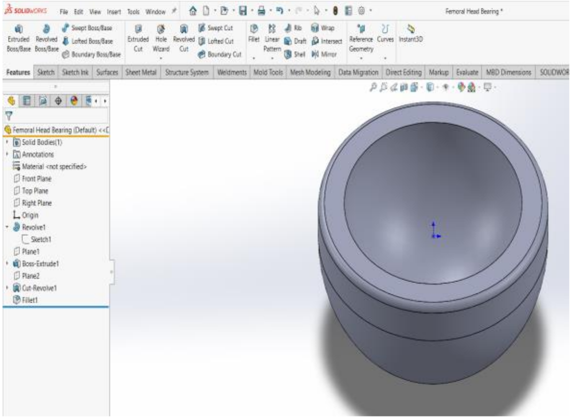


Figure 5-70. The completed Femoral Head Bearing in its final form

The part file is then named according to the planned project; for this study, it will be named "Femoral Head". To create the planned sketch in the part file, the Top Plane is first selected from the Feature Tree on the left-hand side of the CAD interface. A sketching window is then opened on this plane. These steps are detailed in Figure 5-71.

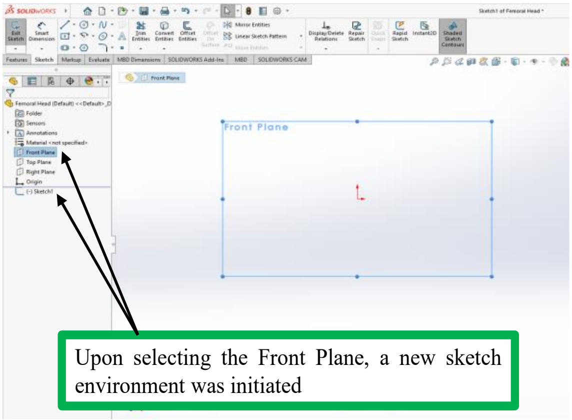


Figure 5-71. Selection of the front plane and creation of the drawing window

Next, the initial 2D sketch for the planned Femoral Head model was created in the opened sketching window (Figure 5-72).

After selecting the circle geometry in the 2D sketch, the central axis of the circle was defined and a centerline was drawn from the center of the circle to divide it into two halves. One half of the divided circle was then removed using the Trim command. In the subsequent step, the Revolved Boss/Base feature was chosen to create the planned solid model. Since the solid model is generated by revolving, only half of the circle is sketched in the 2D drawing.

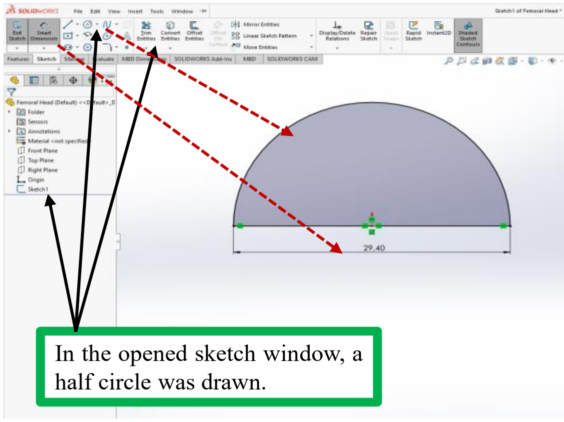


Figure 5-72. The initial stage of the 2D design of the Femoral Head component

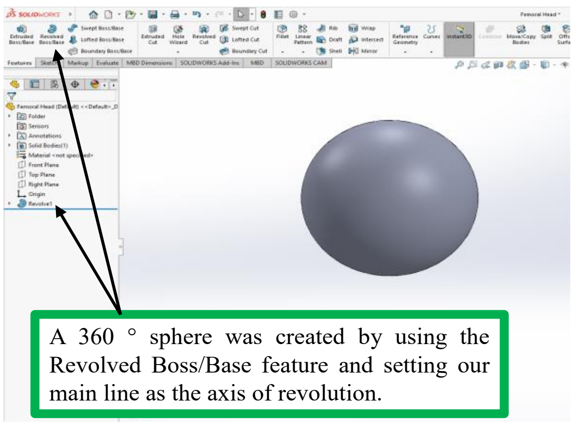


Figure 5-73. Initial development phase of the Femoral Head's 3D solid model

Next, to convert the planned 2D sketch into a 3D solid model, the sketch file was selected, and the Revolved Boss/Base option under the Features tab was accessed. At this stage, the revolving axis appeared, and the axis line defined during the sketch was selected to generate the first 3D solid model (Figure 5-73). Subsequently, to create the region where the Femoral Stem component will connect with the Femoral Head component, a new plane was defined on the previously sketched spherical part. For this purpose, the Reference Geometry Plane option under the Features tab was selected. The original Front

Plane was used as a reference, and the new plane was applied to intersect with this Front Plane (Figure 5-74).

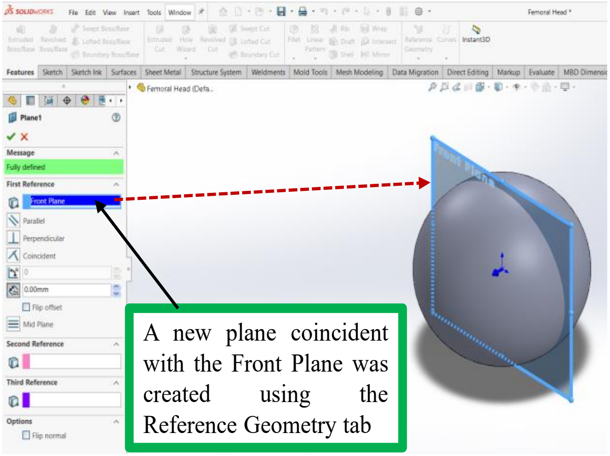


Figure 5-74. Creating a new plane on the femoral head solid model using the Front Plane as the reference

A sketching window was opened on the created Front Plane, and a 2D sketch was drawn to define the socket region where the connection for the Femoral Stem component will be formed (Figure 5-75).

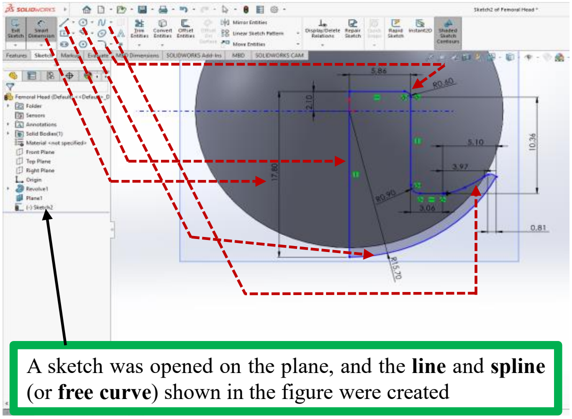


Figure 5-75. Designing the 2D interface geometry for the connection between the femoral head and the femoral stem

Next, using the centerline in the created sketch as a reference, the Revolved Cut command was applied to create the solid cut model of the internal socket (Figure 5-76).

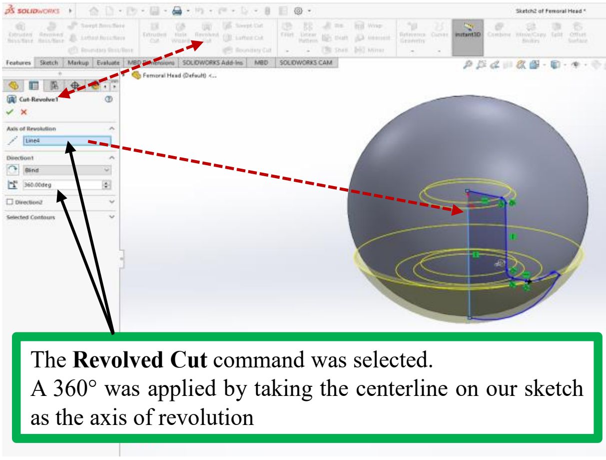


Figure 5-76. Creating the spherical cavity on the base of the Femoral Head solid model by applying a revolve cut

In the subsequent step, a 3 mm fillet was applied to the outer circular edge shown in Figure 5-77 using the Fillet command.

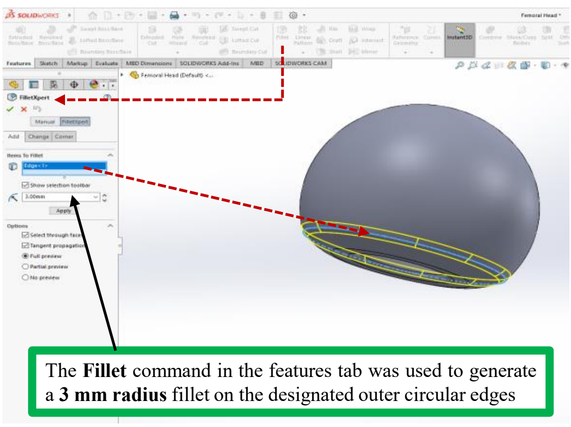


Figure 5-77. Applying a 3 mm radius fillet to the outer circular edge of the solid model of the Femoral Head model

The final view of the completed Femoral Head component after all detailed design steps is shown in Figure 5-78.

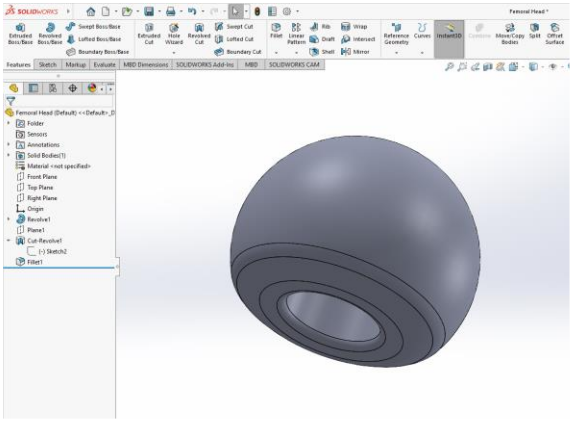


Figure 5-78. The completed appearance of the Femoral Head component

The fifth model component in the hip prosthesis design is the Femoral Stem. For the planned Femoral Stem design, a new part file is first opened in the CAD software. The part file is then named according to the planned project. For this study, the part file will be referred to as Femoral Stem.

To create the planned sketch in the part file, the Front Plane was first selected from the Feature Tree on the left-hand side of the CAD interface. A sketching window was then opened on this plane. These steps are shown in detail in Figure 5-79.

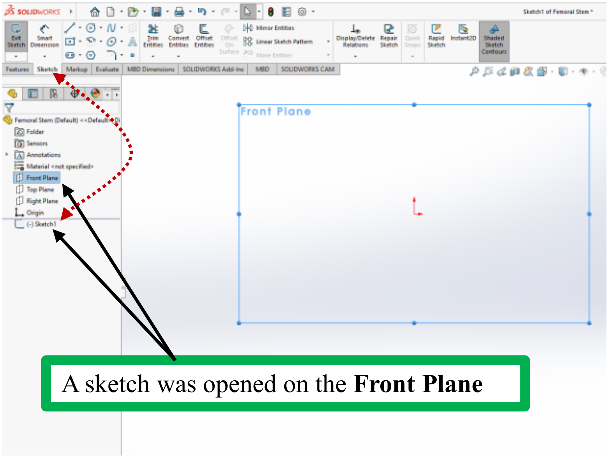


Figure 5-79. Selection of the front plane and creation of the drawing window

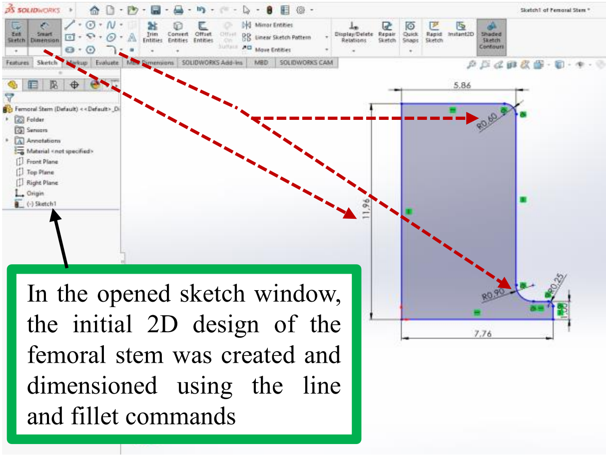


Figure 5-80. The initial stage of the 2D design of the Femoral Stem model

Next, the initial 2D sketch for the planned Femoral Stem model was created in the opened sketching window (Figure 5-80). Since the solid model will be generated using the Revolved Boss/Base feature, only half of the geometry is sketched in the 2D drawing. In the subsequent step, to convert the defined 2D sketch into a 3D solid model, the sketch file was selected, and the Revolved Boss/Base option under the Features tab was accessed. The axis line defined during the sketch was selected as the revolving axis, and the first 3D solid model was created (Figure 5-81).

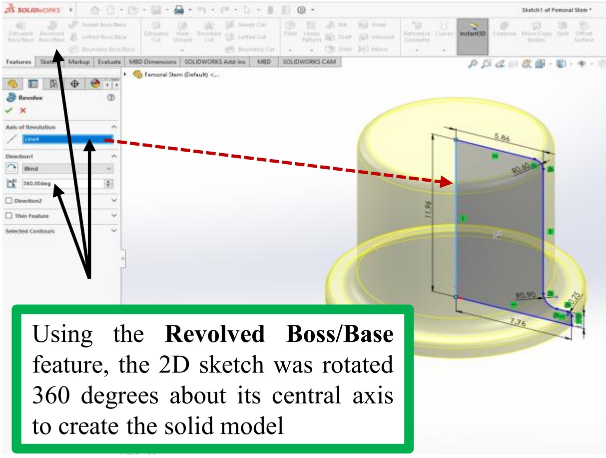


Figure 5-81. Initial development phase of the Femoral Stem's 3D solid model

Next, to make certain modifications to the initial solid model of the Femoral Stem, a new sketching window was opened on the Front Plane, and additional 2D features were drawn (Figure 5-82). The created sketch was then used to modify the initial solid model of the Femoral Stem using the Revolved Cut feature. For this purpose, the Revolved Cut option under the Features tab was selected, Sketch 2 was chosen, and the cutting operation was applied (Figure 5-83).

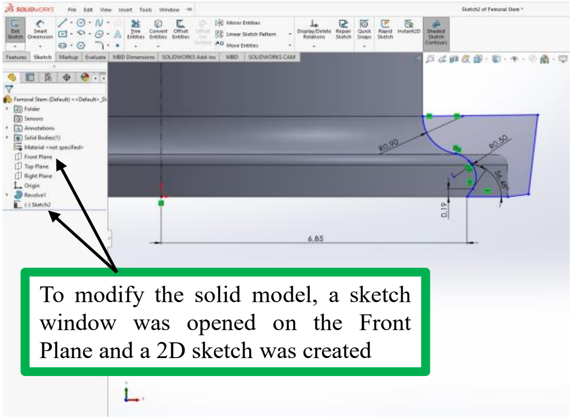


Figure 5-82. Creating a new sketch on the Front Plane of the Femoral Stem solid model

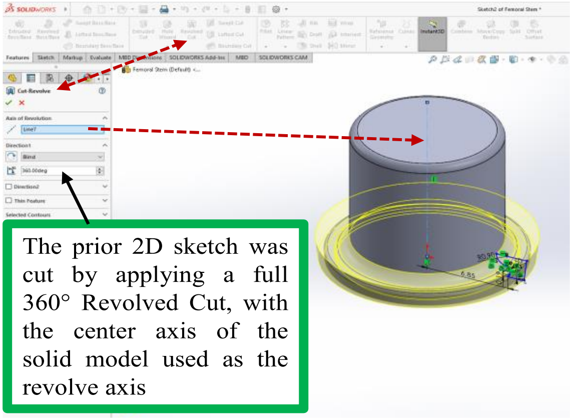


Figure 5-83. Applying a revolved cut on the Femoral Stem solid model using Sketch 2

After completing the solid modeling of the upper part of the Femoral Stem, a new sketching window was opened on the Front Plane, and a 2D design sketch for the lower body's initial section was created (Figure 5-84). In the next step, to convert the created sketch into a solid model, the Extruded Boss/Base feature under the Features tab was selected, and a thickness of 10 mm was applied to the sketch in Sketch 3 (Figure 5-85).

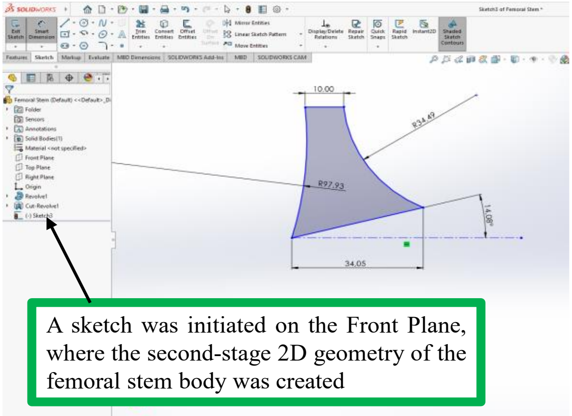


Figure 5-84. Initial 2D design of the Femoral Stem body

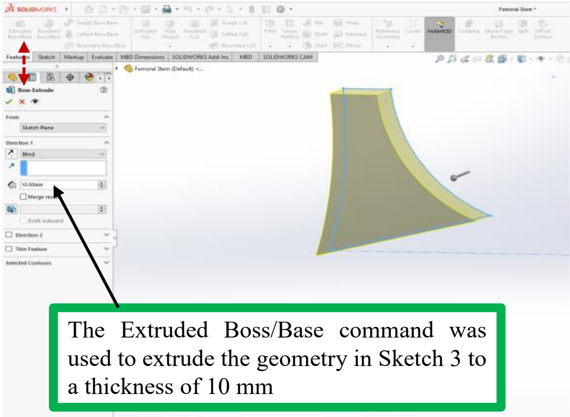


Figure 5-85. Creation of the solid model of the upper section of the Femoral Stem body

Subsequently, intermediate steps were performed to merge the initially created head solid model of the Femoral Stem with the solid model of the upper body. First, a new plane was created on the top surface of the upper body solid model using Reference Geometry Plane under the Features tab (Figure 5-86). A sketching window was then opened on the newly created plane, and a circular geometry was drawn (Figure 5-87).

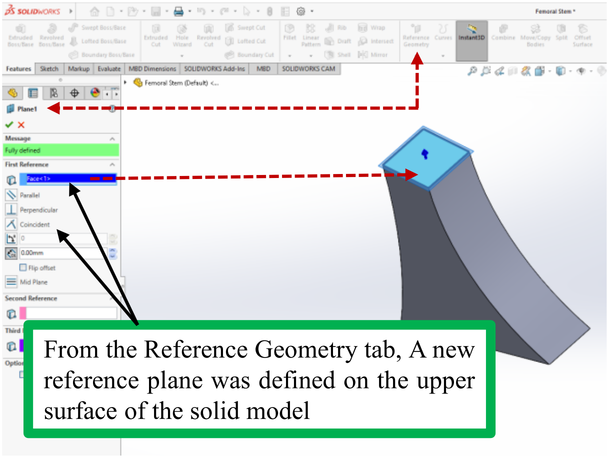


Figure 5-86. Creating a plane on the top surface of the upper body of the Femoral Stem

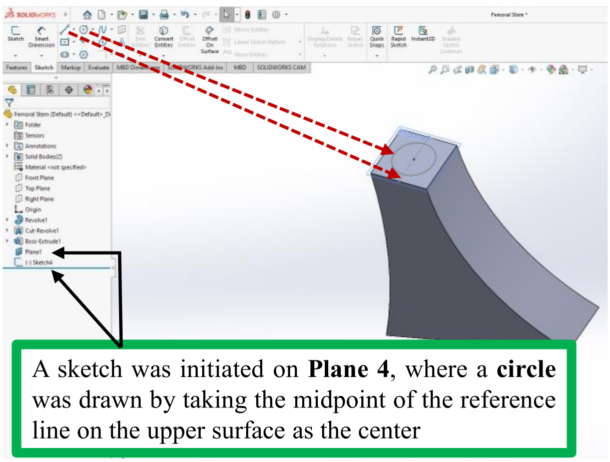


Figure 5-87. Drawing circular geometry on the top surface of the upper body of the Femoral Stem

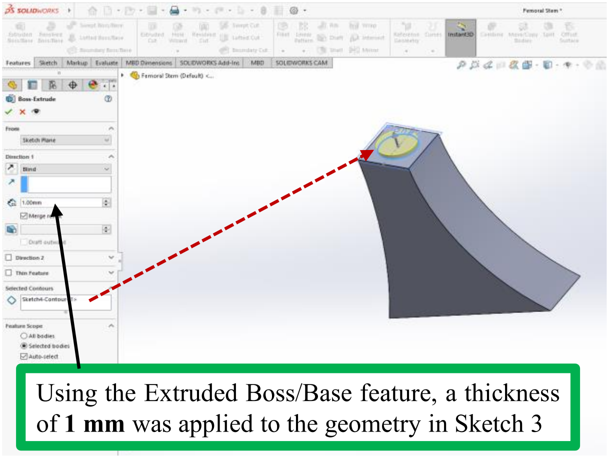


Figure 5-88. Converting the circular geometry drawn on the top surface of the upper body of the Femoral Stem into a solid model with a thickness of 1 mm

Subsequently, the existing circular geometry was given a thickness of 1 mm using the Extruded Boss/Base feature (Figure 5-88). In the next step, the Femoral Stem upper head solid model and the Femoral Stem upper body component, whose solid models were now prepared, were centered in the same position to establish their connection. For this purpose, the Move/Copy Bodies feature under the Features tab was selected. Since the upper body of the Femoral Stem was created last, the other solid model was considered fixed, and thus the upper body was moved. Next, the top circular edge of the extruded circular solid (created in Figure 5-88) and the outer surface of the Femoral Stem head solid model were selected to align them centrally (Figure 5-89-A). Then, the bottom surface of the Femoral Stem head and the top surface of the upper body, specifically the rectangular geometry surface, were selected to ensure contact between the two separate components (Figure 5-89-B). Although the two parts were brought into contact and connected within the same sketching operation, they still remain as two separate solid models.

In the next step, the Combine feature under the Features tab was used to select the two semi-solid models and merge them into a single solid model (Figure 5-90). When the Combine feature was activated, the Add option was selected to perform the merging operation. Subsequently, the merged components of the Femoral Stem will be referred to as the upper section of the Femoral Stem.

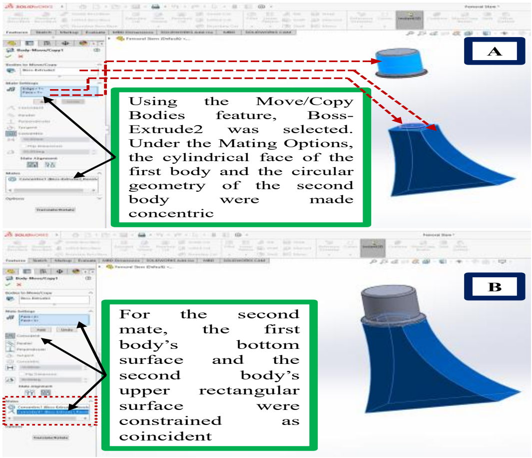


Figure 5-89. Alignment and coincidence relationships of the Femoral Stem components

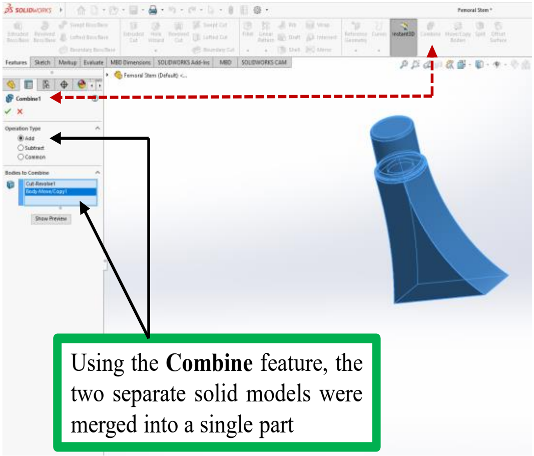


Figure 5-90. Merging the Femoral Stem components using the Combine feature

In the next step, a 2 mm fillet was applied to the selected edges on the upper section of the Femoral Stem using the Fillet feature under the Features tab (Figure 5-91). Subsequently, on the neck region of the upper section, an edge was selected, and the Fillet feature was activated again. Using the Circular Fillet option, a 1.5 mm radius was applied to all surrounding edges along the selected edge (Figure 5-92).

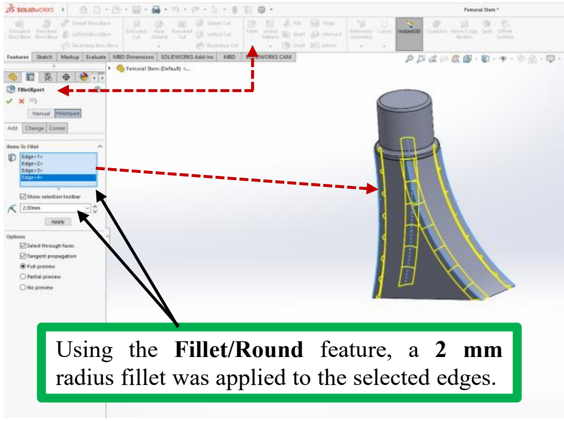


Figure 5-91. Applying a 2 mm fillet to the selected edges on the upper section of the Femoral Stem

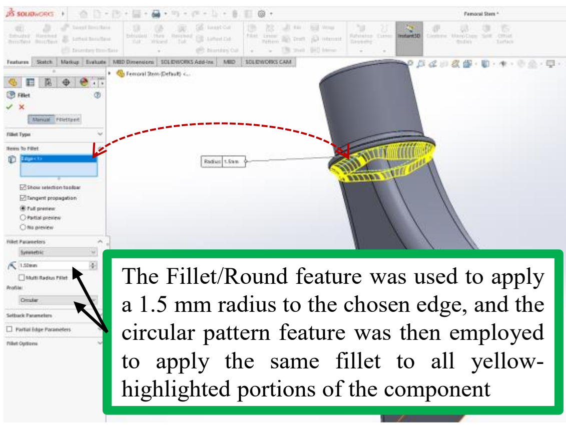


Figure 5-92. Applying a 1.5 mm fillet to the selected edges on the neck region of the upper section of the Femoral Stem

The next step involves designing the lower body of the Femoral Stem. For this purpose, a new sketching window was opened on the Front Plane, and a 2D design sketch for the lower body was created (Figure 5-93). Then, using the Extruded Boss/Base feature, a thickness of 10 mm was applied to the sketch from Figure 5-93 to convert it into a solid model of the lower body (Figure 5-94).

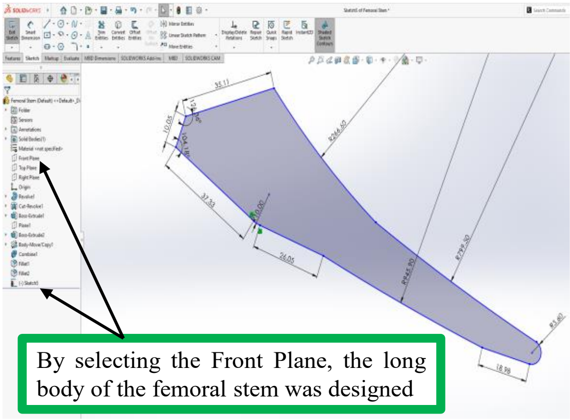


Figure 5-93. 2D design sketch of the lower body of the Femoral Stem

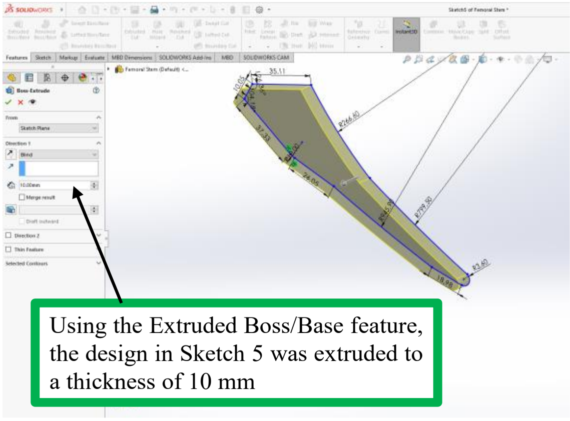


Figure 5-94. 3D solid modeling of the lower body of the Femoral Stem

In the next step, the upper section of the Femoral Stem, whose solid model was already prepared, and the newly modeled lower body component were centered in the same position to establish their connection. For this purpose, the Move/Copy Bodies feature under the Features tab was selected. Since the lower body of the Femoral Stem was created last, the other solid model was considered fixed, and therefore the lower body was moved. Subsequently, the bottom surface of the upper section and the top surface of the lower body were selected to ensure alignment and contact (Figure 5-95-A). Then, the lower edge of the upper section and the lower edge of the lower body were selected to allow the two separate components to make contact (Figure 5-95-B). Fillets with a 2 mm radius were applied to the selected edges on the lower body using the Fillet feature under the Features tab (Figure 5-96). This softened the sharp corners on the lower body, achieving a smoother design. Afterwards, the lateral surface of the lower body was selected, and a new plane was created from Reference Geometry under the Features tab (Figure 5-97).

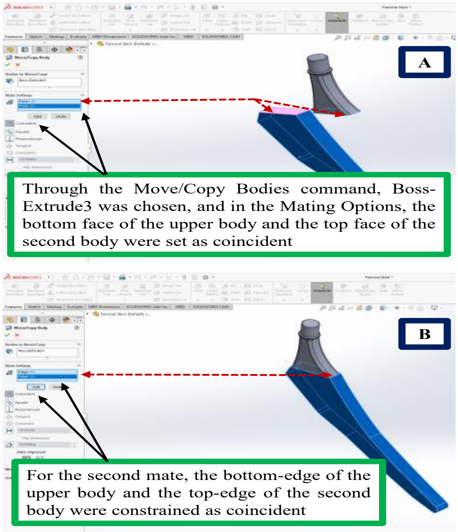


Figure 5-95. Contact and alignment relationships between the upper section and lower body components of the Femoral Stem

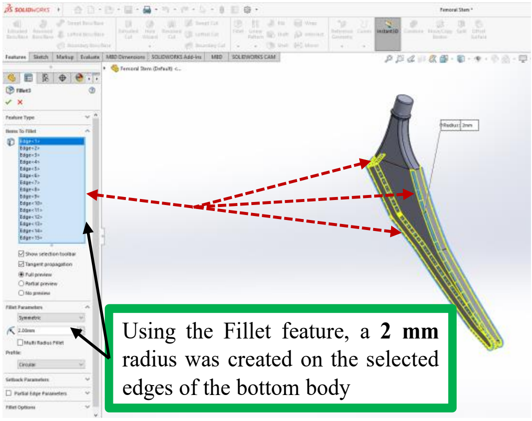


Figure 5-96. Applying a 2 mm fillet to the selected edges of the lower body of the Femoral Stem

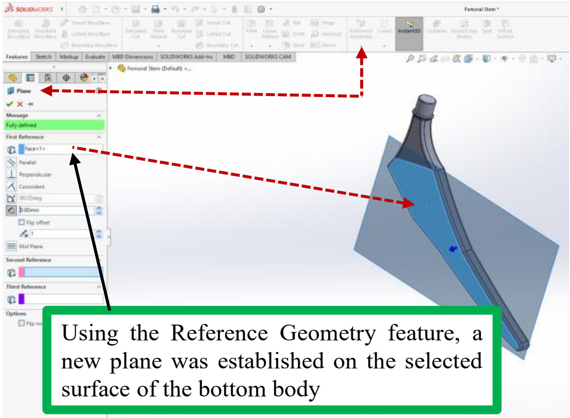


Figure 5-97. Creating a plane on the lateral surface of the lower body of the Femoral Stem

To enhance the osseointegration of the Femoral Stem lower body and facilitate its connection to trabecular bone tissue, channel and surface area augmentation designs were planned. For this purpose, an angled plane will first be created for the lower body. For this purpose, Reference Geometry under the Features tab was used to create a new plane.

The top edge of the lower body, along with the plane from Figure 5-97, was selected, and a 20° reverse-angle orientation was applied (Figure 5-98).

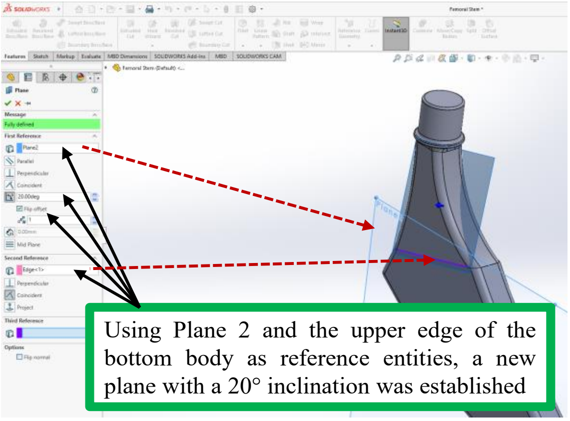


Figure 5-98. Creating a 20° angled plane on the lateral surface of the lower body of the Femoral Stem

On the lower body surface with the angled plane, a sketching window was opened, and dual-level channel geometries were drawn (Figure 5-99). Then, using the Extruded Cut feature, these geometries (from Sketch 6) were employed to create cuts on the lower body (Figure 5-100).

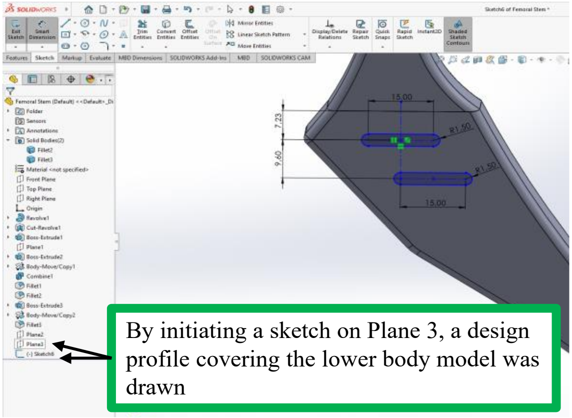


Figure 5-99. Sketching channel geometries on the lateral surface of the lower body of the Femoral Stem

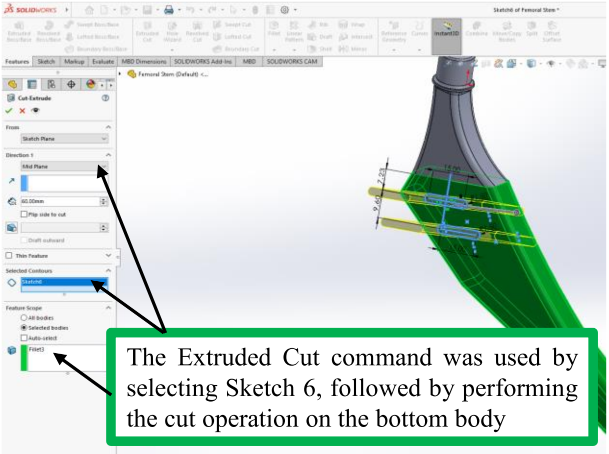


Figure 5-100. Creating the channel design on the lateral surface of the lower body of the Femoral Stem using Extruded Cut

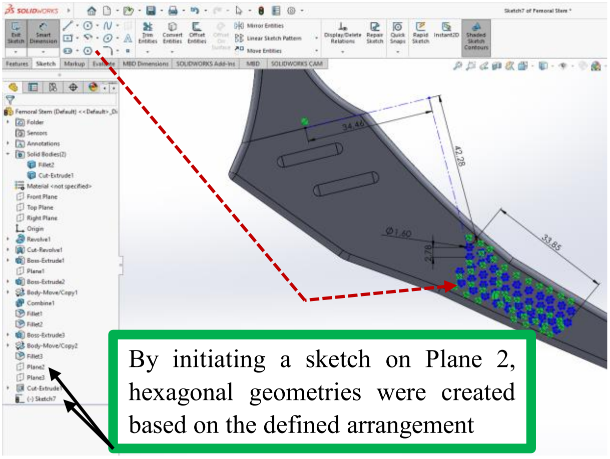


Figure 5-101. Sketching hexagonal geometries on the lateral surface of the lower body of the Femoral Stem

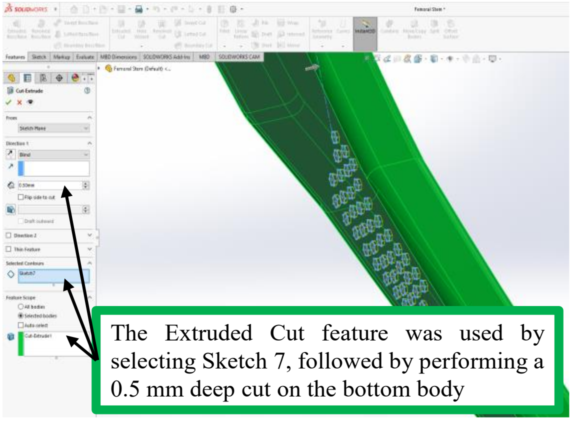


Figure 5-102. Creating hexagonal designs on the lateral surface of the lower body of the Femoral Stem using Extruded Cut

To increase the surface contact area of the lower body of the Femoral Stem, a new sketching window was opened on the lateral surface, and hexagonal geometries were designed (Figure 5-101). Subsequently, Extruded Cut operations with a depth of 0.5 mm were applied (Figure 5-102).

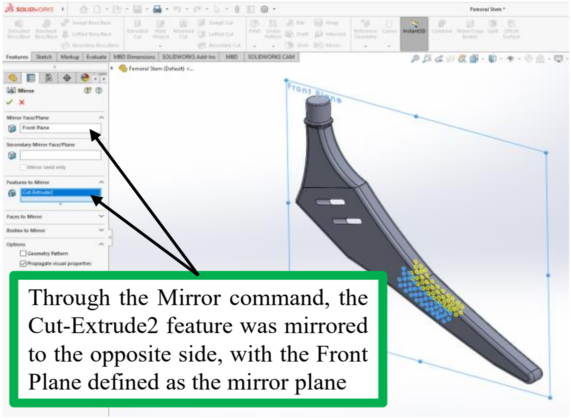


Figure 5-103. Mirroring the hexagonal design on the lateral surface of the lower body of the Femoral Stem to the opposite surface

To apply the hexagonal geometry designs symmetrically to the opposite surface of the lower body, the Mirror command under the Features tab was utilized. After activating the mirror command, the front plane was selected as the mirror axis, and the hexagonal geometry cut from Figure 5-102 was chosen as the feature to be mirrored (Figure 5-103). The edges of the channel geometry on the lower body of the Femoral Stem were selected, and a 0.5 mm fillet was applied using the Fillet feature under the Features tab (Figure 5-104).

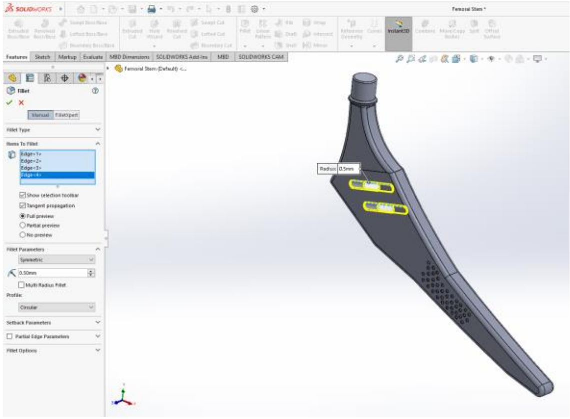


Figure 5-104. Applying a 0.5 mm fillet to the edges of the channel geometry on the lateral surface of the lower body of the Femoral Stem

In the next step, the upper region and the lower body components of the Femoral Stem were combined using the Combine command with the Add option under the Features tab (Figure 5-105).

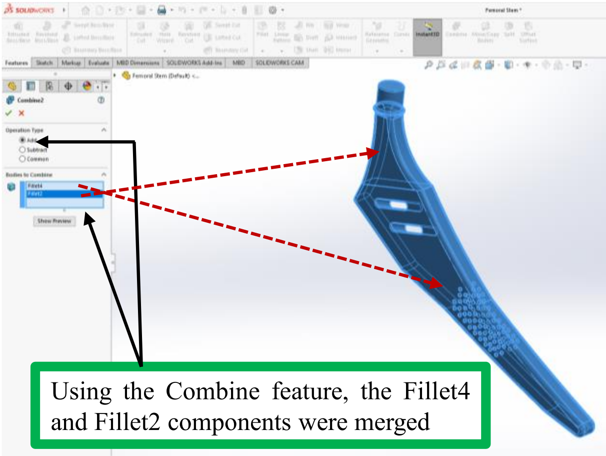


Figure 5-105. Combining the lower body and upper region of the Femoral Stem using the Combine command

The final completed solid model of the Femoral Stem component is shown in Figure 5-106. The sketches and modeling of all individual components for the hip prosthesis have been completed.

Figure 5-106. Views of the completed solid model of the Femoral Stem from different angles

The next step involves assembling these components progressively to form the total hip prosthesis. For this purpose, since the components will be combined through an assembly process, a new assembly file was created in SolidWorks and named Hip Prosthesis Assembly.

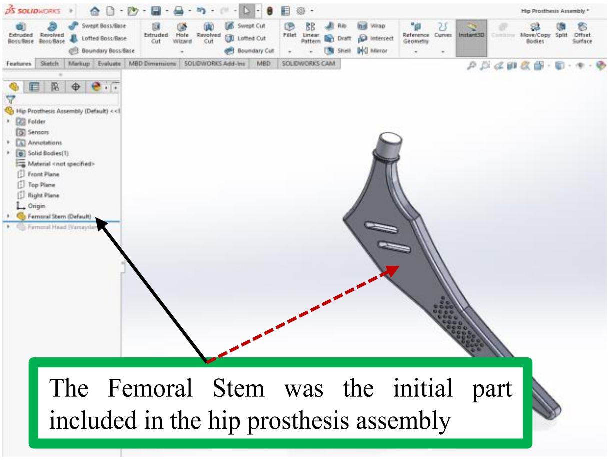


Figure 5-107. Initial insertion of the Femoral Stem component into the hip prosthesis assembly

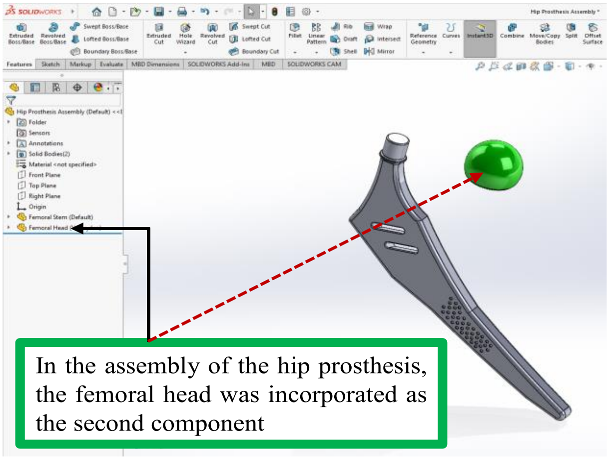


Figure 5-108. Addition of the Femoral Head component as the second step in the hip prosthesis assembly

In the first step, the Femoral Stem component, which was designed last but has the largest dimensions among all prosthesis components, was opened in this part file, and the other components were assembled onto it (Figure 5-107). Subsequently, the second component, the Femoral Head, was added (Figure 5-108). It should be noted that this assembly procedure was not performed using the SolidWorks Assembly environment but instead by utilizing the Move/Copy Bodies feature available in the Part environment. After inserting the Femoral Head component, the Move/Copy Bodies feature was activated, and the Femoral Head part was selected.

The mating relationships between the Femoral Head and the Femoral Stem were then defined. First, the top surface of the Femoral Stem and the circular inner surface of the Femoral Head were selected to establish a coincident alignment (Figure 5-109-A). Subsequently, the corresponding cylindrical surfaces of both components were selected to define a concentric alignment relationship (Figure 5-109-B). In this manner, the positioning and alignment between the Femoral Head and the Femoral Stem were completed.

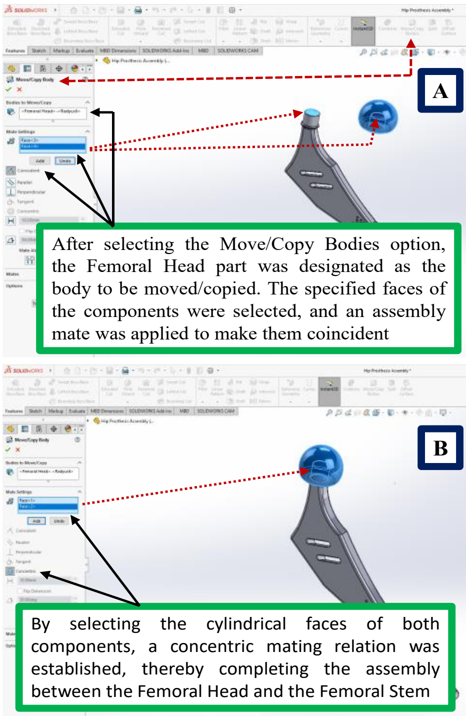


Figure 5-109. Assembly relationships between the Femoral Stem and Femoral Head components

In the next step, the newly added component was the Femoral Head Bearing. The Move/Copy Bodies feature under the Features tab was activated, and the Femoral Head Bearing component was selected. First, the circular edges of the Femoral Head Bearing and the Femoral Head components were selected to define a concentric alignment relationship (Figure 5-110-A). Subsequently, the outer spherical surface of the Femoral Head and the inner spherical surface of the Femoral Head Bearing were selected to establish a coincident relationship (Figure 5-110-B). In this way, the assembly relationships between the Femoral Head and the Femoral Head Bearing were completed.

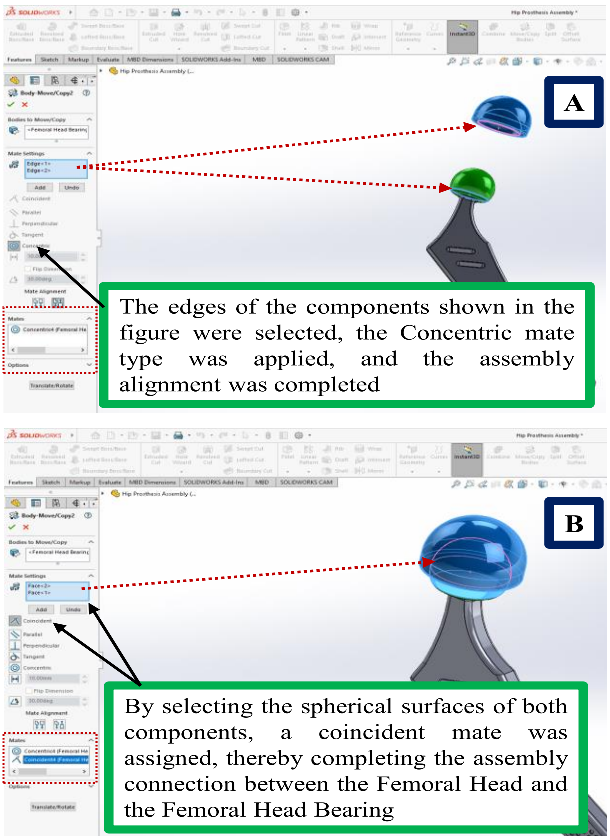


Figure 5-110. Assembly relationships between the Femoral Head and Femoral Head Bearing components

In the next step, the newly added component was the Liner Part. The Move/Copy Bodies feature under the Features tab was activated, and the Liner Part component was selected. First, the base surfaces of the Femoral Head Bearing and the Liner Part components were selected to establish a coincident relationship (Figure 5-111-A). Subsequently, the circular edges of the Liner Part and the Femoral Head Bearing components were selected to define a concentric alignment relationship (Figure 5-111-B). In this way, the assembly relationships between the Liner Part and the Femoral Head Bearing were completed.

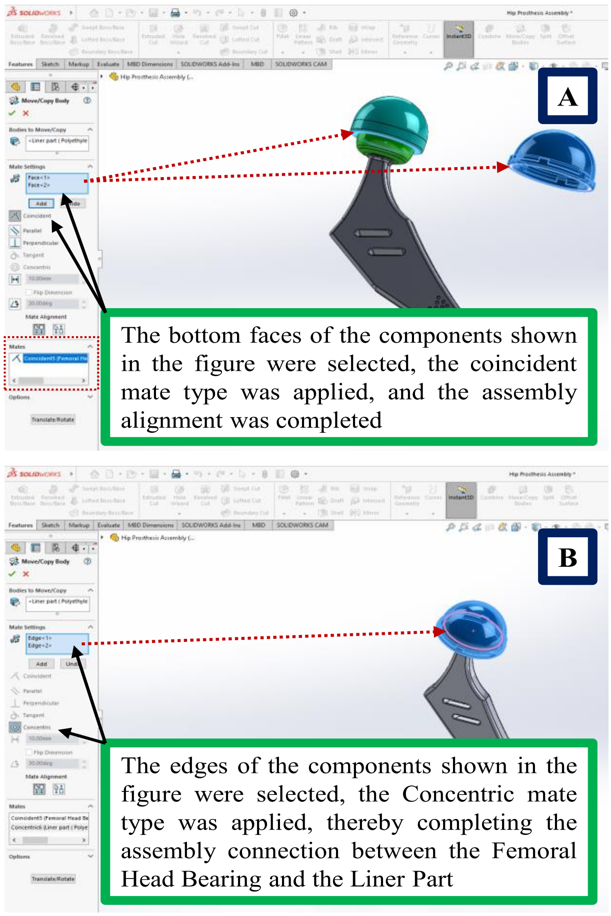


Figure 5-111. Assembly relationships between the Femoral Head Bearing and the Liner Part components

In the next step, the newly added component was the Acetabular Cup. The Move/Copy Bodies feature under the Features tab was activated, and the Acetabular Cup component was selected. First, the circular edges of the Acetabular Cup and the Liner Part components were selected to define a concentric alignment relationship (Figure 5-112-A).

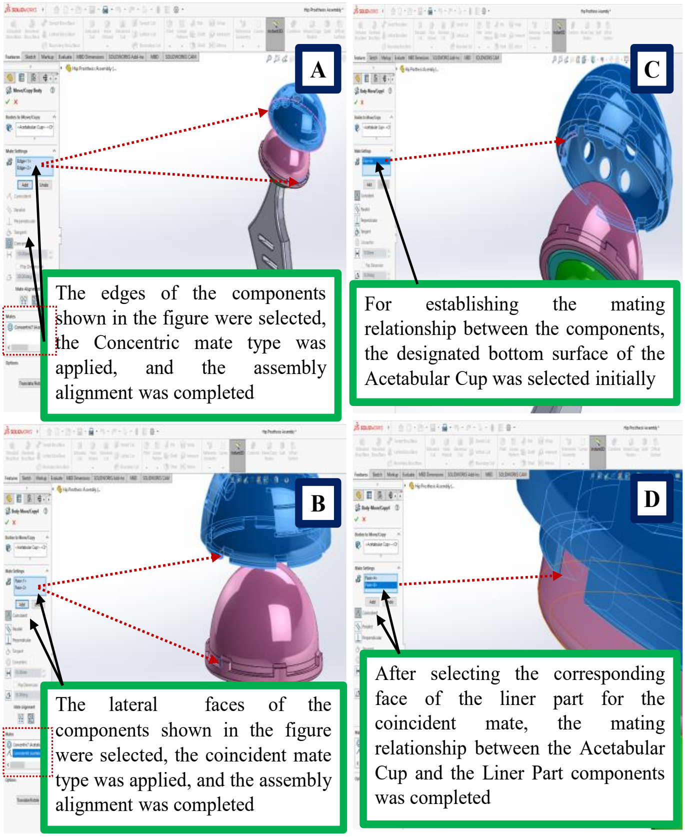


Figure 5-112. Assembly relationships between the Liner Part and the Acetabular Cup components

Subsequently, the lateral surfaces forming the connection region between the Liner Part and the Acetabular Cup components were selected, and a coincident relationship was defined (Figure 5-112-B). To complete the assembly, the base surface of the Acetabular Cup component was selected (Figure 5-112-C). The corresponding surface of the Liner Part component facing the base surface of the Acetabular Cup was then selected, and a coincident relationship was established (Figure 5-112-D). After completing the assembly of all components of the total hip prosthesis, the final view from different angles is presented in Figure 5-113.

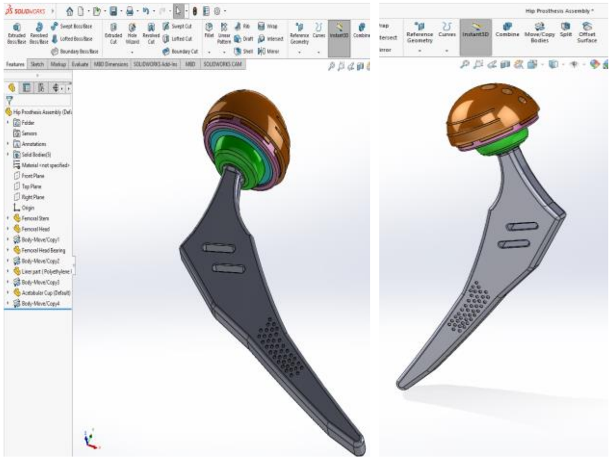


Figure 5-113. Completed assembly view of the hip prosthesis components

6. DETAILED OF THE DENTAL PROSTHESIS AND DESIGN MODEL

Over different time intervals, humans may lose all or part of their teeth due to pathological conditions or trauma. Consequently, there is a need for models that can replace the missing teeth and restore essential oral functions such as chewing and speaking, similar to the conditions prior to tooth loss [13]. For this reason, dental prostheses are effective treatment options for both fully edentulous and partially edentulous patients. Dental prostheses are classified into three main categories: fixed prostheses, removable prostheses, and implant-supported prostheses [54, 55]. Among these, fixed prostheses generally demonstrate higher clinical success rates [56, 57]. Dental prostheses are manufactured from materials such as zirconium, titanium, and metal-ceramic alloys, which are known for their long-term fatigue resistance [55, 56]. In particular, the success and long-term stability of dental implants within the jawbone are critically influenced by the appropriate selection of implant-supported prostheses. Clinical studies have shown that patients express high satisfaction with implant-supported dental prostheses. Accordingly, implant-supported prostheses can be categorized into three main types: overdentures retained by implants, implant-supported removable prostheses, and implant-supported fixed prostheses [13, 58, 59, 60]. A detailed examination of the literature reveals that among various treatment approaches, implant-supported prostheses stand out as the most prominent and successful solutions. Therefore, in the present study, a solid model of a full-arch implant-supported dental prosthesis has been developed to represent this prosthetic system

6.1. Design Model of the Full Arch Implant Supported Dental Prosthesis

For the planned design of the dental prosthesis supported by dental implants, a new part file is first created within the solid modeling software. At the very beginning of the study, the targeted project name is assigned to this part file; in this work, the name "Full-arch implant-supported dental prosthesis" will be used.

In order to generate the planned sketch in the part file, the *Front Plane* is initially selected from the feature tree located on the left panel of the solid modeling software, and a sketch environment is subsequently initiated. These procedures are illustrated in detail in Figure 6-1. Afterwards, within the newly opened sketch window, the initial 2-dimensional sketch is generated for the planned dental prosthesis model (Figure 6-2).

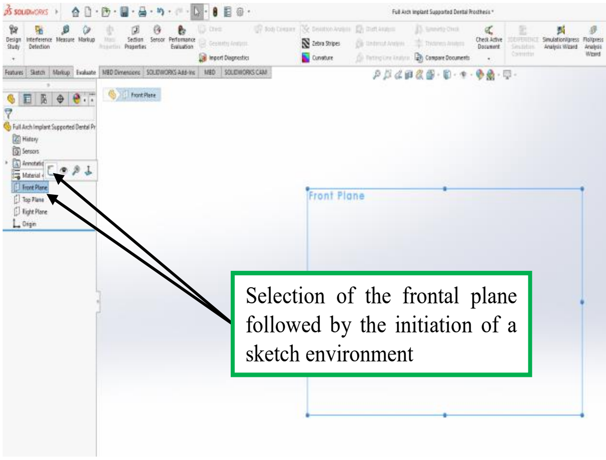


Figure 6-1. Selection of the front plane and creation of the drawing window

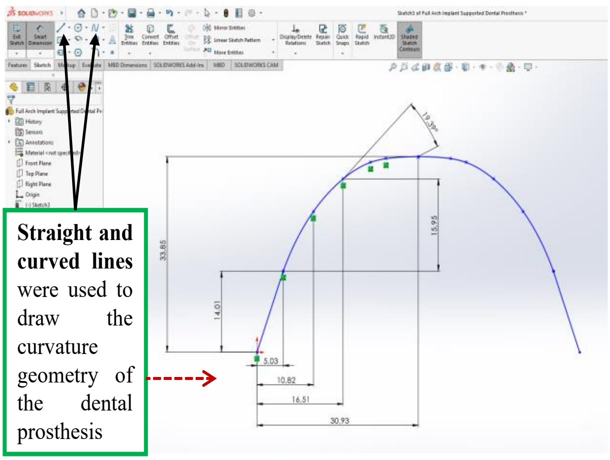


Figure 6-2. 2D Curved Geometry Path Drawing of the Dental Prosthesis

In this initial two-dimensional (2D) sketch, the curvature geometry of the mandible for the dental prosthesis was generated based on an assumed anatomical contour. (In standard practice, the mandibular curvature geometry should be reconstructed using patient-specific CT data when a custom design is intended. Subsequently, these CT data are processed in dedicated modeling software to generate the three-dimensional solid model. Within the resulting solid model, the available bone volume is evaluated, and the curvature of the dental prosthesis is established according to the optimal implant seating configuration). While drawing the 2D mandibular curvature, a combination of straight-line segments and spline/curve entities was utilized. The dimensions are presented in detail in the sketch (Figure 6-2). When constructing the cross-section (profile) of the dental prosthesis, the first step is to select the start point on the dental-prosthesis path curve, and then select the path itself to enforce normality to the curve. Next, a reference plane is created at the start point, oriented perpendicular (normal) to the curvature geometry (Figure 6-3).

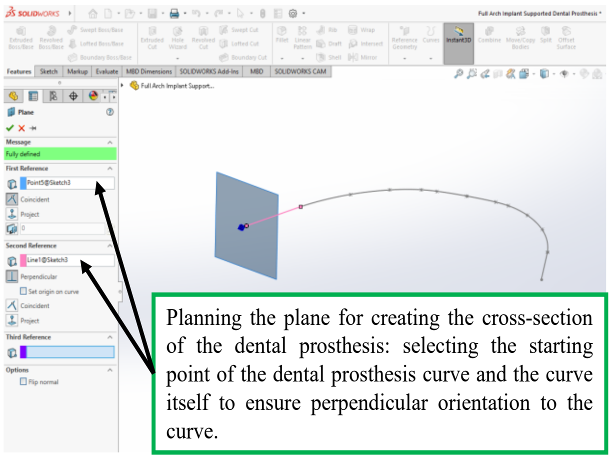


Figure 6-3. A new plane created for drawing the cross-section of the dental prosthesis

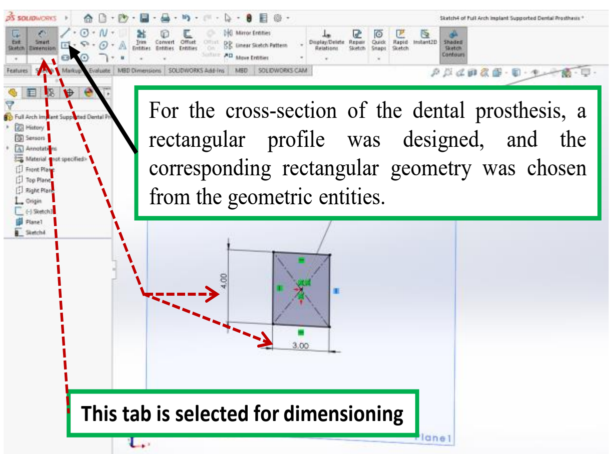


Figure 6-4. 2D cross-sectional geometry drawing of the dental prosthesis

A sketch window is then initiated on the generated plane, and the cross-sectional geometry of the planned dental prosthesis model is constructed as a two-dimensional sketch (Figure 6-4). After completing the dimensioning, this stage is finalized. During the dimensioning process, the intended dimensional values are assigned individually to each relevant edge. In the subsequent step, for the designed dental prosthesis in which the sectional profile and its corresponding trajectory (mandibular curvature path) have already been defined, the *solid sweep* feature available in the features tab is selected. Within the solid sweep dialog window, the cross-sectional geometry of the dental prosthesis (Sketch 4) and the path geometry (Sketch 3, representing the mandibular curvature design) are individually selected, and the solid sweeping operation is executed accordingly (Figure 6-5). After the solid sweep operation is applied, the resulting geometry appears as shown in Figure 6-6.

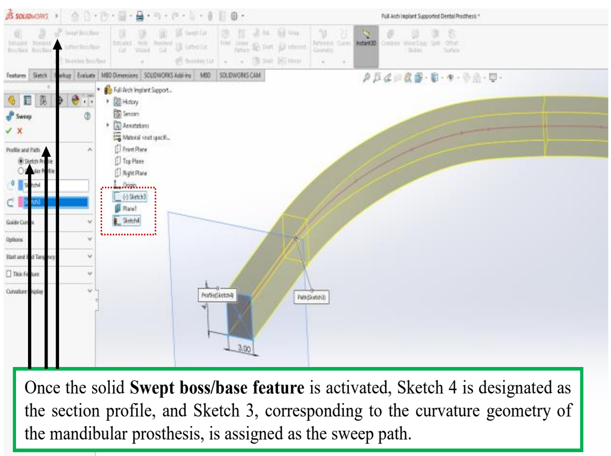


Figure 6-5. Extension of the cross-sectional geometry of the dental prosthesis using the swept boss/base command

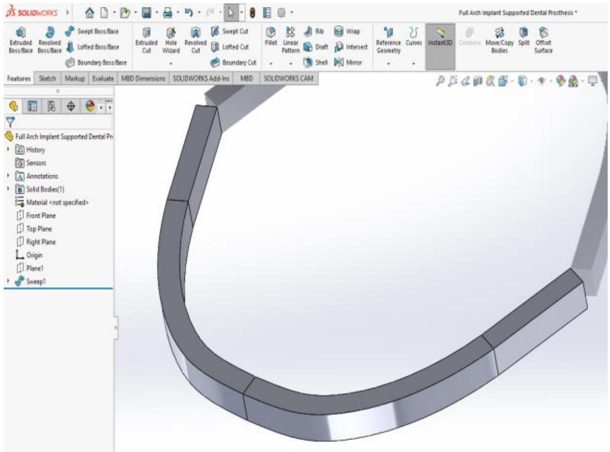


Figure 6-6. Solid model of the dental prosthesis cross-section created using the swept boss/base command

In the following stage, for the dental prosthesis whose initial solid model has now been generated, the detailed designs of both the abutment seating regions located beneath the prosthesis and the screw channels through which the prosthetic screws (that secure the prosthesis to the abutments) pass will be constructed.

For this purpose, based on the pre-defined positions of the implants and abutments, reference points are placed at the circular center locations on the prosthesis, and planes that are normal to the curvature (path) geometry are generated at these corresponding regions (Figure 6-7). In other words, this stage corresponds to establishing the reference planes required for designing the prosthetic coping interface (the bar-prosthesis–abutment connection region) on the dental prosthesis. Although the entire dental prosthesis has been modeled, the subsequent operations are intended to be performed only on half of the model, since the design will later be duplicated using the mirror command.

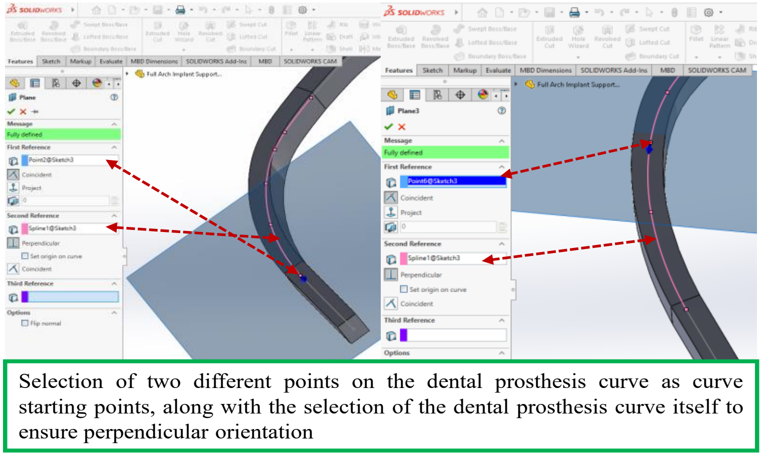


Figure 6-7. Reference planes created for designing the prosthetic coping on the dental prosthesis

Furthermore, since the total dental prosthesis is intended to be supported by four implants, two prosthetic coping regions will be constructed on the half-model of the dental prosthesis. For this reason, two separate planes were generated at two distinct locations on the dental prosthesis curvature geometry (Figure 6-7). Sketch windows were then initiated on these planes to design the prosthetic coping regions (Figure 6-8).

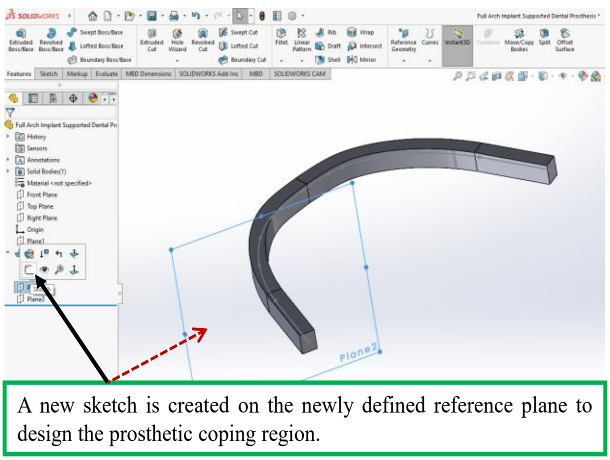


Figure 6-8. A sketch is created on the selected plane to initiate the design of the prosthetic coping region

Since the procedure and the sketching steps performed at the two designated locations are identical, only one representative location is described in detail without repeating the same explanation for each case. In the initiated sketch window, the detailed two–dimensional design of the prosthetic coping region was constructed (Figure 6-9).

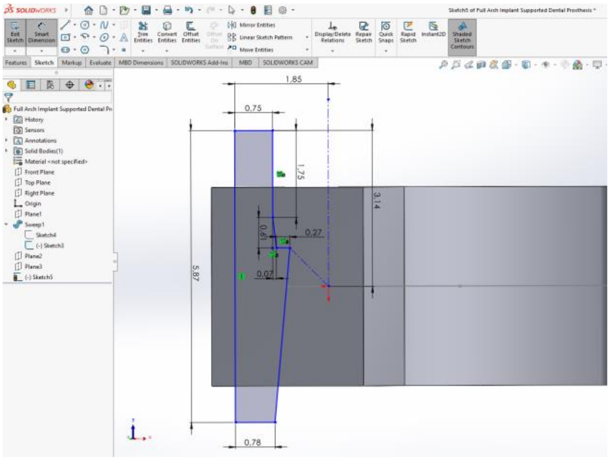


Figure 6-9. 2D design of the prosthetic coping region

These geometries were constructed under the assumption that the spatial configuration of the abutments and prosthetic screws was already known. The two–dimensional sketches generated for the prosthesis–abutment interface regions were revolved about their central axes to obtain the corresponding solid bodies (Figure 6-9). The final form of the constructed solid model is presented in Figure 6-10.

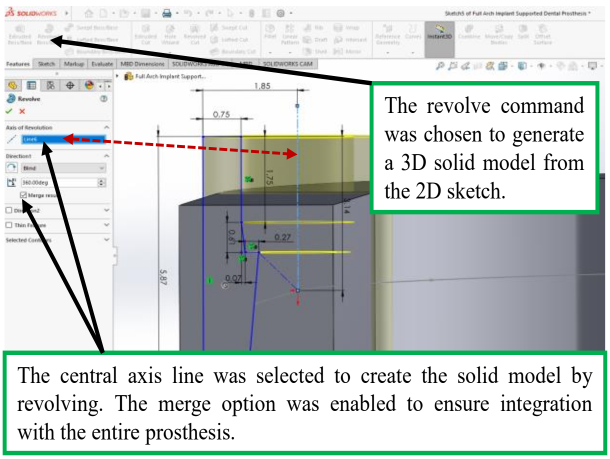


Figure 6-9. Application of the transformation from the 2D prosthetic coping sketch to the 3D solid model

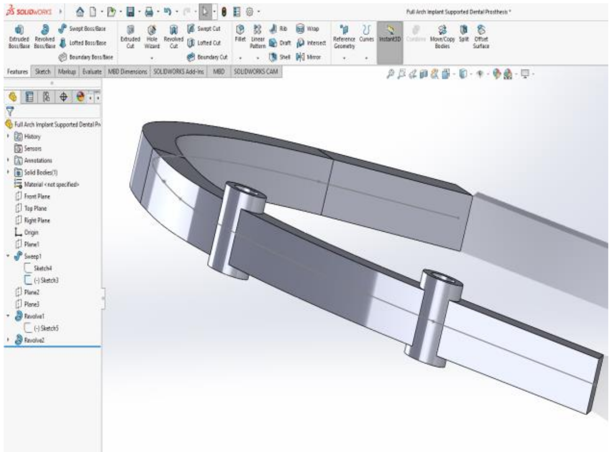


Figure 6-10. Solid model view of the prosthetic coping

Subsequently, cutting operations were performed within the solid models to generate the internal clearance required for seating the prosthetic screws.

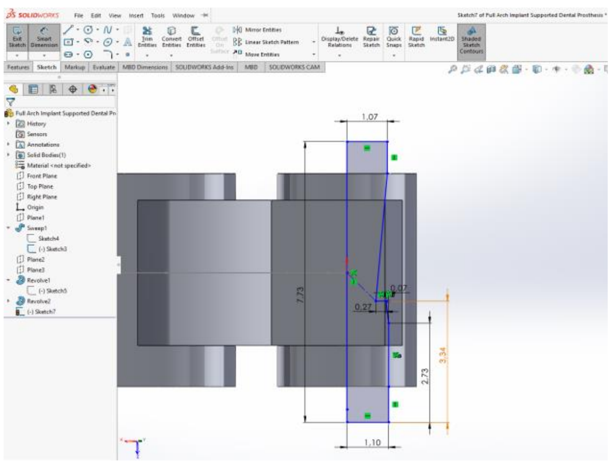


Figure 6-11. 2D design drawing for creating the inner socket part of the prosthetic coping region

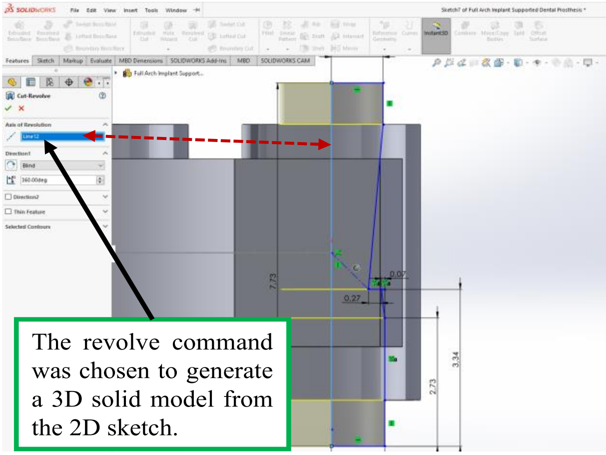


Figure 6-12. Application of the transformation from the 2D prosthetic coping socket sketch to the 3D solid model

These cutting operations were conducted by initiating a new sketch window on the previously defined prosthesis–abutment planes, and generating a two–dimensional profile sufficient to remove the required volume from the solid model (Figure 6-11). Since the purpose of this sketch was to create the internal housing of the prosthetic coping region, the *revolve cut* feature was utilized (Figure 6-12). The resulting final appearance of the prosthetic coping region after these operations is presented in Figure 6-13.

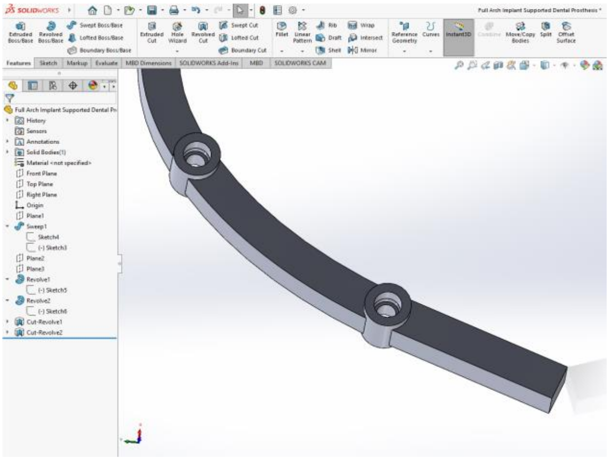


Figure 6-13. Application of the transformation from the 2D prosthetic coping socket sketch to the 3D solid model

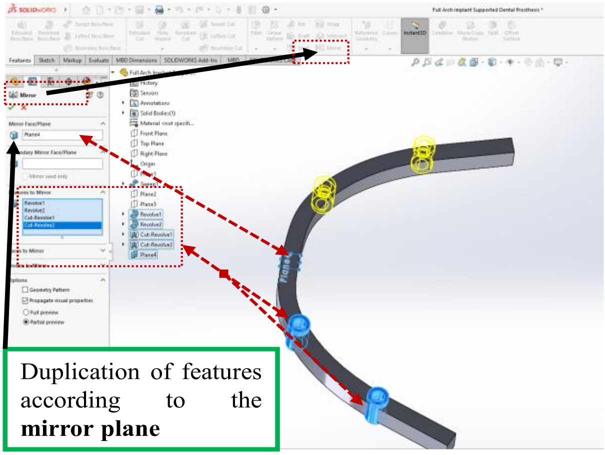


Figure 6-14. Application of the transformation from the 2D prosthetic coping socket sketch to the 3D solid model

To replicate the prosthetic coping designs whose solid models were generated on the half-section of the dental prosthesis across the entire prosthesis structure, the mirror command was utilized. For this purpose, a reference plane was first created at the mid-region of the dental prosthesis. Subsequently, the *mirror* feature was selected from the features tab. After the mirror dialog window was opened, the plane on which the mirroring would be performed was specified (Figure 6-14).

Thereafter, all solid-model elements belonging to the prosthetic coping located on the half-region of the dental prosthesis were selected, and the mirroring operation was executed accordingly. To soften the sharp edges of the dental prosthesis, filleting radii were applied using the *fillet* command (Figure 6-15). The final form of the complete dental prosthesis is presented in Figure 6-16.

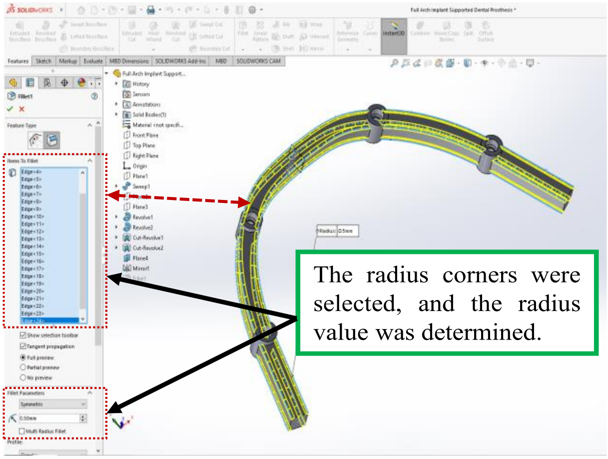


Figure 6-15. Editing the sharp edges of the dental prosthesis using the fillet command

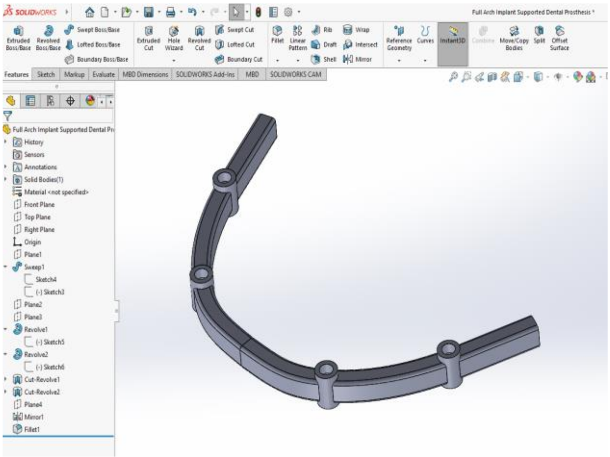


Figure 6-16. Completed view of the dental prosthesis design model

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