



Ioan SZABO



**COMPUTER-AIDED GRAPHICS
and TECHNICAL DRAWING**

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Preface

This volume represents the first edition of the teaching material developed for the Computer-Aided Graphics and Technical Drawing course, covering the fundamental commands of the Part module in SolidWorks, with particular emphasis on the Sketch and Features environments.

The content is organized progressively, ensuring a natural transition from the introductory concepts of 2D parametric sketching to the three-dimensional parametric modeling of mechanical parts. Each lecture unit provides the necessary theoretical background, while the associated laboratory sessions offer the practical context in which the acquired knowledge is applied and consolidated.

Attention has been given to learning outcomes. Each chapter follows a unified didactic structure comprising clearly defined learning objectives, theoretical and applied content supported by step-by-step demonstrations, exercises proposed for independent completion, and review questions. Complementing these, the volume includes a dedicated chapter on practical competency assessment, containing representative examination models aligned with the requirements of the course. This chapter serves as a concrete preparation and self-assessment tool for both instructors and students alike.

Upon completing this material, the student will have acquired the competencies required to create 2D parametric sketches using the Sketch module commands, to generate three-dimensional parametric models through the Features module commands, and to apply these skills effectively in the modeling of mechanical parts of varying geometric complexity.

The Author

1. FUNDAMENTALS OF COMPUTER-AIDED GRAPHICS AND TECHNICAL REPRESENTATION

1.1. Chapter Objectives

This introductory chapter familiarizes the student with the field of computer-aided graphics and its role in current engineering practice. It covers the fundamental concepts of CAD/CAM/CAE systems, the general structure of the SolidWorks platform, the working interface, and the basic principles of parametric modeling, which are necessary for the subsequent study of parts, assemblies, and technical drawing.

1.2. Learning Outcomes

Upon completion of this chapter, the student will be able to:

- explain the role of computer-aided design in modern engineering practice;
- distinguish between the functions of CAD, CAM, and CAE systems;
- identify the main modules of the SolidWorks platform and their areas of application;
- recognize the essential elements of the working interface;
- apply the basic principles of parametric modeling in the creation of simple part models.

1.3. Theoretical and Applied Content

1.3.1. The Evolution of Computer-Aided Design

Before the first computer entered a technical office, engineers worked with drafting tables, parallel rulers, and ink pens. Every modification to a drawing meant hours of corrections, and reproducing the same part across multiple sheets required considerable manual effort. Everything changed in the early 1960s, when Ivan Sutherland, a researcher at the Massachusetts Institute of Technology (MIT), developed Sketchpad the world's first interactive graphical application. For the first time, an engineer could draw directly on a computer screen, move, scale, and copy geometric shapes with a few simple commands. Although the computers of that era occupied entire rooms and carried a prohibitive cost, the seed of a revolution had been planted [1].

The decades that followed brought gradual but accelerating technological maturation. During the 1970s, major industrial companies in the aerospace, automotive, and naval sectors began investing in dedicated CAD systems developed by companies such as IBM and Dassault Systèmes. These systems were expensive, difficult to operate, and accessible only to large corporations. The real turning point came in 1982 with the launch of AutoCAD the first computer-aided design program intended for personal computers. A small design office could suddenly abandon the drafting table without spending a fortune. The rapid worldwide adoption

of AutoCAD demonstrated that CAD technology was no longer the exclusive domain of heavy industry but was becoming a standard working tool for every engineer [1].

A decisive qualitative leap occurred in the 1990s, when two-dimensional drafting began to be replaced by three-dimensional parametric modeling. The emergence of platforms such as Pro/Engineer, CATIA, and, in 1995, SolidWorks, fundamentally changed the way engineers conceive a product. Rather than drawing separate views front, lateral, and cross-section the designer now constructs a complete 3D model of the part, and the software automatically generates all required views. Any modification to the 3D model is instantly reflected across all associated production drawings, eliminating one of the most common sources of error in traditional practice. The ability to verify whether two components of an assembly interfere before either is manufactured dramatically reduced costs associated with redesign and scrap [2].

Today, computer-aided design has far exceeded the boundaries of a simple drafting tool. Modern platforms integrate structural and thermal simulations, fluid flow analysis, automatic generation of CNC toolpaths, and even generative design algorithms in which the software proposes optimized geometries based on constraints defined by the engineer. Cloud-based solutions such as Onshape and Fusion 360 have added a further dimension: teams distributed across multiple continents can work simultaneously on the same model without exchanging files by email. In the context of Industry 4.0, where digital and physical production are increasingly intertwined, proficiency in CAD tools is no longer an optional advantage but a baseline requirement for any engineer seeking to remain competitive in the labor market [3].

1.3.2. General Concepts of CAD/CAM/CAE Systems

CAM, and CAE systems represent the three fundamental components of modern computer-aided design and manufacturing. CAD (Computer-Aided Design) is the tool through which the engineer defines a product geometrically: shape, dimensions, tolerances, and materials. Unlike classical technical drawing, the CAD model is a precise mathematical representation from which production drawings, bills of materials, and technical documentation are automatically derived. Any modification propagates instantly across all associated documents, eliminating redundancy and transcription errors.

CAM (Computer-Aided Manufacturing) takes the geometric model and automatically generates programs for CNC machine tools, including tool paths, cutting speeds, and the sequence of operations. Information flows directly from design to manufacturing without manual re-entry, significantly reducing production setup time and the risk of error.

CAE (Computer-Aided Engineering) functions as a virtual laboratory in which the digital model is subjected to structural, thermal, and fluid simulations before any physical prototype is built. Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD) allow the identification and correction of design weaknesses during the design phase itself, reducing the costs and time associated with subsequent redesigns.

The three systems do not operate in isolation but form an integrated workflow known as PLM (Product Lifecycle Management). The SolidWorks platform reflects this philosophy through its Part, Assembly, Simulation, and Drawing modules, which coexist within the same environment and exchange data in real time, providing the engineer with a complete ecosystem from initial concept to final documentation [8].

1.3.3. General Overview of the SolidWorks Platform

SolidWorks is a computer-aided design (CAD) software developed by Dassault Systèmes, released in 1995 and intended for three-dimensional parametric modeling. Unlike the CAD solutions of that era, SolidWorks was designed to run on personal computers with the Windows operating system, which significantly reduced the cost of accessing 3D technology for design offices of any size.

The platform provides an integrated working environment that covers the entire product development workflow: from the initial sketch and individual part modeling, through component assembly and structural analysis, to the generation of technical documentation. All these stages take place within the same software environment, without the need to export data between different applications, ensuring information consistency throughout the entire project.

Over three decades of continuous development, SolidWorks has become one of the most widely used CAD programs worldwide, with applications spanning the automotive, aerospace, and medical industries, as well as higher technical education. The combination of advanced functionality, a large user community, and extensive documentation establishes it as a reference tool for developing computer-aided design competencies.

1.3.4. Main Modules of the SolidWorks Platform

1.3.4.1. The Part Module

The Part module (Figure 1.1) represents the starting point of any modeling process carried out in SolidWorks and constitutes the foundation upon which all subsequent stages of the design are built. Within this module, the user creates individual three-dimensional parts, starting from a two-dimensional sketch drawn on one of the reference planes of the working space and progressively transforming it into a fully defined geometric solid.

The modeling process follows a sequential and intuitive logic. The basic geometry of the part is first established through volume-generating operations, of which the most frequently used are extrusion (Extruded Boss/Base), which projects a planar sketch to a specified depth, and revolution (Revolved Boss/Base), which rotates a profile around an axis to produce shaft, flange, or ring geometries. For more complex geometries, the module provides operations such as Sweep, through which a profile is guided along a curved path, and Loft, which interpolates between two or more distinct cross-sections.

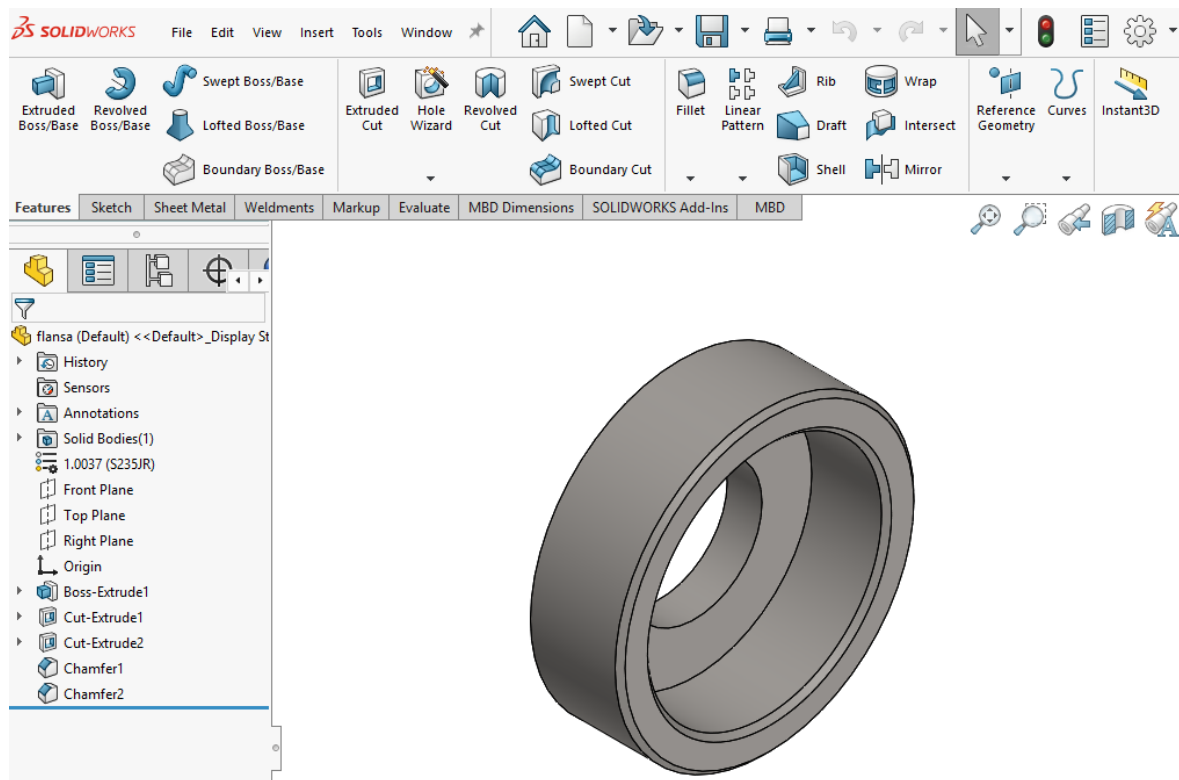


Figure 1.1 Parametric Modeling in the Part Module

Once the base volume is obtained, the model is refined through editing operations: edge rounding with fillet radii (Fillet), chamfering (Chamfer), wall thinning for shell-type parts (Shell), and the duplication of geometric elements according to a linear or circular pattern (Pattern). Internal and external threads may be represented cosmetically or modeled geometrically, depending on project requirements.

A fundamental aspect of the Part module is the parametric nature of the resulting models. Every dimension entered can be modified at any point, and the entire geometry updates automatically as a consequence. This property is particularly valuable in automotive part design, where the same component may exist in different dimensional variants for distinct vehicle models, without requiring the modeling process to be restarted from scratch. The operation history, visible in the FeatureManager Design Tree, provides complete traceability of all construction stages and allows the user to return to and modify any previous operation.

1.3.4.2. The Assembly Module

The Assembly module (Figure 1.2) represents the natural stage that follows the modeling of individual parts, in which separate components acquire meaning through their relationships with one another. While the Part module focuses the engineer's attention on the geometry of a single part, Assembly shifts the focus to how parts interact, support each other, and together form a coherent functional system.

Component assembly is achieved through geometric constraints known as Mates, which define the spatial relationships between parts: two surfaces may be declared coincident, two axes

concentric, two faces parallel, or separated by a precise distance. By progressively applying these constraints, each component loses its degrees of freedom until its position within the assembly is fully determined. This approach faithfully reflects the logic of physical assembly in a workshop and helps the student understand, prior to any hands-on experience, why a flange requires centering, why a bearing must be concentric with the shaft, or why a gasket requires planar contact with both adjacent surfaces.

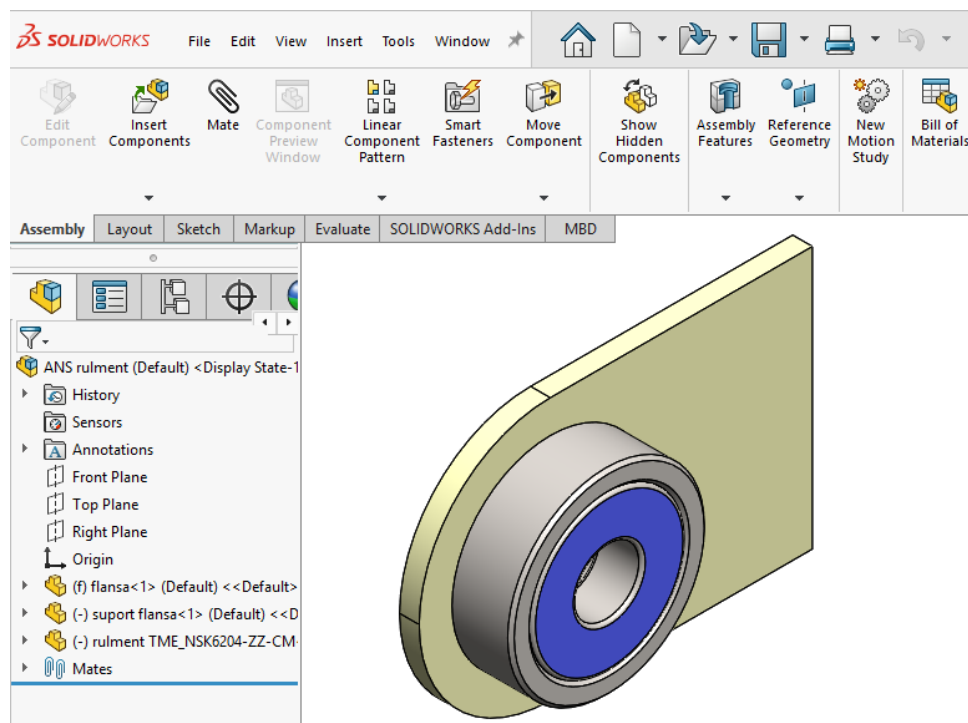


Figure 1.2 Assembly Example in SolidWorks

A major advantage of the Assembly module is its ability to automatically detect interferences between components. In the automotive industry, where hundreds of parts coexist within extremely confined spaces, digital interference checking prior to manufacturing eliminates costly errors that would otherwise only be discovered at the physical prototype stage. SolidWorks identifies and visually highlights any common volume between two parts that could not occupy the same space.

The module also allows the simulation of motion in assembled mechanisms. It is thus possible to animate a connecting rod and crank mechanism, verify the stroke of a piston, or observe the relative motion of a suspension system, gaining an understanding of the kinematic behavior of the assembly before it is physically built. This capability transforms the 3D model from a simple volumetric drawing into an analysis and validation tool for the design solution.

1.3.4.3. The Drawing Module

The Drawing module (Figure 1.3) is the stage at which the previously created three-dimensional model becomes an official technical document, suitable for use in manufacturing,

inspection, and assembly. While the earlier modules deal with building the part in virtual space, Drawing deals with communicating that part to those who will physically manufacture or assemble it.

Drawings generated in SolidWorks are directly linked to the 3D model, meaning that any modification made to the part is automatically reflected in the drawing as well. This connection eliminates one of the most common errors in classical design practice, namely the situation in which the drawing and the model no longer correspond after several rounds of revisions.

The placement of views on the sheet, the addition of dimensions, and the creation of cross-sections are performed quickly and intuitively, in compliance with applicable technical standards. Beyond geometry, the module allows the addition of dimensional tolerances, surface roughness indications, and other specifications required for the correct manufacture of the part.

Working with the Drawing module develops two complementary skills. The first is the ability to produce a correct and complete technical drawing. The second is the ability to read and interpret technical drawings received from colleagues, suppliers, or industrial partners. In the day-to-day practice of the automotive industry, both prove equally important and are used routinely in every design office.

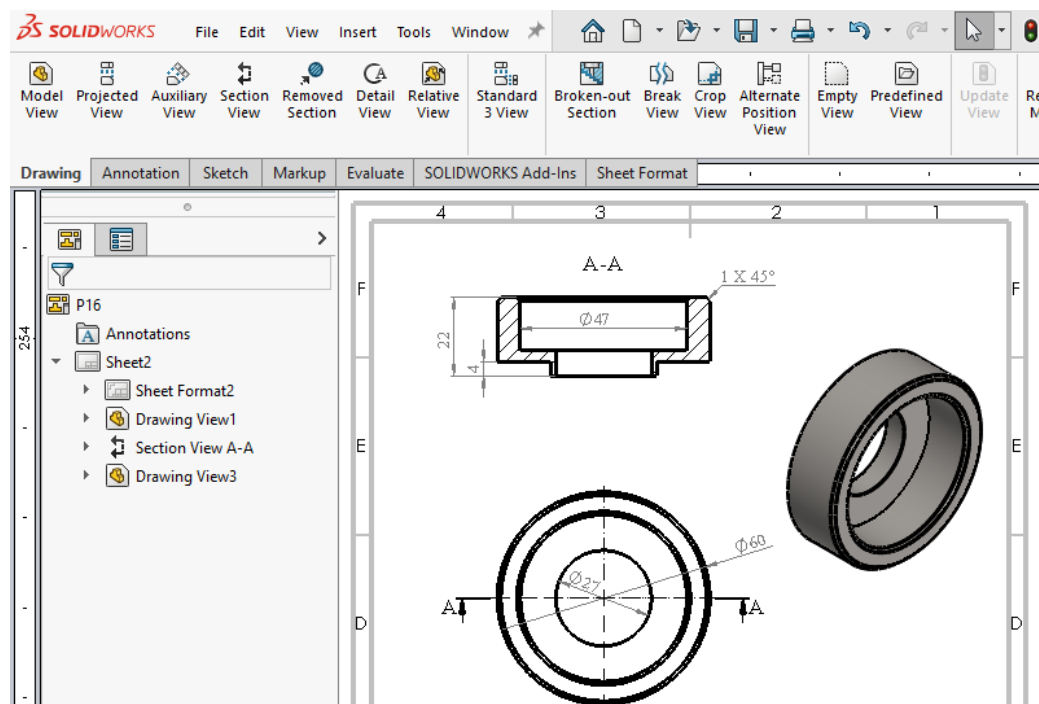


Figure 1.3 Technical Drawing Example in SolidWorks

1.3.4.4. The Sheet Metal Module

The design of sheet metal components occupies a central role in the automotive industry, and the Sheet Metal module, presented in Figure 1.4, provides the tools required to address this specific category of parts. Body panels, brackets, housings, heat shields, and numerous other

structural and functional vehicle components are produced through cold plastic deformation, and this module faithfully reflects the technological logic of these processes.

Unlike standard modeling in the Part module, Sheet Metal operates according to material-specific logic: the material thickness remains constant throughout the entire part, and the available operations correspond directly to real manufacturing processes. Bends are defined with precise radii of curvature and angles, cutouts are applied to the flat surfaces of the part, and corners are managed automatically to prevent material overlaps that would render the part unmanufacturable.

One of the most useful features of the module is the automatic flat pattern generation. Any component modeled in Sheet Metal can be unfolded with a single command, producing the flat blank geometry with the actual dimensions of the semi-finished part and the bend line markings. This flat pattern can be exported directly for laser cutting or stamping, eliminating manual calculations and reducing the risk of error in manufacturing preparation.

The relationship between the three-dimensional geometry of the part and its flat pattern is maintained permanently and bidirectionally. Any modification to the 3D geometry automatically updates the flat pattern and vice versa, making Sheet Metal a valuable tool both in design and in the analysis of existing parts, when the objective is to understand the original flat geometry of an already manufactured component.

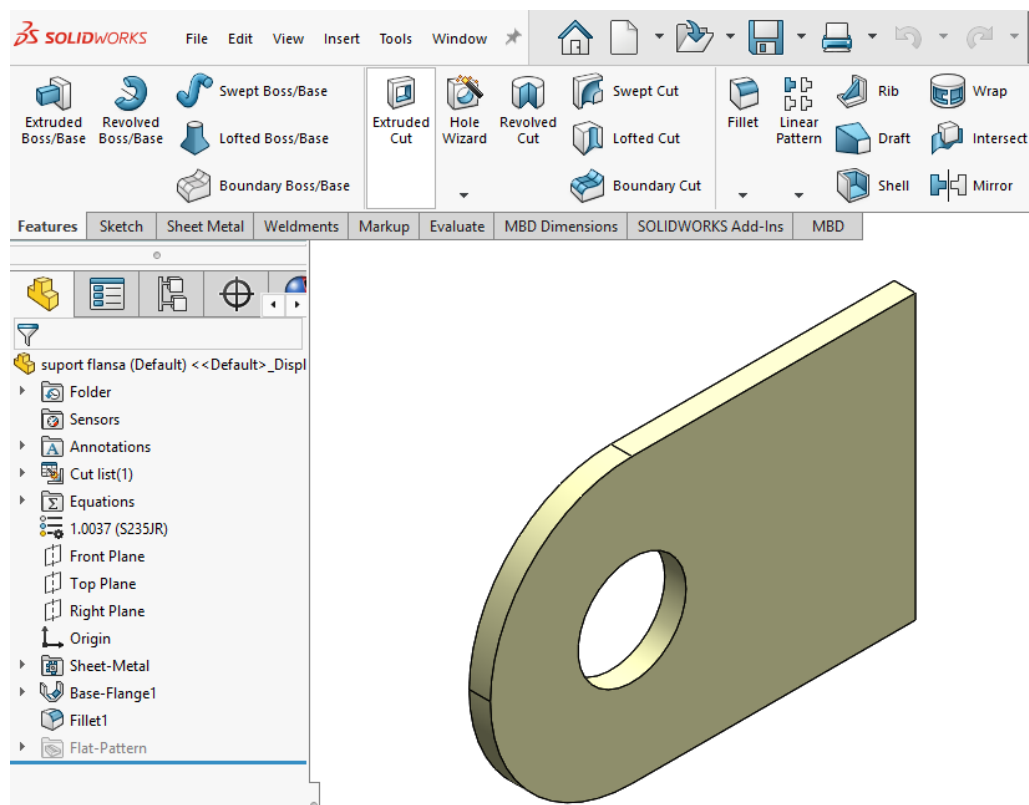


Figure 1.4 Sheet Metal Part Example in SolidWorks

1.3.4.5. The Weldments Module

Welded structures are an essential component of mechanical engineering and are present in virtually every industrial sector, from body frames and powertrain support structures to chassis sub-assemblies and working platforms for special-purpose vehicles. The Weldments module in SolidWorks (Figure 1.5) is specifically designed for the design of such structures, offering a workflow adapted to the constructive logic of welded fabrication.

The working principle of the module starts from a three-dimensional sketch of the structure, representing the axes of the constituent elements. Standardized structural profiles — such as rectangular tubes, L, U, and I sections — are automatically applied along each segment of this sketch, instantly generating the complete three-dimensional structure. The profile library covers the most common industry standards and can be extended with custom profiles tailored to the specific requirements of a given project.

At element intersections, the module automatically manages the joints, calculating and applying the end cuts required for profiles to meet correctly without material overlaps. Once the structure is complete, SolidWorks automatically generates the cut list with the exact lengths of each element, information that can be used directly in manufacturing preparation.

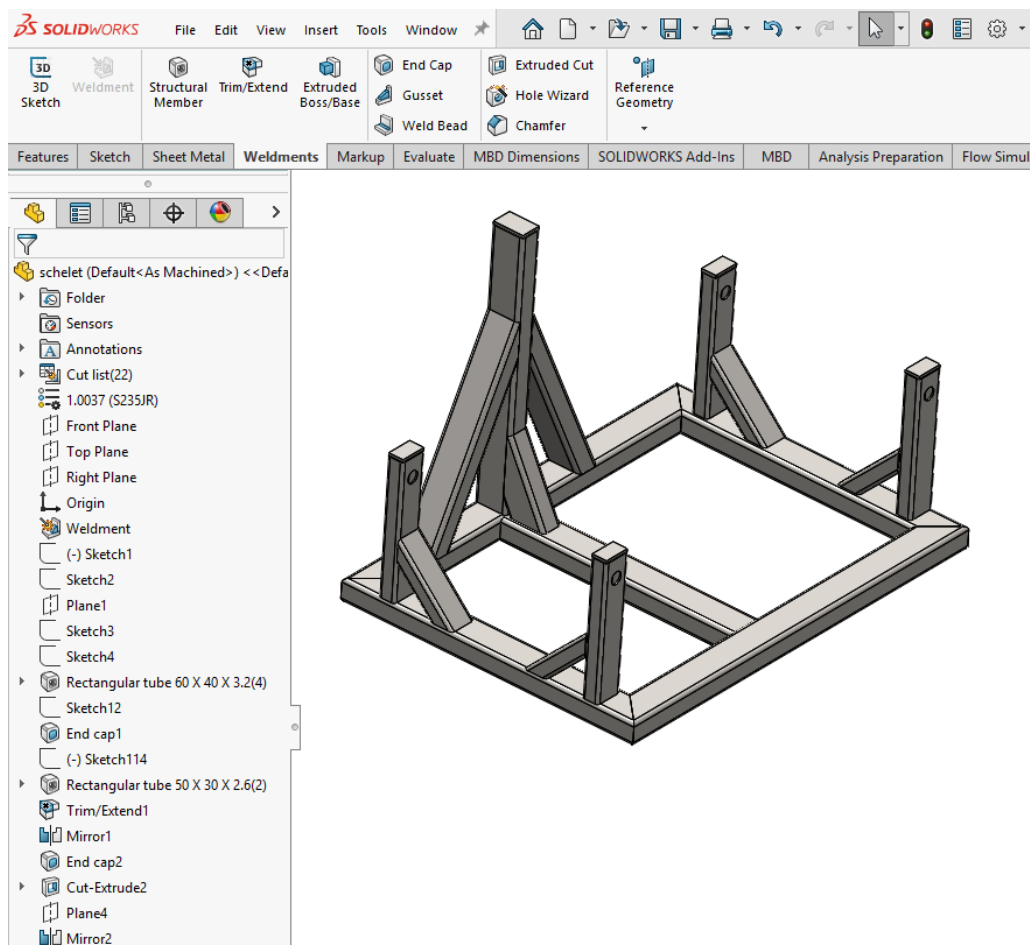


Figure 1.5 Welded Structure Example in SolidWorks

1.3.4.6. *The Simulation Module*

One of the fundamental challenges of product engineering is validating the behavior of a component before it is manufactured and subjected to physical testing. The answer to this challenge, traditionally provided only through prototyping and experimental testing, can today be obtained far more rapidly and economically through numerical simulation. The Simulation module in SolidWorks (Figure 1.6) provides precisely these tools, transforming the existing 3D model into a virtual testing environment.

Structural analysis using the Finite Element Method (FEA) is the central functionality of the module. By applying forces, pressures, or moments to the surfaces of the part and defining the support conditions, the software calculates the distribution of stresses and deformations throughout the entire volume of the component. Regions subject to excessive loading are visually highlighted through a color map, directing the designer toward the constructive modifications necessary to improve the mechanical performance of the part.

Beyond structural analysis, the module covers other types of loading conditions frequently encountered in the automotive industry. Thermal analysis allows the study of temperature distribution and heat flux in components exposed to thermal sources, such as parts in the vicinity of the engine or exhaust system. Fatigue analysis estimates the service life of a part subjected to cyclic loading, which is particularly relevant for suspension and transmission components. Fluid flow simulation, available through the complementary Flow Simulation module, extends this analysis environment to include the study of aerodynamic behavior and cooling system performance.

The practical value of the Simulation module lies in reducing the number of physical prototypes required during product development. Each design iteration can be numerically validated before being sent to manufacturing, shortening the development cycle and reducing the costs associated with late-stage redesigns.

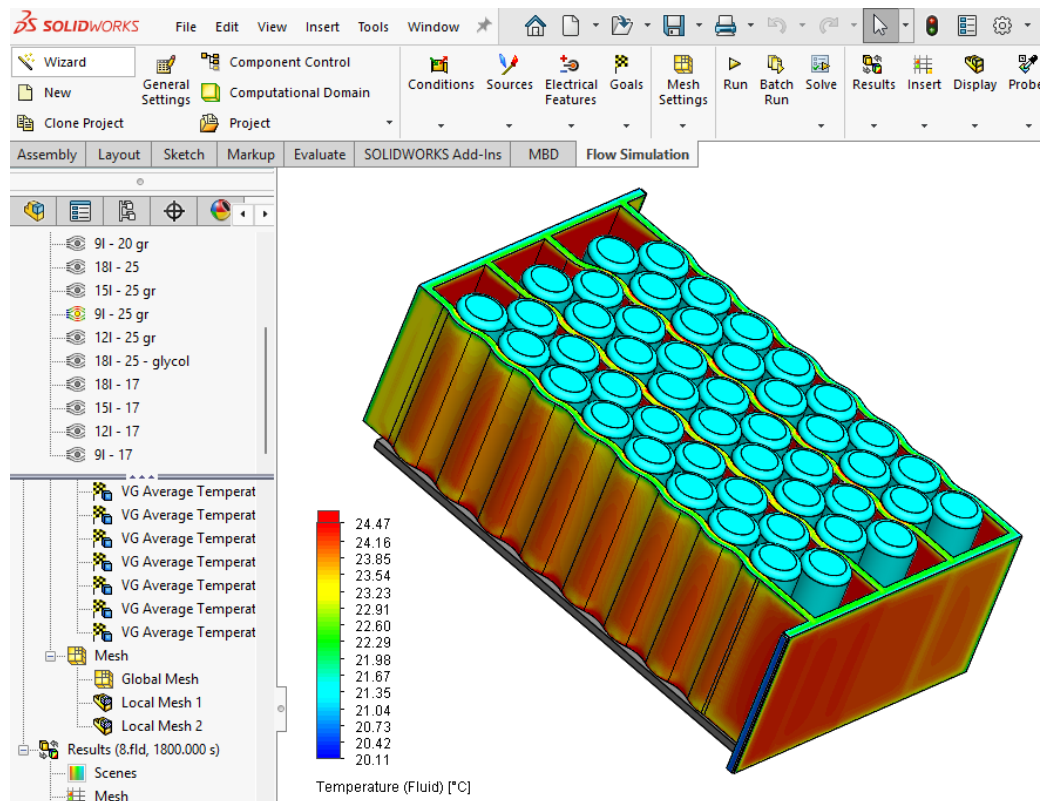


Figure 1.6 Simulation Example in SolidWorks

1.3.4.7. The Electrical Module

In the modern automotive industry, the complexity of electrical and electronic systems has grown exponentially over recent decades. A contemporary automobile contains kilometers of wiring, hundreds of connectors, and dozens of electronic control units, and managing this complexity within the same design environment as the mechanical components has become an increasingly pressing practical requirement. The Electrical module in SolidWorks (Figure 1.7) addresses precisely this need, providing dedicated tools for the design and integration of electrical systems directly within the 3D product model.

The workflow of the module covers the entire electrical design process, from the schematic diagram through to the physical routing of wiring harnesses within the three-dimensional space of the assembly. Wires, cables, and harness bundles are defined with their actual properties — cross-sectional area, conductor material, insulation type, and color — and are routed through the available space of the product, respecting minimum bend radii and avoiding interferences with surrounding mechanical components.

A significant advantage of integrating electrical design within the 3D CAD environment is the ability to verify in real time the compatibility between wiring routes and the mechanical geometry of the product. Modifications made to the mechanical structure are immediately reflected in the electrical routes, alerting the designer to potential conflicts before they become manufacturing issues. At the conclusion of the design process, the module automatically generates the documentation required for harness manufacturing, including wire lists, connector

tables, and wiring diagrams, significantly reducing the time allocated to technical documentation.

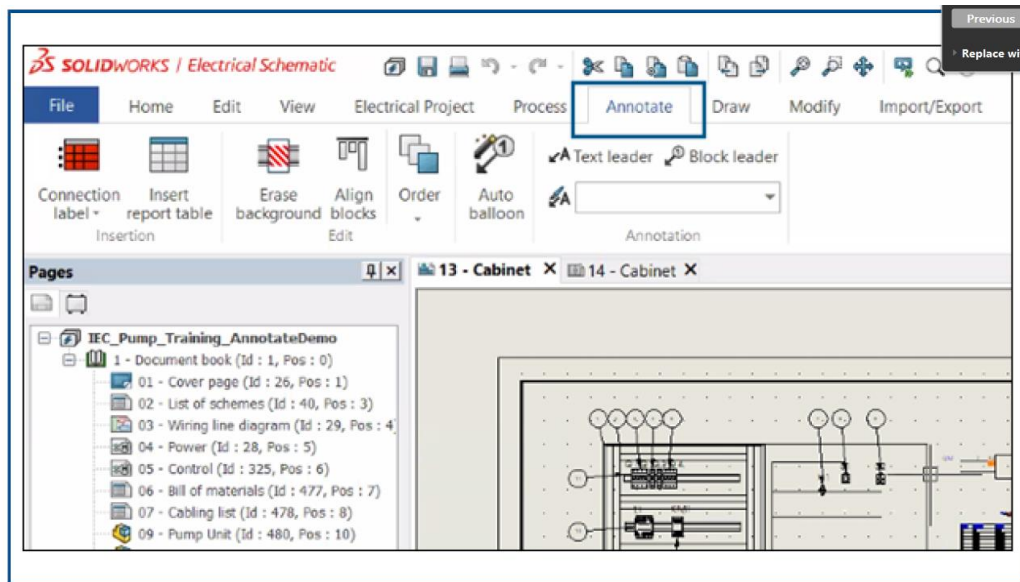


Figure 1.7 Electrical Schematic Module in SolidWorks

1.3.4.8. The Plastics Module

Plastic materials occupy an increasingly important role in the construction of modern vehicles, being present in components such as instrument panels, headlamp housings, body panels, interior brackets, and numerous other functional and aesthetic parts. The design of these components presents a specific challenge: the final geometry of the part is determined not only by functional requirements but also by the behavior of the molten plastic material during the injection molding process. The Plastics module in SolidWorks (Figure 1.8) addresses precisely this challenge, numerically simulating the entire injection molding process before the mold is manufactured.

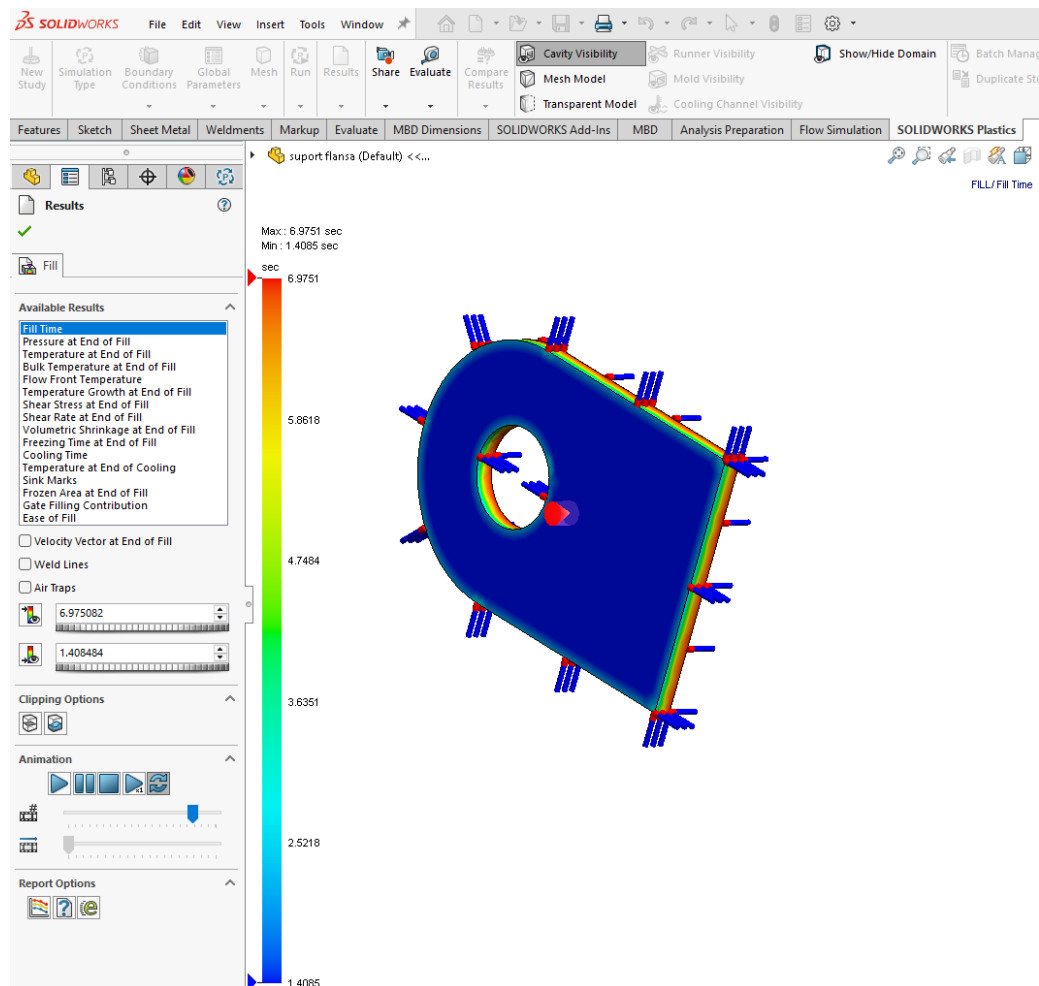


Figure 1.8 Plastic Injection Simulation Example in SolidWorks

The simulation covers the essential phases of the injection process: filling the mold cavity with molten plastic material, compacting the material under pressure, and progressive cooling through to solidification. Throughout these phases, the module calculates and visualizes the distribution of temperature, pressure, and material flow velocity, identifying regions where typical defects may occur, such as weld lines, sink marks, warping, or incomplete cavity fill.

The information obtained from the simulation allows the designer to optimize both the part geometry and the process parameters — including gate location, mold temperature, and pressure profile — prior to any investment in the manufacture of the physical mold. In the automotive industry, where injection molds for large components can represent investments in the range of tens or hundreds of thousands of euros, prior numerical validation through the Plastics module significantly reduces the financial risk associated with a non-optimized design and shortens the time to launch a new product on the production line.

1.3.5. The SolidWorks Working Environment

First contact with the SolidWorks interface may seem overwhelming, but its organization follows a clear and consistent logic that becomes familiar after the first few hours of use. The working environment is structured into distinct zones, each with a well-defined role in the design process, so that all necessary tools are quickly accessible without interrupting the workflow.

At the core of SolidWorks lies the principle of parametric modeling. Every dimension, every geometric relation, and every operation applied to a model is stored and remains editable throughout the entire design process. This means that a modification applied to a base parameter propagates automatically through all geometry dependent on it, eliminating the need to manually rework affected elements and significantly reducing the risk of error during design iterations.

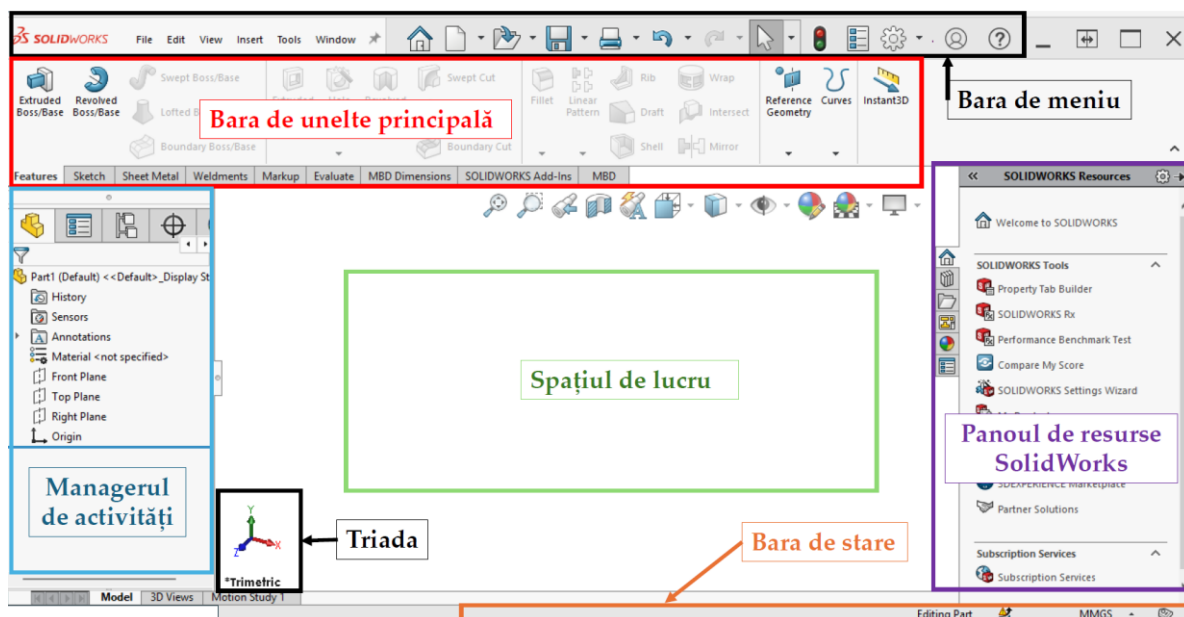


Figure 1.9 The SolidWorks Interface

The main interface, presented in Figure 1.9, is composed of several functional zones that work together. The menu bar provides access to all program commands, organized into logical categories such as **File**, **Edit**, **View**, **Insert**, and **Tools**. The main toolbar groups the most frequently used commands, automatically adapting to the active module – in the Part module, modeling tools are displayed, while in the Drawing module, dimensioning and annotation tools appear.

On the left side of the screen is the Activity Manager, also known as the **FeatureManager Design Tree**, highlighted in Figure 1.9. It displays the complete history of operations applied to the model in a tree structure that allows access to and editing of any previous operation. The coordinate triad in the workspace indicates the orientation of the coordinate system, providing a permanent visual reference of the model's position in three-dimensional space. The status bar, located at the bottom of the screen, displays contextual information about the current state of the model, the active units of measurement, and guidance messages useful during command execution. On the right side of the screen, the **SolidWorks Resources Panel** provides quick access to tutorials, documentation, and installation diagnostics tools.

1.3.5.1. The Menu Bar and Toolbars

The menu bar provides access to all program functions and is organized into clearly defined categories. The **File** menu manages document-related operations: creating, opening, saving, and exporting files. The **Edit** menu contains standard editing commands, such as undoing operations and selecting entities. The **View** menu controls the display mode of the model, the **Insert** menu is used for adding new operations and reference elements, and the **Tools** menu provides access to analysis tools and the general settings of the application (Figure 1.10).



Figure 1.10 The Menu Bar in SolidWorks

The main toolbar groups quick-access commands for file management, operation history navigation, and display control, and remains permanently visible regardless of the active module.

Below it sits the contextual toolbar, whose content automatically adapts according to the current working module. It is organized into tabs that can be activated with a simple click on their name. The main available tabs are as follows. The **Sketch** tab contains the tools for creating and editing two-dimensional sketches. The **Features** tab groups the commands for generating three-dimensional elements, such as extrusion, revolution, cut operations, and fillet operations. The **Sheet Metal** tab provides the tools specific to sheet metal component design, while the **Weldments** tab is dedicated to welded structures. The **Evaluate** tab provides tools for model verification and analysis.

The tabs visible in the toolbar depend on the type of document open and the modules installed. They can be customized by accessing the **Tools** menu, then **Customize**, where the user can activate or deactivate additional tabs, add new commands to existing toolbars, or create custom toolbars adapted to their specific workflow. This flexibility allows the configuration of an efficient working environment in which the most frequently used commands are always readily accessible (Figure 1.11).

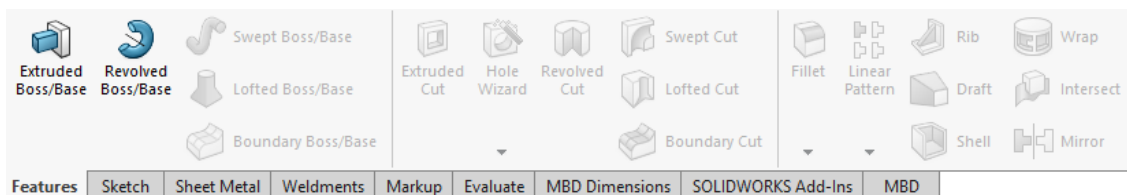


Figure 1.11 The Main Toolbar in SolidWorks

1.3.5.2. The FeatureManager Design Tree

The Activity Manager, located on the left side of the interface, provides an efficient means of organizing and navigating between the components of the project. It includes the history of completed operations, the assembly structure, and details about the properties of the model (Figure 1.12).

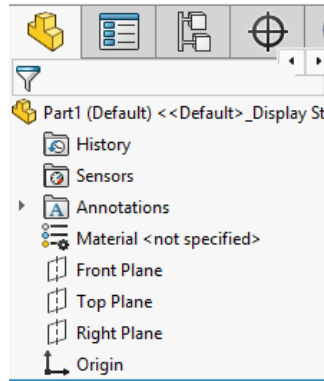


Figure 1.12 The Activity Manager in SolidWorks

1.3.5.3. The Graphics Area and Spatial Orientation

The graphics area represents the central space of the SolidWorks interface, the place where the model actually takes shape. Two-dimensional sketches are created here, three-dimensional parts are constructed, and components are assembled into complex assemblies. All modeling activity takes place within this space, and the quality of visualization and ease of navigation directly influence the efficiency of the design process.

A permanently present reference element in the graphics area is the coordinate triad, which indicates the orientation of the X, Y, and Z axis system relative to the current model (Figure 1.13). The Z axis, represented in blue, indicates the vertical direction of the workspace. The Y axis, represented in green, corresponds to the depth direction, while the X axis, represented in red, indicates the horizontal direction. This triad provides a permanent and essential visual reference, particularly when the model is rotated and the viewing perspective changes frequently during work.

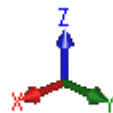


Figure 1.13 The Coordinate Triad in SolidWorks

Navigation in three-dimensional space is performed through the visualization commands grouped in the navigation bar of the graphics area (Figure 1.14). Each command in this bar has a specific and well-defined role in the interaction with the model.



Figure 1.14 The Graphics Area Navigation Bar in SolidWorks

Zoom to Fit automatically adjusts the zoom level so that the entire model is visible in the graphics area, regardless of its position or size. It is particularly useful after operations that significantly alter the dimensions of the model or after navigation in which the model has partially moved out of the field of view. **Zoom to Area** allows the manual definition of a rectangular region on the screen to which the zoom is applied, providing precise control over the portion of the model to be examined in detail. **Zoom In/Out** performs progressive approach or withdrawal from the model and is most quickly accessed by scrolling the mouse wheel.

Rotate View allows the free rotation of the model around any axis, enabling the examination of the geometry from any angle. In practice, this is most efficiently activated by pressing and holding the middle mouse button while moving the cursor in the desired direction. **Pan** translates the model within the plane of the screen without modifying the viewing angle or zoom level, and is useful for recentering the model in the graphics area without losing the current perspective. It is activated through the combination of the **Ctrl** key and the middle mouse button.

Section View generates a cross-sectional view of the model along a user-defined plane, allowing the examination of internal geometry without modifying the model itself. It is particularly useful for verifying wall thicknesses, internal bores, or the relative position of components within an assembly. **View Orientation** provides quick access to standard model orientations, such as front, back, lateral, top, and isometric views, allowing rapid repositioning of the viewing perspective with a single click. **Display Style** controls the visual representation mode of the model, offering options such as shaded display with materials, display with visible edges, wireframe display, and transparent display, each being useful in different contexts throughout the design process.

Mastering these navigation and visualization commands is one of the first practical skills to be consolidated when working with SolidWorks, as the ability to examine the model quickly and efficiently from any perspective directly conditions the quality and speed of the entire modeling process.

1.3.5.4. The Status Bar

The status bar, visible at the bottom of the SolidWorks interface (Figure 1.15), is a discreet yet extremely useful element in day-to-day design work. Although it occupies little screen space, the information it displays provides a constant and clear picture of the state of the active document and the current working context.

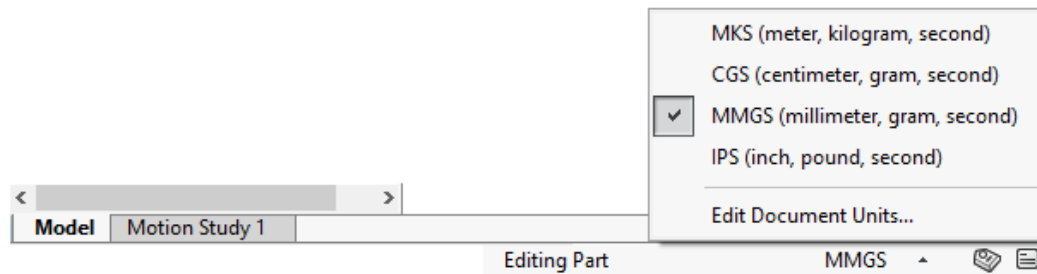


Figure 1.15 The Status Bar in SolidWorks

On the left side of the status bar are the navigation tabs between the available visualization types for the current document. The **Model** tab displays the three-dimensional model in the graphical workspace and represents the primary visualization used during the modeling process. The **3D Views** tab provides access to predefined or user-saved three-dimensional perspectives, useful for presenting and documenting the model. The **Motion Study** tab is dedicated to motion and animation studies, allowing the simulation of the kinematic behavior of assembled mechanisms.

In the central area of the status bar, the active working context is displayed. The **Editing Part** indication signals that the user is in part editing mode, which is particularly relevant when working within an assembly and editing an individual component from within it.

On the right side of the status bar, the active units of measurement of the current document are displayed. SolidWorks provides four predefined unit systems: **MKS** (meter, kilogram, second), used in large-scale engineering applications and applied physics calculations; **CGS** (centimeter, gram, second), specific to scientific fields and less commonly encountered in industrial mechanical design; **IPS** (inch, pound, second), standard in the American industry and in design according to ANSI standards; and **MMGS** (millimeter, gram, second), the recommended system for European mechanical engineering work.

Warning or error messages relating to incomplete operations, unsatisfied constraints, or geometric conflicts may also appear in this area, directing the user quickly to the source of the problem without the need for a manual search through the operation history.

1.3.5.5. The SolidWorks Resources Panel

The SolidWorks Resources Panel is located on the right side of the interface and represents the centralized access point for support tools, documentation, and services associated with the platform. Although it is often overlooked in the early stages of learning, this panel becomes a valuable resource as project complexity increases and situations arise that require additional documentation or problem diagnosis (Figure 1.16).

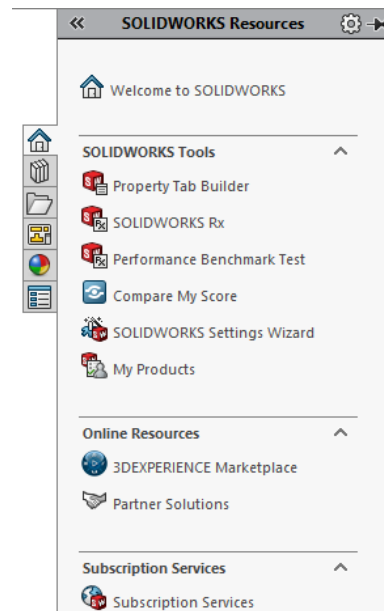


Figure 1.16 The SolidWorks Resources Panel

The **SOLIDWORKS Tools** section groups a series of utilities dedicated to the configuration and optimization of the working environment. **Property Tab Builder** allows the creation of custom forms for entering document properties, useful for managing part and assembly metadata within a large-scale project. **SOLIDWORKS Rx** is an installation diagnostics tool that checks hardware compatibility, identifies potential driver conflicts, and provides recommendations for optimizing the graphical performance of the application. **Performance Benchmark Test** evaluates the performance of the computer against SolidWorks requirements, providing a clear picture of the system's capacity to handle high-complexity models.

The **Online Resources** section connects the user directly to the digital ecosystem of the platform. Access to the **3DEXPERIENCE Marketplace** is available here the collaboration and 3D model exchange platform developed by Dassault Systèmes along with access to the network of partners and complementary solution providers.

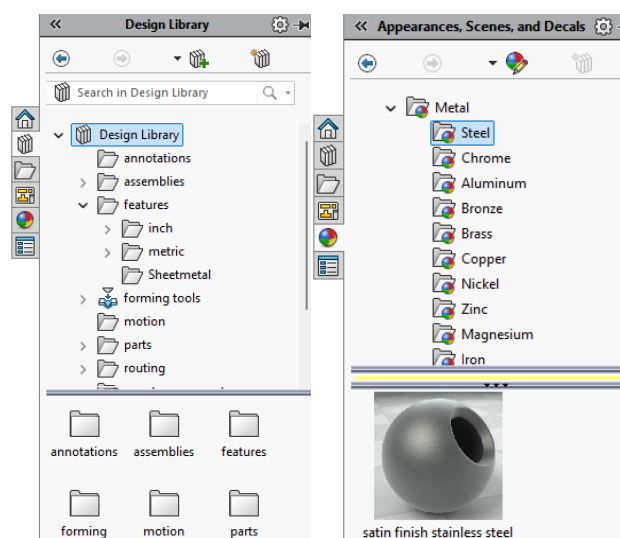


Figure 1.17 The Design Library and Appearance/Materials Panel in SolidWorks

The integrated tutorials, also accessible from this panel, represent a valuable starting point for consolidating knowledge of the platform's commands and workflows. They cover both basic operations and advanced functionality, are structured progressively, and are accompanied by practice models. Before beginning any modeling session, it is essential to correctly configure the units of measurement in **Document Properties** and to save the file initially in an organized directory simple practices that prevent data loss and ensure document consistency throughout the entire project.

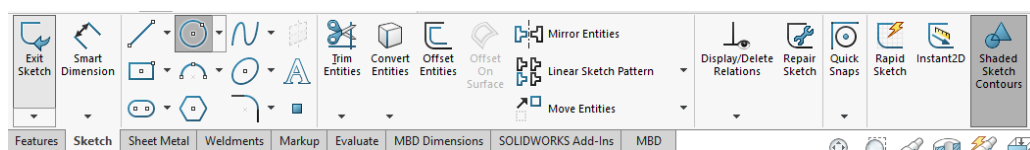
1.3.6. Fundamental Principles of Parametric Modeling

Parametric modeling represents the central philosophy upon which SolidWorks is built and which fundamentally distinguishes it from a simple three-dimensional drawing program. In a parametric model, geometry is not defined by fixed coordinates but by parameters and relations, meaning that the shape of the part is governed by rules rather than rigid values. This approach gives models remarkable flexibility: modifying a single parameter can automatically update dozens of dependent geometric elements without any additional manual intervention.

1.3.6.1. 2D Sketching and Geometric Relations

Every three-dimensional model created in SolidWorks starts from a two-dimensional sketch. The sketch is the basic profile of the part, drawn on a reference plane or on a flat face of an existing solid, and represents the geometry that will subsequently be extruded, revolved, or guided to generate the desired three-dimensional volume.

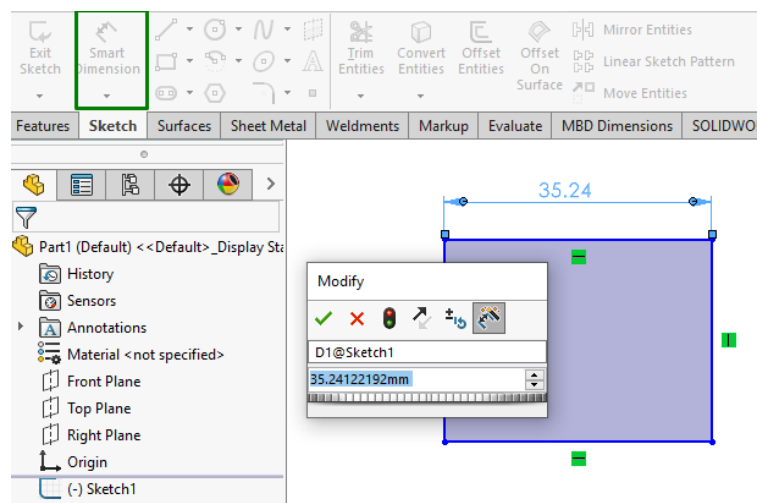
Creating a sketch begins with the selection of the working plane, which can be one of the three standard planes of space — **Front Plane**, **Top Plane**, or **Right Plane** — or any flat face of the existing model. After activating Sketch mode, the user has a series of drawing tools available: **Line** for straight segments, **Circle** for circles, **Rectangle** for rectangles, **Arc** for circular arcs, and **Spline** for free-form curves. These tools allow the rapid definition of any planar geometry, from simple profiles to complex contours with dozens of entities. **Centerline** is used in sketch mode to trace construction lines that do not form part of the actual geometry of the part but serve as auxiliary references. These are represented graphically as dashed lines and are ignored during extrusion or any other modeling operation. The primary uses of the **Centerline** command are the definition of axes of symmetry for applying the **Symmetric** relation, the tracing of the axis of rotation for the **Revolved Boss/Base** operation, and the creation of dimensioning references for diameters and angular positions. A construction line can also be obtained by converting an ordinary line, by checking the **For Construction** option in its PropertyManager.



A fundamental aspect of parametric sketching is the use of geometric constraints, which define the relationships between drawn entities independently of their numerical dimensions. A **Coincident** constraint requires two points to occupy the same position in space. **Parallel** keeps two lines permanently parallel regardless of subsequent modifications to the sketch. **Perpendicular** guarantees a right angle between two entities, **Tangent** ensures tangential continuity between a line and an arc, and **Concentric** keeps two circles or arcs on the same center. These relations are applied automatically by the program when the drawn geometry suggests such an intent, or they can be added manually by the user by selecting the desired entities. A fully defined sketch, in which all entities are governed either by geometric constraints or by dimensions, is represented by black lines and constitutes the basis of a robust and predictable parametric model.

1.3.6.2. Sketch Dimensioning

While geometric relations define the shape and relative position of entities within a sketch, dimensional dimensioning is what assigns them precise numerical values. In SolidWorks, dimensioning is performed through the **Smart Dimension** command, an intelligent tool that automatically recognizes the type of selected entity and proposes the corresponding dimension. Applied to a line segment, it generates a length dimension. Applied to a circle, it offers the option of dimensioning the diameter or the radius. Applied between two parallel lines, it measures the distance between them, and applied between two intersecting lines, it measures the angle between them.



Entered dimensions are not mere decorative labels but active parameters that govern the geometry of the sketch. Modifying the value of a dimension causes the immediate update of the associated geometry, preserving all previously defined relations and constraints. This property is essential in the iterative design process, in which the dimensions of a part are progressively adjusted as the design solution is refined.

1.3.6.3. Basic Operations: Extrude and Revolve

Extrusion and revolution are the most fundamental volume-generating operations in SolidWorks and constitute the starting point for modeling the majority of mechanical parts. A thorough understanding of these two operations and their operating logic is essential, as they appear in virtually every modeling sequence regardless of the complexity of the final part.

Extrude is the operation by which a two-dimensional sketch is projected in a direction perpendicular to its plane, generating a solid volume by adding the third dimension. The extrusion distance, referred to as depth, is entered numerically and can be applied in a single direction from the sketch plane or in both directions simultaneously through the **Mid Plane** option, which distributes the depth equally on either side of the profile. The termination condition of the extrusion can be defined in several ways: at a fixed distance through the **Blind** option, up to a selected surface through the **Up to Surface** option, or through the entire model through the **Through All** option. The **Extrude Cut** command uses the same mechanism but removes material rather than adding it, serving as the digital equivalent of a milling or drilling operation in a machining workshop.

Revolve generates volumes of revolution by rotating a two-dimensional profile around an axis of symmetry. The axis can be a line included in the sketch, traced as a construction line, or an edge of the existing model. The angle of rotation defaults to 360 degrees, generating a complete solid of revolution, but can be reduced to obtain partial sectors. This operation is the natural tool for modeling parts with axial symmetry, a category extremely common in manufacturing: transmission shafts, wheel spindles, pistons, valves, bearing rings, and numerous other powertrain and braking system components are routinely generated through **Revolve** operations. Similarly, the **Revolve Cut** command uses the same principle for material removal, allowing the creation of circular grooves or internal relief features.

1.3.6.4. Generation Operations: Sweep and Loft

Sweep and **Loft** are advanced modeling operations used when the geometry of a part exceeds the capabilities of the basic **Extrude** and **Revolve** operations. Both allow the generation of complex three-dimensional forms with variable cross-sections or curvilinear paths, and are frequently encountered in the design of geometrically elaborate components in the automotive industry.

Sweep generates a three-dimensional volume by guiding a two-dimensional profile along a defined path. Unlike **Extrude**, which projects the profile in a straight and fixed direction, **Sweep** follows any path, whether straight, two-dimensional curvilinear, or three-dimensional. The operation requires two mandatory elements: the profile, which defines the cross-section of the generated volume, and the path, which defines the trajectory the profile follows through space. Applications of this operation are numerous in the automotive field: exhaust system pipes, cooling hoses, wiring harnesses, bumpers, and door sealing profiles are typical examples of parts

generated through **Sweep** operations. The **Sweep Cut** command uses the same principle for material removal, allowing the creation of helical channels or splines with complex paths.

Loft generates a three-dimensional volume by interpolating between two or more two-dimensional profiles located on different planes. While **Sweep** maintains the same profile throughout the entire path, **Loft** allows the progressive modification of the cross-sectional shape from one profile to another, resulting in forms with geometry that varies along their length. The profiles can be of different shapes — a rectangle at one end and a circle at the other, for example — and SolidWorks automatically calculates and generates the transition between them. For more precise control over the generated shape, guide curves can be added to constrain the manner in which the interpolation is carried out between profiles. In the automotive industry, the **Loft** operation is used in the modeling of aerodynamic elements, fluid body panels, and engine intake components, where the cross-section varies continuously to optimize fluid flow.

Both **Sweep** and **Loft** can be used in either the material-adding or material-removing variant, providing complete flexibility in the construction or machining of any complex geometry.

1.3.6.5. *Editing Commands*

Editing operations represent the set of tools applied to an already generated solid with the purpose of refining, optimizing, or transforming the existing geometry. They cover a wide range of interventions, from simple edge rounding to complex deformation or geometry duplication operations.

Fillet rounds the edges of a solid by replacing them with curved surfaces of a specified radius. The radius can be constant, variable, or defined by a control curve, and the operation can be applied simultaneously to multiple edges. Beyond its aesthetic role, filleting reduces stress concentrations and improves the mechanical behavior of the part in service.

Chamfer creates chamfers at selected edges, replacing the sharp edge with a flat surface defined by a distance and an angle or by two distances. It is frequently used at shaft ends, thread entries, and on parts intended for automated assembly, where it facilitates the insertion and alignment of components.

Shell transforms a solid body into a thin-walled shell of uniform thickness by removing the interior material and retaining an outer envelope of specified thickness. Selected faces are removed, creating the necessary openings. The operation is essential in the design of housings, covers, and any shell-type components.

Draft applies a taper angle to the vertical faces of a solid relative to a reference direction known as the pull direction. This operation is mandatory for parts produced by casting or injection molding, as an insufficient draft angle makes it impossible to extract the part from the mold cavity without damaging the surfaces.

Rib generates stiffening ribs from an open sketch traced on a reference plane. Ribs are thin structural elements that reinforce part walls without adding significant mass, and are frequently used in the design of plastic parts or cast metal housings.

Pattern duplicates a geometric element according to a defined pattern. **Linear Pattern** distributes entities along one or two linear directions, **Circular Pattern** arranges them around an axis at equal angular intervals, and **Curve Driven Pattern** distributes them along an arbitrary curve. **Fill Pattern** automatically fills a surface with instances of the selected element, distributed uniformly according to a chosen geometric pattern such as diamonds, polygons, or squares.

Mirror creates a symmetrical copy of one or more operations relative to a selected plane of symmetry. It is used whenever the part exhibits symmetry about a plane, eliminating the need to repeat operations on the opposite side of the model and ensuring perfect geometric symmetry.

Wrap projects a two-dimensional sketch onto a curved surface of the model, either by engraving or by adding material. It is used for inscribing text or symbols on cylindrical or conical surfaces, such as markings on shafts or embossed labels on plastic housings.

Dome deforms a flat or curved face of the model, creating a spherical or ellipsoidal protrusion of specified height. It is used in the design of covers, tanks, or components that require domed shapes for additional rigidity or aesthetic purposes.

Indent creates an imprint of one component in the surface of another, useful in the design of gaskets, sealing parts, or any situation in which two surfaces must conform precisely to one another.

Flex globally deforms a solid model through bending, twisting, tapering, or stretching operations, applicable to the entire body or to a selected portion. It is used for rapidly adjusting the overall shape of a part without modifying the underlying sketches and base operations.

Scale modifies the overall dimensions of a model by a uniform scaling factor or by differentiated factors along each axis. It is frequently used in mold design, where the cavity dimensions must be larger than the final part to compensate for material shrinkage upon cooling.

Intersect generates a new solid from the common volume of two or more overlapping bodies, or performs complex Boolean operations between multiple bodies. It is useful in the design of parts whose shapes are defined by the intersection of simpler geometries.

Split divides a solid into two or more distinct bodies using a separating plane or surface. It is used in the design of multi-segment parts or in preparing models for analysis on separate components.

Move/Copy Body translates or copies a solid body within the same document through translation, rotation, or by defining constraints relative to existing geometry. It is useful in managing multi-body models and in rapidly creating positional variants of the same element.

Delete Body permanently removes a solid body from the document, useful for cleaning up intermediate bodies that served as references or tools in the construction of the final geometry but do not form part of the finished part.

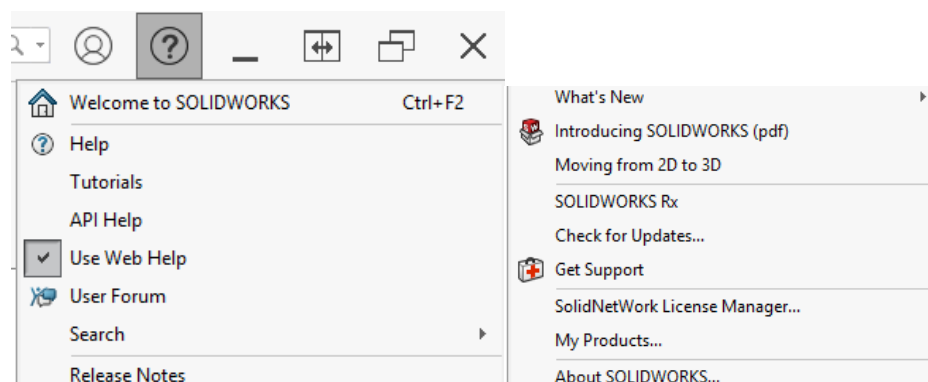
1.3.7. Additional Learning Resources

Mastering design software such as SolidWorks is a continuous process that does not end once the basic concepts have been covered. The complexity of the platform and the diversity of its application domains constantly require the discovery of new techniques, the optimization of workflows, and the resolution of domain-specific problems. Fortunately, the resource ecosystem available around SolidWorks is one of the richest in the CAD industry.

The first point of reference is the tutorials integrated directly into the application, accessible through the **Help** menu. These cover the fundamental operations of each module, from the first sketch to the generation of a complete technical drawing, and are structured progressively, guiding the user step by step through each command. The advantage of these tutorials is that they are synchronized with the installed version of the program and accurately reflect the interface and options available in the current working environment.

The official SolidWorks documentation, available both within the application and online at **help.solidworks.com**, represents the complete technical reference for the platform. Every command, every option, and every parameter is described in detail, accompanied by illustrated examples and notes on version compatibility. Consulting the official documentation is the recommended practice whenever the behavior of a command is not fully understood or when exploring the advanced options of a given feature.

The global SolidWorks user community represents a resource of exceptional practical value. The official **MySolidWorks** forum, user groups, and dedicated platforms such as the **SolidWorks Forum** bring together professionals and enthusiasts from around the world, who share solutions to concrete problems, modeling techniques, practice files, and tips from industrial practice. A modeling problem that appears unsolvable frequently finds a detailed answer within these communities, posted by someone who has encountered a similar situation. Dedicated channels on platforms such as YouTube complement this offering with video demonstrations of modeling techniques, freely accessible and organized by difficulty level.



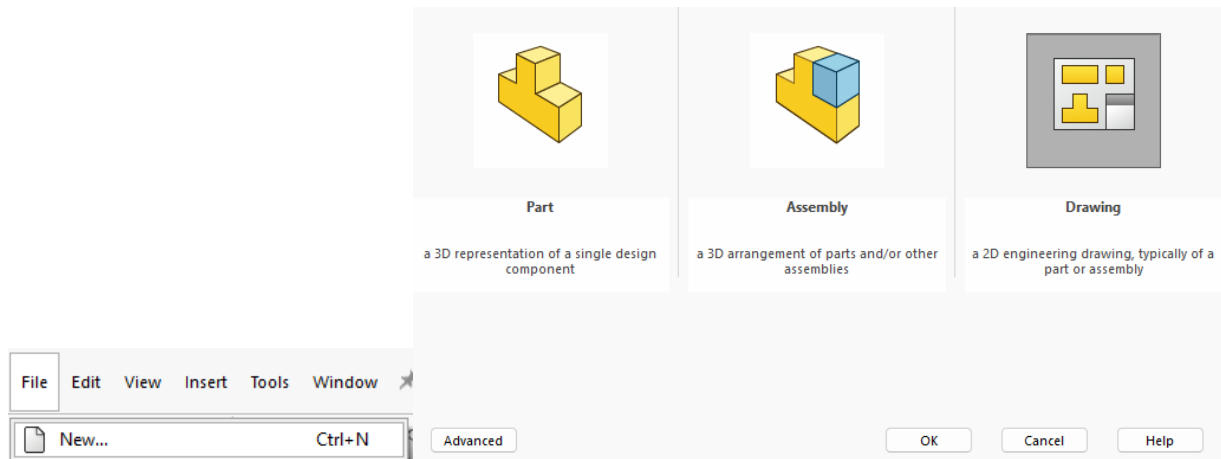
1.3.8. Practical Application

1.3.8.1. Modeling a Three-Dimensional Rectangular Block

By following these steps, a beginner user will be able to model a simple three-dimensional body (rectangular block) using basic commands such as **Rectangle**, **Smart Dimension**, and **Extruded Boss/Base**. The process introduces essential concepts such as sketching on 2D planes, adding dimensions, and transforming a planar profile into a 3D model.

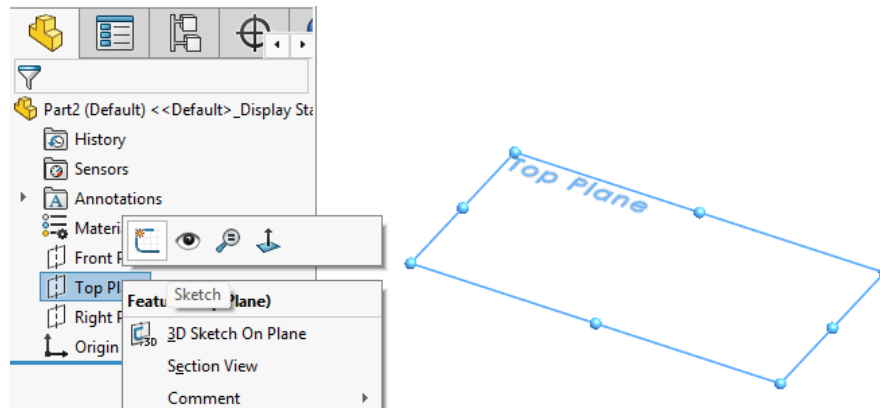
1. Creating a New Document

To initiate the modeling process, the SolidWorks application is launched and the **File** → **New** option is selected from the main menu, or the keyboard shortcut **Ctrl + N** is used. From the **New Document** window, the **Part** file type is selected, which is used for modeling individual parts. The selection is confirmed by clicking the **OK** button, opening the working interface.



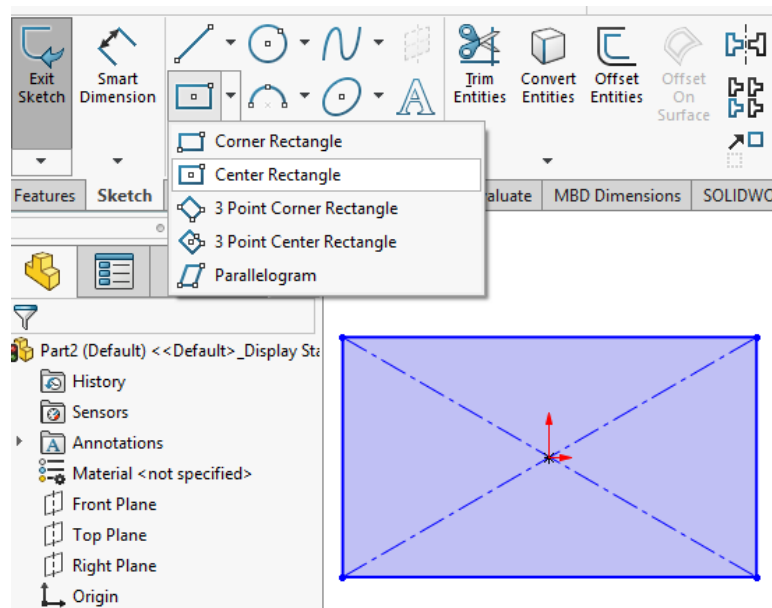
2. Accessing the Sketch Module

The next step involves activating the Sketch module. This is done by selecting the **Sketch** tab from the main toolbar. The user then selects one of the available planes — **Front Plane**, **Top Plane**, or **Right Plane**. These planes, visible in the Activity Manager, establish the base for drawing the sketch. After selecting the plane, the interface automatically switches to Sketch mode, preparing the workspace for geometry definition.



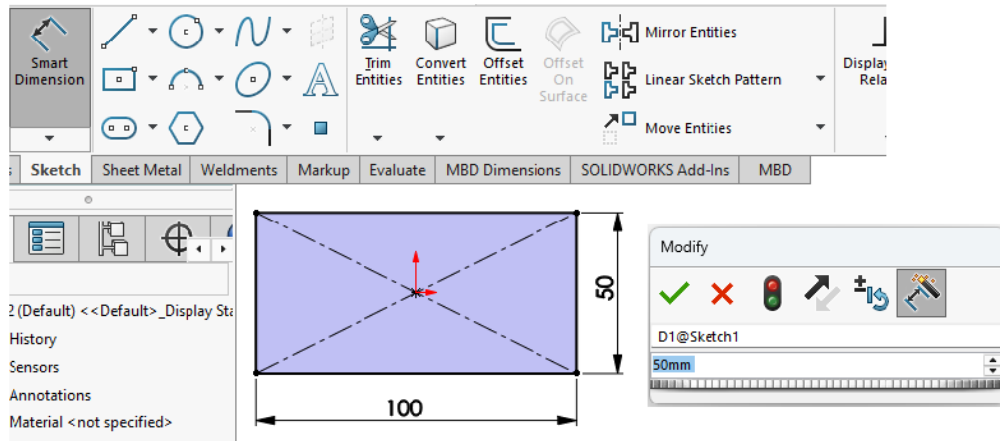
3. Creating the Geometry

To define the 2D geometry, the **Rectangle** command is accessed from the **Sketch** toolbar and the **Corner Rectangle** option is selected. The drawing process begins by placing the first corner of the rectangle at the **Origin** point of the coordinate system, indicated graphically by an orange point. The shape is extended by moving the cursor, and the second corner is fixed with an additional click.



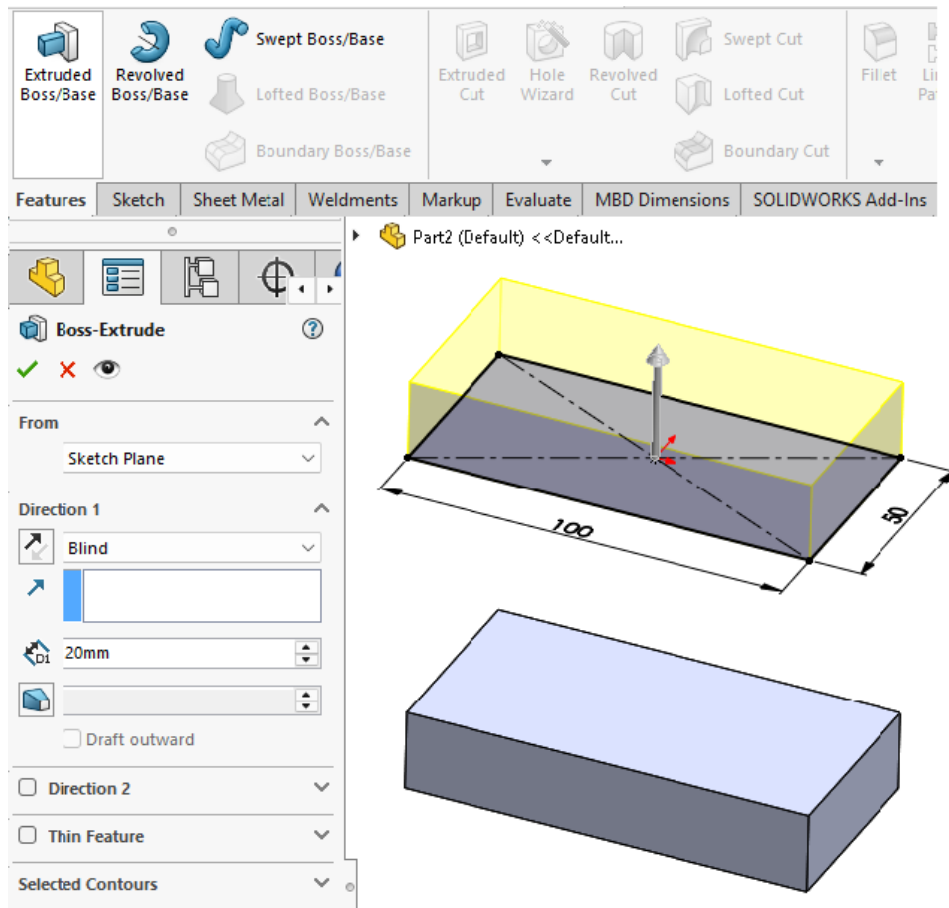
4. Adding Dimensions

To define the proportions of the geometry, the **Smart Dimension** command is used, located in the **Sketch** toolbar. The dimensioning process involves selecting a side of the rectangle and entering the corresponding value (for example, 100 mm for the long side and 50 mm for the short side). These steps are repeated for both sides, and upon completion, the defined geometry is highlighted in black lines, indicating that the dimensions have been correctly applied.



5. Extruding the Sketch

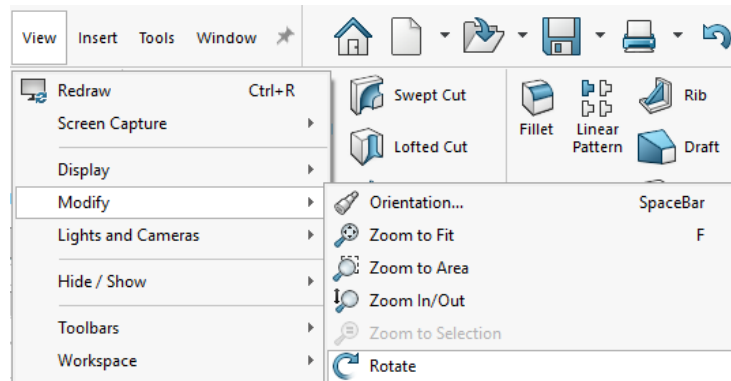
Once the 2D sketch is defined, it is used to generate the three-dimensional body. From the **Features** tab, the **Extruded Boss/Base** command is activated. The created sketch is selected, and the extrusion height is entered in the configuration window (for example, 20 mm). The operation is confirmed by clicking the **OK** button, resulting in a three-dimensional model visible in the graphics area.



6. Viewing the Model

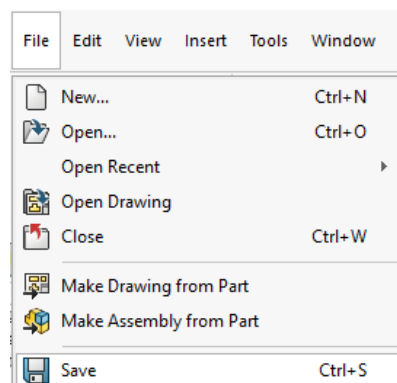
For a detailed inspection of the generated model, the user may employ the following 3D navigation commands:

- **Rotate** allows the model to be explored from various angles.
- **Zoom to Fit** automatically adjusts the display so that the entire model is visible on screen.



7. Saving the Model

To preserve the progress made, the file is saved by accessing the **File** → **Save** option or by using the keyboard shortcut **Ctrl + S**. The save location is specified and the model is given a descriptive name (for example, RectangularBlock). The saving process is completed by confirming the **Save** command.



1.3.8.2. Cylinder Modeling

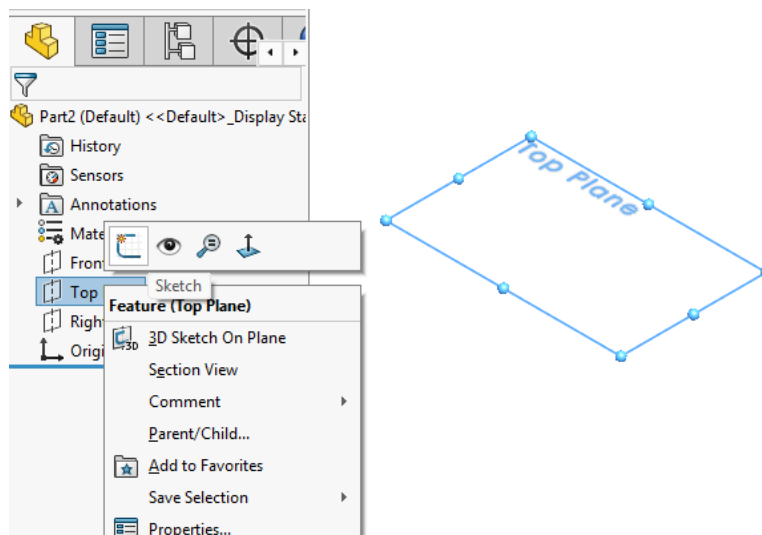
Cylinder modeling is performed using the fundamental SolidWorks commands **Circle**, **Smart Dimension**, and **Extruded Boss/Base**. The steps are designed to ensure a methodical understanding of the modeling process and are suitable for didactic use.

1. Creating a New Document

Cylinder modeling begins by launching the SolidWorks application. From the main menu, the user selects **File** → **New** or uses the keyboard shortcut **Ctrl + N**. In the **New Document** window, the **Part** file type is selected, which is specific to the creation of an individual part. Once this option is chosen, the process is confirmed by clicking the **OK** button, opening the main working interface.

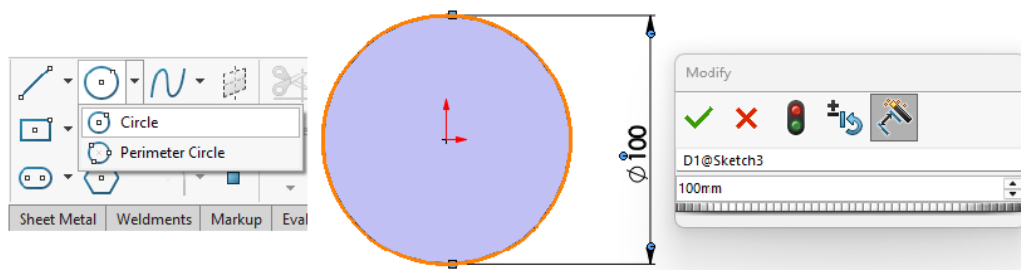
2. Selecting the Working Plane and Accessing the Sketch Module

To begin creating the sketch, the **Sketch** tab is selected from the main toolbar. From the Activity Manager on the left side of the screen, the **Top Plane** is chosen as the working plane, which will serve as the base of the cylinder. Upon selecting the plane, SolidWorks automatically enters Sketch mode, enabling the definition of the part geometry.



3. Defining the Circular Base Profile

To define the base of the cylinder, the **Circle** command is accessed from the **Sketch** toolbar, located in the upper right corner of the working environment.

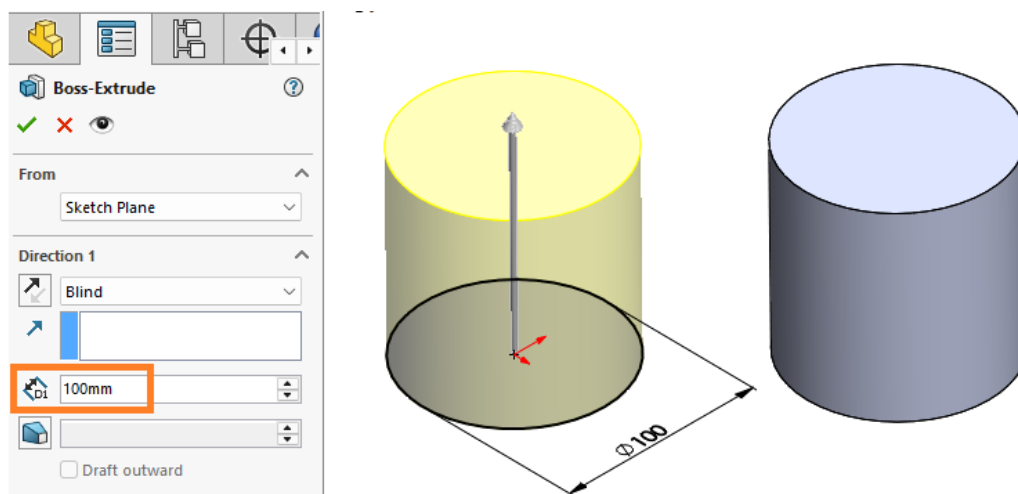


The center of the circle is placed at the **Origin**, indicated graphically by an orange point. The radius is defined by moving the cursor outward and clicking on the circumference. The

diameter is then precisely set using the **Smart Dimension** command: the circumference is selected and the desired value is entered in the dimension dialog box (for example, 100 mm). Once the dimension is applied, the circle becomes fully defined and is displayed in black.

4. Transforming the Sketch into a Three-Dimensional Body

Once the sketch is fully defined, Sketch mode is exited by selecting the **Exit Sketch** command. To transform the 2D sketch into a three-dimensional solid, the **Features** tab is accessed and the **Extruded Boss/Base** command is activated. The previously created circle is selected as the base sketch, and the desired cylinder height is entered in the configuration window (for example, 100 mm). The extrusion direction can be adjusted as required, and the operation is confirmed by clicking the **OK** button. The three-dimensional model of the cylinder is thus generated and displayed in the graphics area.

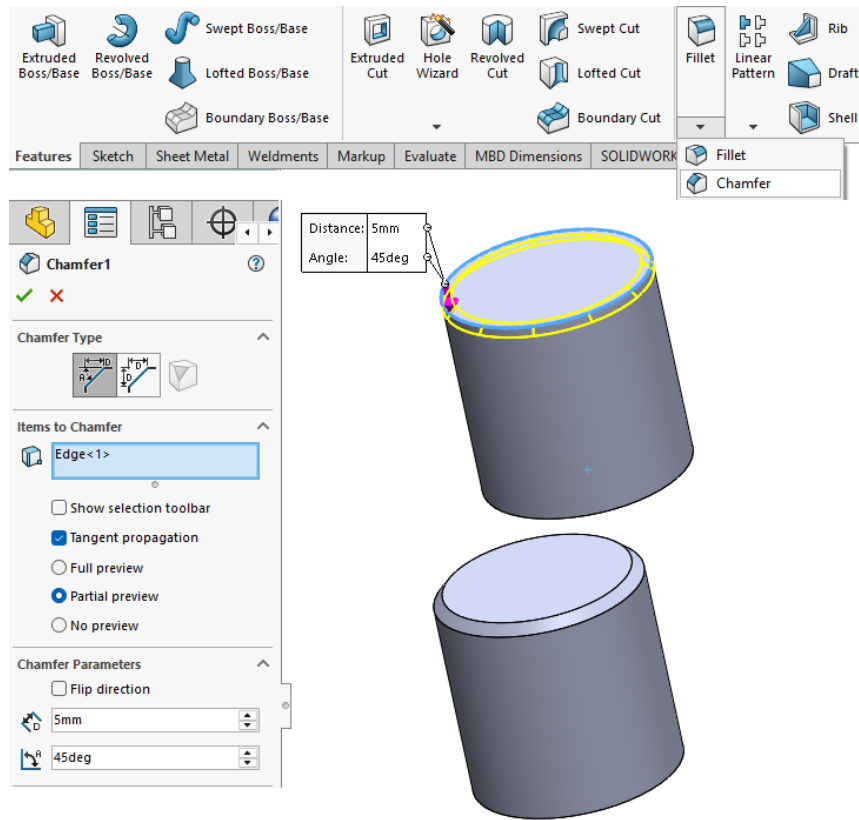


5. Model Visualization and Exploration

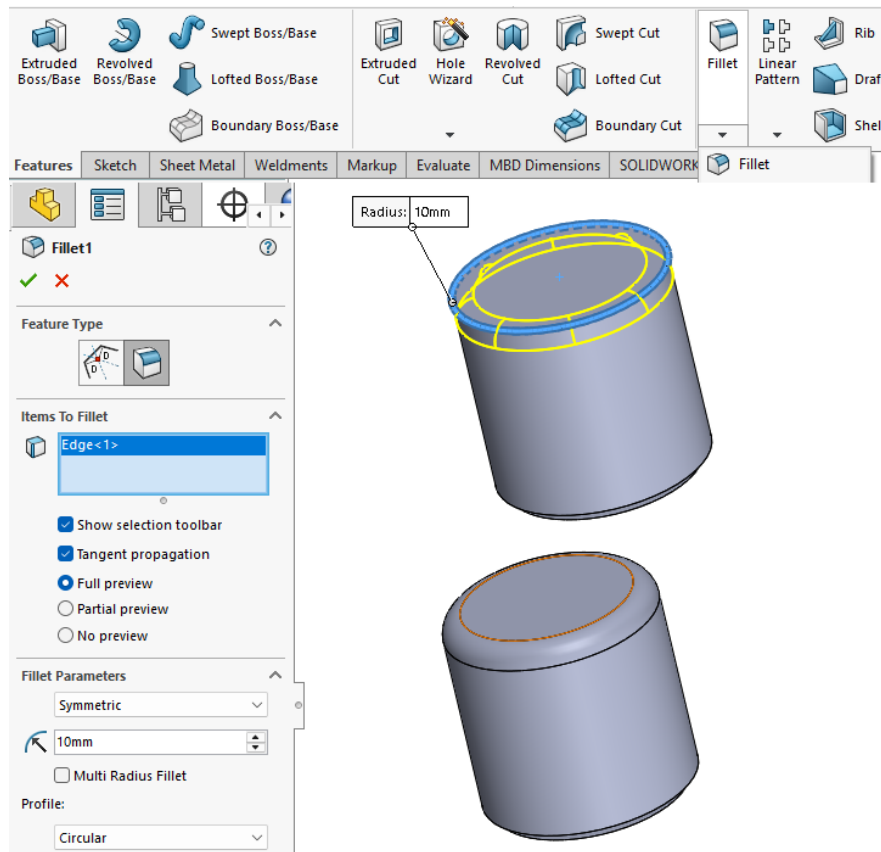
The cylindrical model can be explored using the 3D navigation commands available in SolidWorks. **Rotate View** allows the model to be rotated for examination from different perspectives. **Zoom to Fit** adjusts the display so that the entire cylinder is visible on screen. **Pan View** facilitates the translation of the model within the graphics window for more detailed inspection. Additionally, the **Ctrl + scroll wheel** combination can be used to quickly adjust the zoom level within the workspace.

6. Applying a Chamfer at the Cylinder Ends

To apply an additional finishing feature, a chamfer is added at the ends of the cylinder. From the **Features** tab, the **Chamfer** command is selected, which allows the creation of an angled edge at the boundary of a face. In the **Chamfer** configuration window, the upper face of the cylinder is selected and the required chamfer parameters are entered (for example, 5 mm width at a 45° angle). The operation is confirmed by clicking the **OK** button.

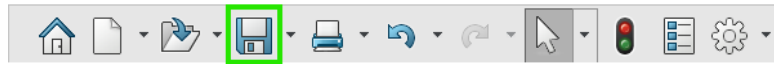


The procedure for the **Fillet** command follows the same logic. Edge rounding is performed by selecting the desired faces or edges, specifying the fillet radius in the configuration window (for example, 10 mm), and confirming the operation by clicking **OK**.



7. Saving the Created Model

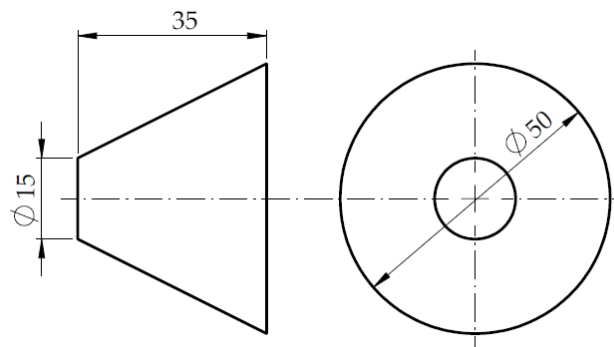
To preserve the generated model, the **File** → **Save** option is selected or the keyboard shortcut **Ctrl + S** is used. In the save dialog window, the file location is specified and a descriptive name is assigned to the model, such as Cylinder. Once these details are confirmed by clicking the **Save** button, the saving process is complete and the model is stored in SolidWorks format.



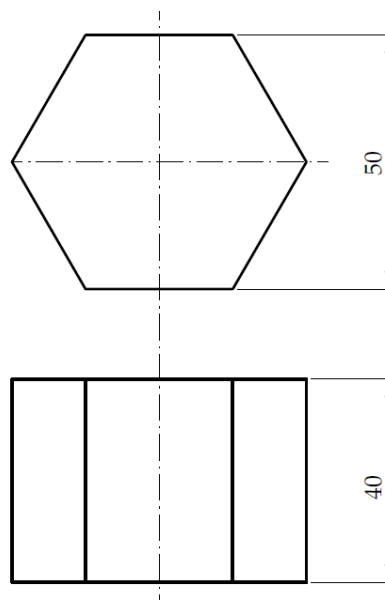
1.3.9. Independent Practice Exercises

The following exercises are proposed for independent completion and feature parts of simple geometry, accessible upon completion of the practical applications presented earlier in this chapter. Each proposed part consolidates one or more of the operations studied and can be completed by combining the sketching, dimensioning, and modeling techniques introduced throughout this chapter.

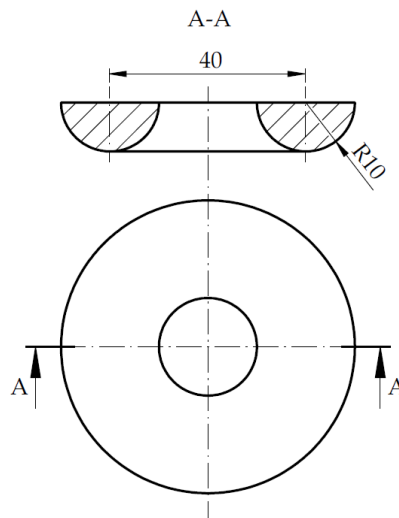
1.3.9.1. Modeling a Truncated Cone



1.3.9.2. Modeling a Hexagonal Prism



1.3.9.3. Washer with Spherical Relief



1.3.10. Assessment Questions

1. *What do the terms CAD, CAM, and CAE represent?*

2. *What is the role of parametric modeling in engineering design?*

3. *Describe the difference between the Fillet and Chamfer operations and provide a specific use case for each in the automotive industry.*

4. *Which modules of the SolidWorks platform are involved in the development of a technical product?*

2. PARAMETRIC PART MODELING IN SOLIDWORKS

2.1. Chapter Objectives

This chapter aims to deepen the parametric modeling of parts within the Part environment, through the analysis and completion of components specific to mechanical engineering. Emphasis is placed on selecting an appropriate modeling strategy, on the use of constructive features, and on understanding the relationship between the geometry of a part, its function, and the sequence of modeling operations.

2.2. Learning Outcomes

Upon completion of this chapter, the student will be able to:

- analyze a part from both a geometric and functional perspective;
- establish a coherent parametric modeling strategy;
- use advanced features to create geometries specific to mechanical parts;
- correctly apply tools such as **Hole Wizard**, **Helix/Spiral**, and **Sweep**;
- verify and optimize a three-dimensional model with respect to design requirements.

2.3. Theoretical and Applied Content

2.3.1. General Considerations on Advanced Part Modeling

In engineering practice, real parts rarely present simple geometries. A pump body, a threaded shaft, a helical spring, or a clamping bracket all require a more elaborate approach, both in terms of the commands used and the modeling strategy adopted.

Before beginning any modeling session, the student should analyze the part and consider several key questions: what is the basic shape, which operations need to be added and in what order, and which elements can be achieved through symmetry or repetition. This prior analysis makes the difference between a robust model that can be easily modified at a later stage and one that fails at the first dimensional change.

SolidWorks provides several advanced tools that extend modeling capabilities. The **Hole Wizard** command allows the rapid creation of standardized, threaded, or tapered holes without requiring the student to memorize the dimensions imposed by standards, as the program manages these automatically. The **Helix/Spiral** command generates helical paths commonly used in the modeling of threads and springs.

Regardless of the complexity of the part, modeling in SolidWorks remains an orderly and logical process in which each step builds upon the previous one, and the correct selection of the operation sequence constitutes the foundation of a well-constructed parametric model.

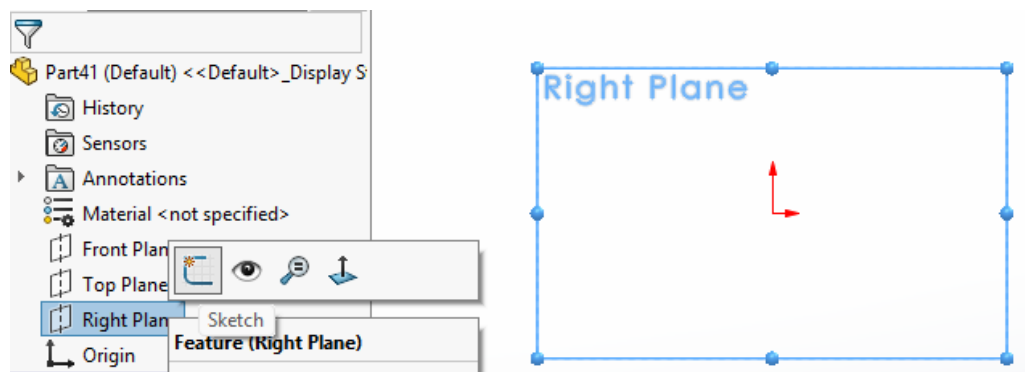
2.3.2. Modeling the Housing of an INA Bearing

1. Creating a New Document

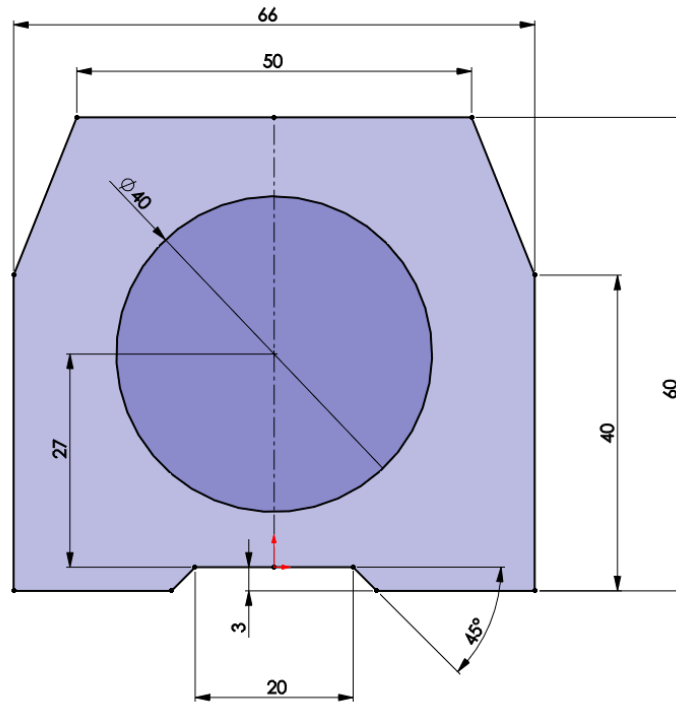
The first step in the part modeling process consists of opening the SolidWorks application and selecting **File** → **New** → **Part** to create a new part file.

2. Creating the Base Sketch

Once the document is open, the **Right Plane** is selected and a new sketch is created.

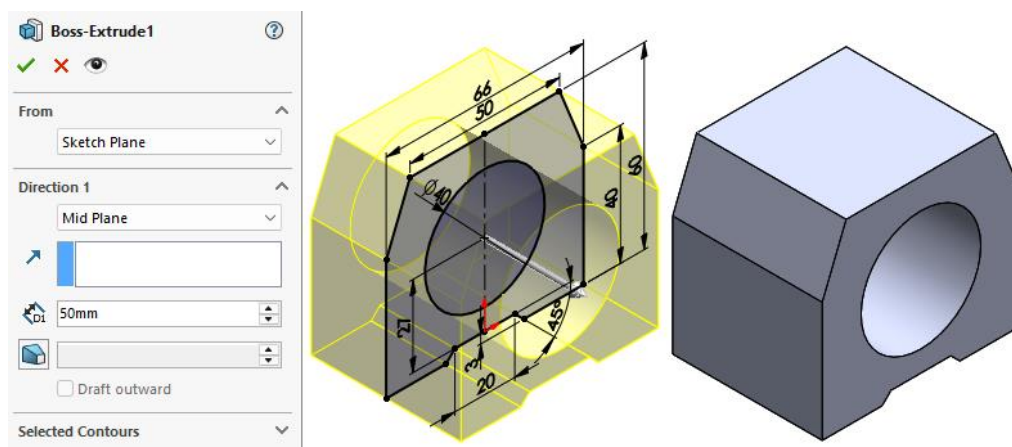


Within the selected plane, the base sketch of the part is drawn using the **Line** command, as illustrated below. The sketch is constructed starting from a vertical construction axis traced through the origin, which serves as the symmetry reference for all entities. A centered rectangle is drawn using **Center Rectangle**, dimensioned at 66 mm in width and 60 mm in height, after which the upper corners are symmetrically chamfered using **Sketch Chamfer**, with the **Symmetric** relation applied relative to the axis. The lower notch, 20 mm wide and 3 mm deep, is also defined using the **Symmetric** relation, and the 45° angle of its corners is dimensioned using an angular dimension. The inner circle with a diameter of 40 mm is placed on the axis at 27 mm from the origin, with its center constrained by the **On Line** relation. The sketch is fully defined when all entities are displayed in black.



3. Extruding the Profile

Once the base sketch is complete, it is extruded using the **Extruded Boss/Base** command. Within this command, **Mid Plane** is selected as the extrusion direction, a depth of 50 mm is entered, and the operation is confirmed by clicking the **OK** button to obtain the desired volume.

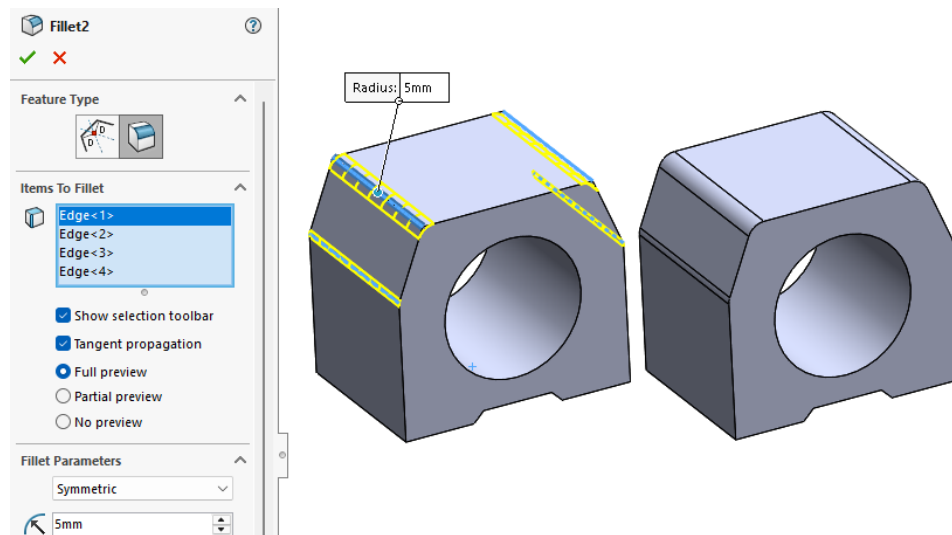


4. Rotating the Model

To facilitate the selection of the lower edges, the model is reoriented by pressing and holding the middle mouse button and rotating until a convenient perspective is obtained, or by selecting a predefined orientation from the **View Orientation** command in the **View** toolbar.

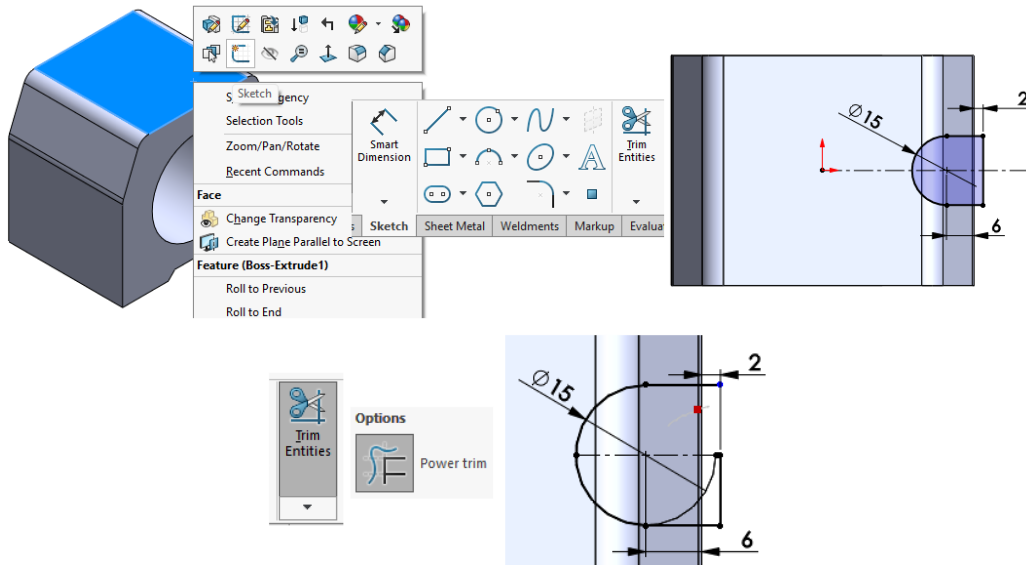
5. Rounding the Upper Edges

Edge rounding is performed using the **Fillet** command, accessible from the **Features** toolbar. In the command's **PropertyManager**, the four upper edges of the part are selected in the **Items to Fillet** field, and a radius of 5 mm is entered in the **Radius** field. SolidWorks previews the result in real time prior to confirmation, allowing the student to verify the correctness of the selection. The operation is completed by clicking the **OK** button.



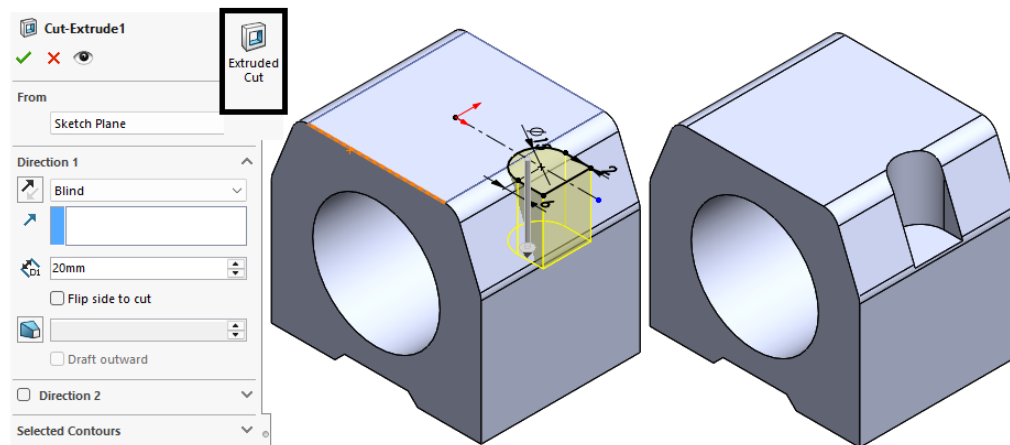
6. New Sketch on the Upper Surface

A new sketch is inserted on the upper surface of the part by right-clicking and selecting the **Sketch** option. From the **Sketch** toolbar, the **Circle** command is selected, and the center position and radius of the circle are defined. The circle is placed on the surface of the part with a diameter of 15 mm, positioned at a distance of 6 mm from the lateral edge using the **Smart Dimension** tool. A **Centerline** is then drawn to mark the center axis of the part, accessible from the **Sketch** toolbar. To complete the sketch, the **Line** command is used to draw two segments tangent to the circle and a closing line located 2 mm from the lateral edge. The segments lying inside the sketch contour are subsequently removed using the **Trim Entities** command with the **Power Trim** function activated. This function allows the rapid removal of multiple segments in a single operation: the left mouse button is held down and the cursor is dragged across the segments to be removed, which are deleted as the cursor intersects them.



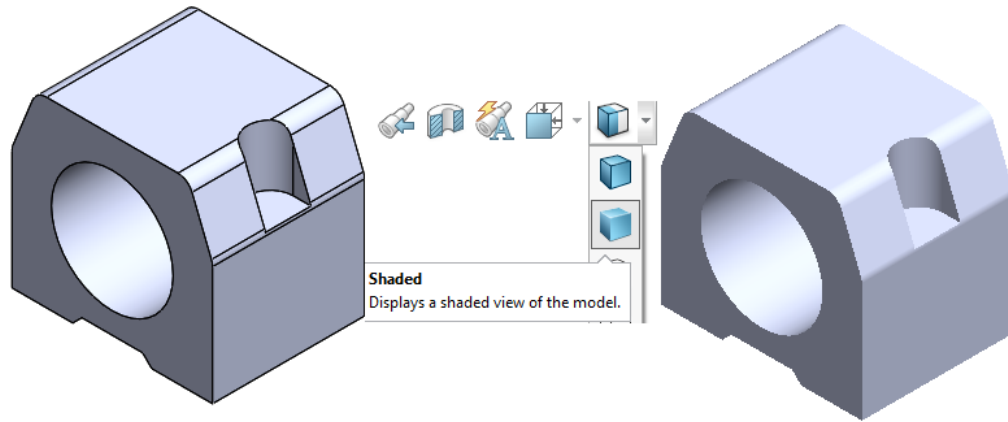
7. Extrude Cut

From the **Features** toolbar, the **Extruded Cut** command is selected, and a cutting depth of 20 mm is defined in the settings panel. This operation removes material from the part to the specified depth of 20 mm in accordance with the configured parameters, ensuring the precision and control required in the 3D modeling process and allowing accurate cuts tailored to the project requirements.



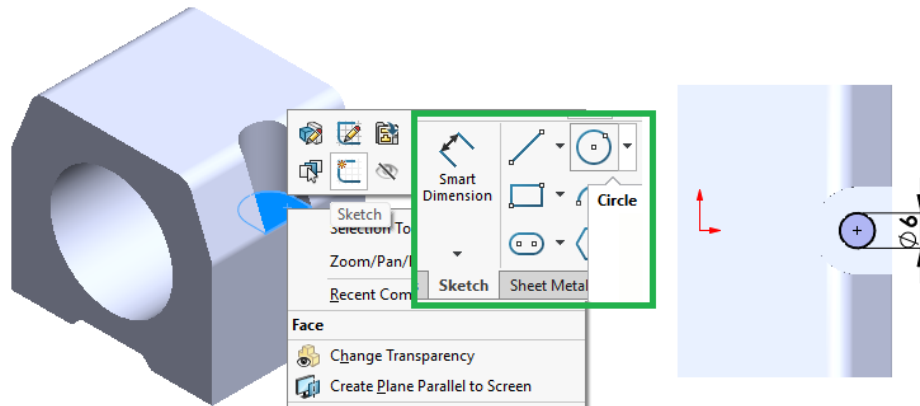
8. Hiding Tangent Edges After Applying the Fillet Command

After applying the **Fillet** command, the tangent edges automatically generated by the operation can be hidden to improve the visual clarity of the model. This is achieved by accessing the display options in SolidWorks. The **View** button is selected from the toolbar, and the **Hide/Show Edges** option is navigated to. Here, the **Shaded** option can be selected to hide the tangent edges, resulting in a cleaner and clearer representation of the part.



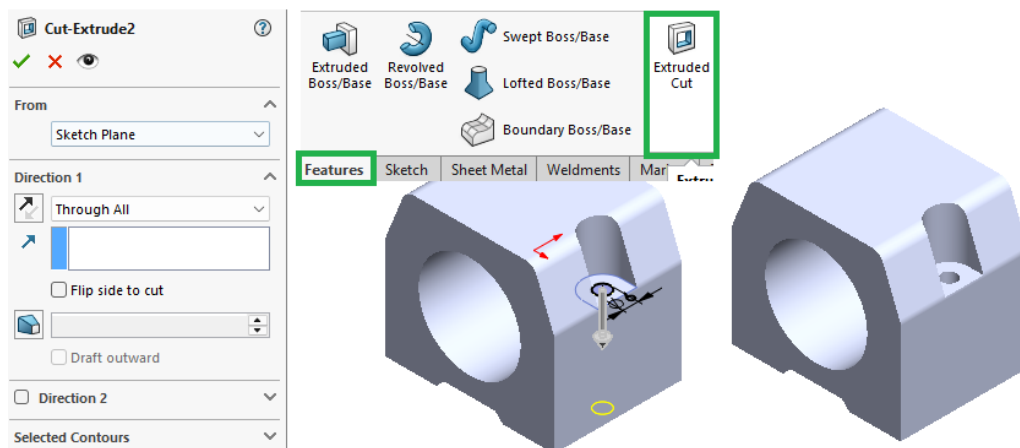
9. Through Hole

A new sketch is created on the surface resulting from the cut operation. Using the **Circle** command from the **Sketch** toolbar, a concentric circle with a diameter of 6 mm is drawn at the center of the existing 15 mm diameter circle.



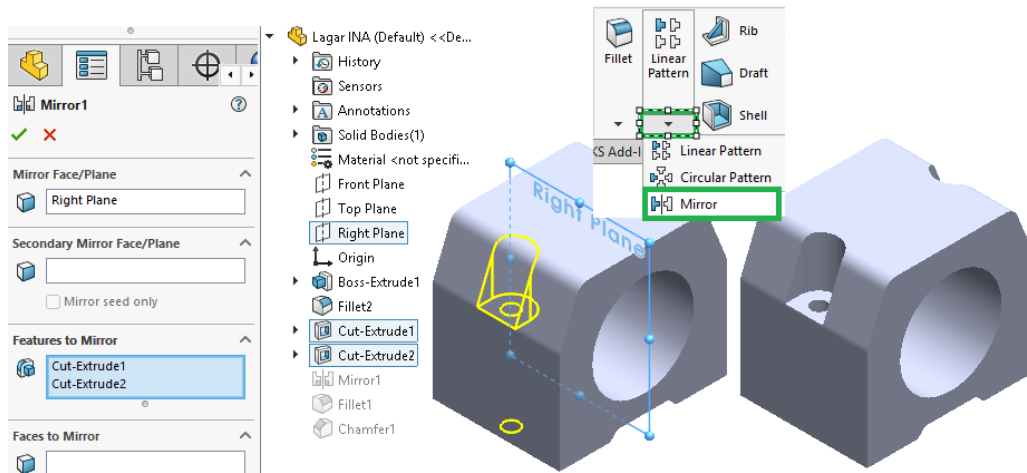
10. Through Hole Cut

To create the required hole, the **Extruded Cut** command is accessed from the **Features** toolbar. Once activated, the termination condition is changed by selecting the **Through All** option, ensuring complete penetration through the part. The operation is confirmed by clicking the **OK** button, producing the hole in accordance with the specified requirements.



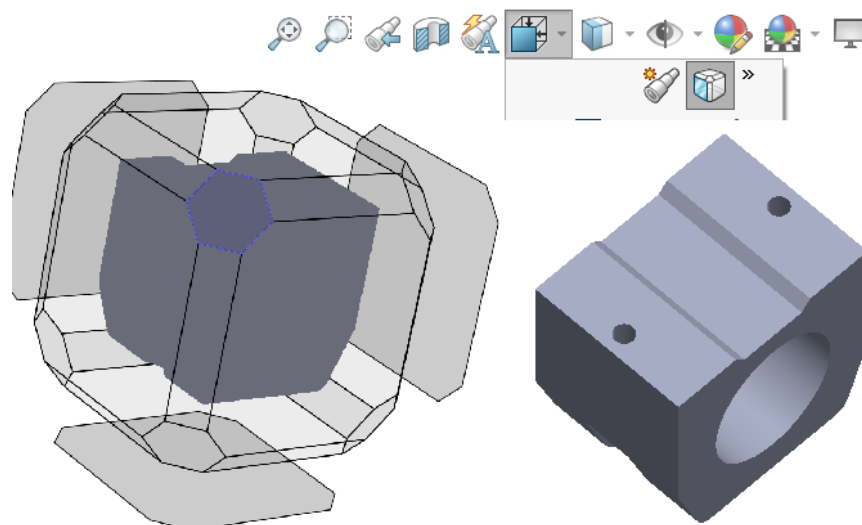
11. Mirroring the Cutouts Using the Mirror Command

To streamline the modeling process, the **Mirror** command is used to create a symmetrical replication of the existing features. From the **Features** toolbar, the **Mirror** command is selected by clicking the arrow next to the **Linear Pattern** command. In the **Mirror Face** field, the **Right Plane** is selected as the plane of symmetry, and the previously created cut features are selected in the **Features to Mirror** field. Upon confirming the operation by clicking the **OK** button, the cutout and the through hole are mirrored symmetrically, ensuring the geometric uniformity of the part.



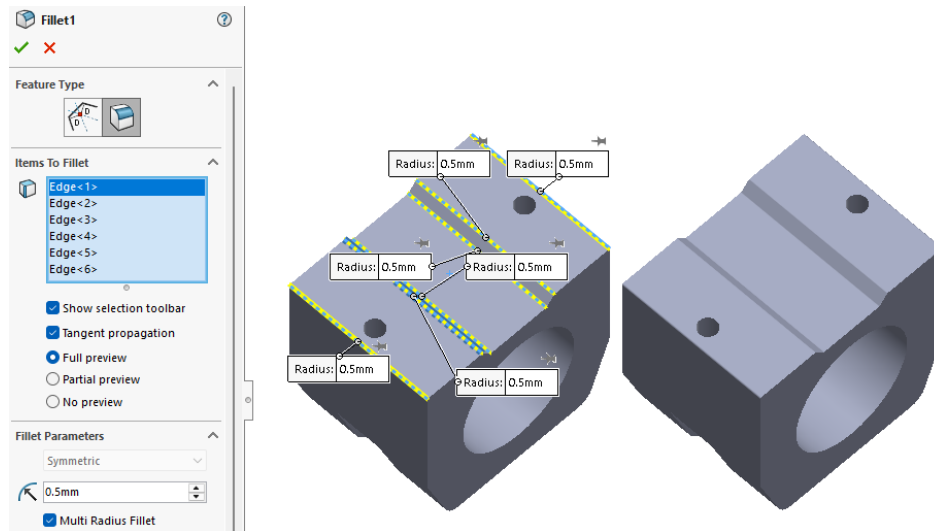
12. Reorienting the Model

To facilitate the rounding of the lower edges, the model must be reoriented. This is achieved by pressing and holding the scroll wheel of the mouse and rotating the model until a convenient perspective is obtained. From the **View** toolbar, the **View Orientation** command is selected, and a corner of the model is chosen from the displayed selector so that the part is viewed in an isometric position.



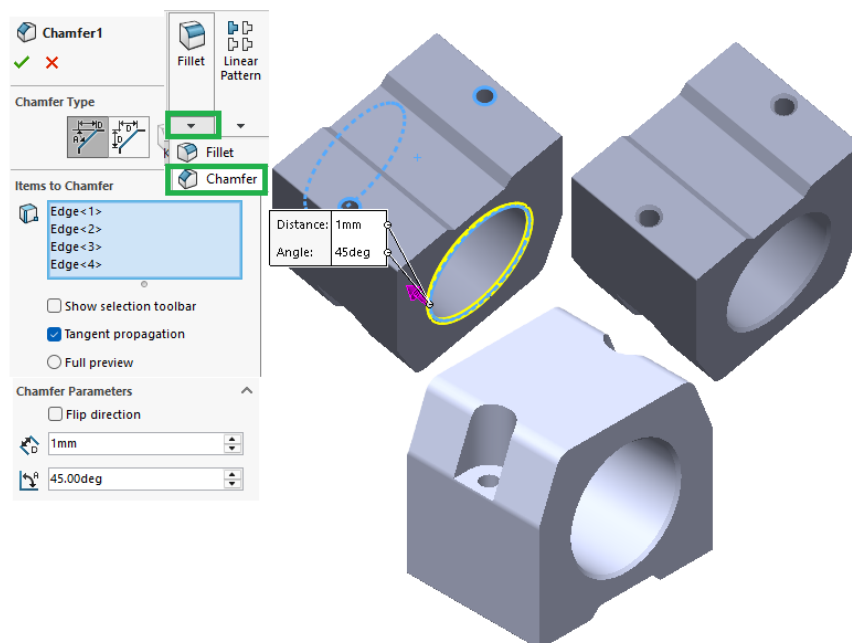
13. Applying Fillets to the Lower Edges

To apply the fillets, the **Fillet** command is accessed from the **Features** toolbar. In the Activity Manager, the **Items to Fillet** field is clicked and the edges requiring rounding are selected individually, with a radius of 0.5 mm specified. The operation is confirmed by clicking the **OK** button, applying the desired fillet to the selected edges.



14. Applying a Chamfer at the Ends of the Central Hole

In the final stage, a 1 mm \times 45° chamfer is applied at both ends of the central hole to improve the surface finish of the part. The **Chamfer** command is accessed from the **Features** toolbar. In the Activity Manager, the **Items to Chamfer** field is clicked and the two circular edges of the hole are selected. The chamfer parameters of 1 mm and 45° are entered in the designated fields, and the operation is confirmed by clicking the **OK** button, applying the desired modification.



2.3.3. Modeling the Lower Guide Shoe of the Timing Chain Drive in an Internal Combustion Engine

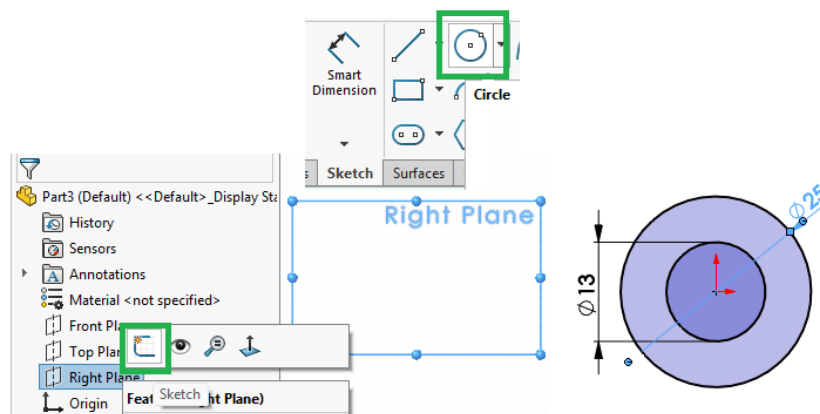
2.3.3.1. Functional Role of the Part in the Valve Train System

The timing chain guide shoe may be either a fixed or a movable element, depending on its role within the system. Fixed guide shoes are mounted on the engine block and ensure the guidance of the chain along a predefined path, preventing lateral deviation. Movable guide shoes are articulated on a pivot shaft and act dynamically on the chain through a hydraulic piston controlled by the oil pressure from the engine lubrication circuit. The piston permanently keeps the movable shoe in contact with the chain, automatically compensating for chain elongation due to wear and maintaining optimal tension throughout the entire service life of the valve train.



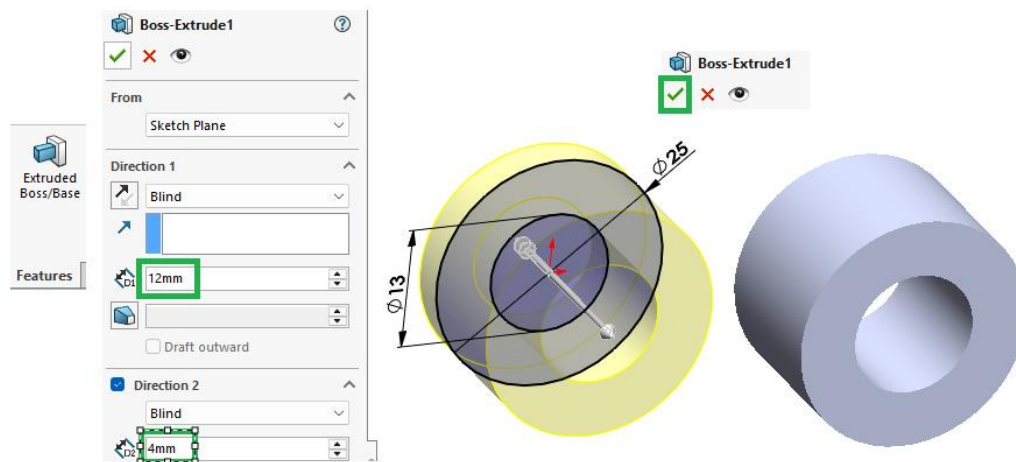
1. Creating the Base Circles

A new Part file is opened in SolidWorks and the working plane is selected. The choice of plane is important as it determines the final orientation of the part in space: **Front Plane** generates the part in a frontal orientation, **Top Plane** generates it as seen from above, and **Right Plane** generates it as seen from the side. For ring or bushing-type parts, **Right Plane** is recommended, as the axis of revolution naturally coincides with the horizontal axis of the coordinate system. The **Circle** command from the **Sketch** toolbar is used to draw two concentric circles, which serve as the base elements for the subsequent construction of the 3D model.



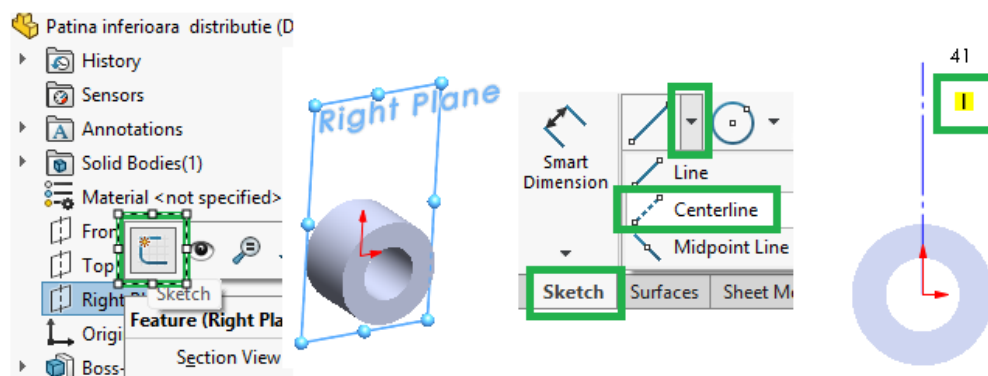
2. Extruding the Base Circles

The base circles are extruded using the **Extruded Boss/Base** command from the **Features** toolbar. Two extrusion directions are defined: a thickness of 12 mm is entered for the first direction and 4 mm for the second direction in the Activity Manager. The command is completed by clicking the **OK** button, resulting in a cylinder with the specified dimensions.

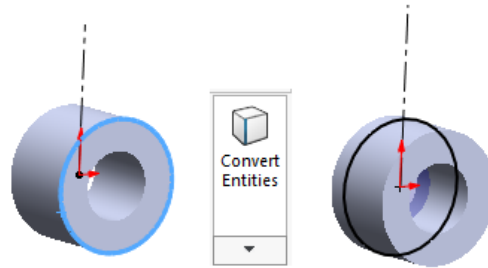


3. Creating the Base Sketch for the Lower Guide Shoe

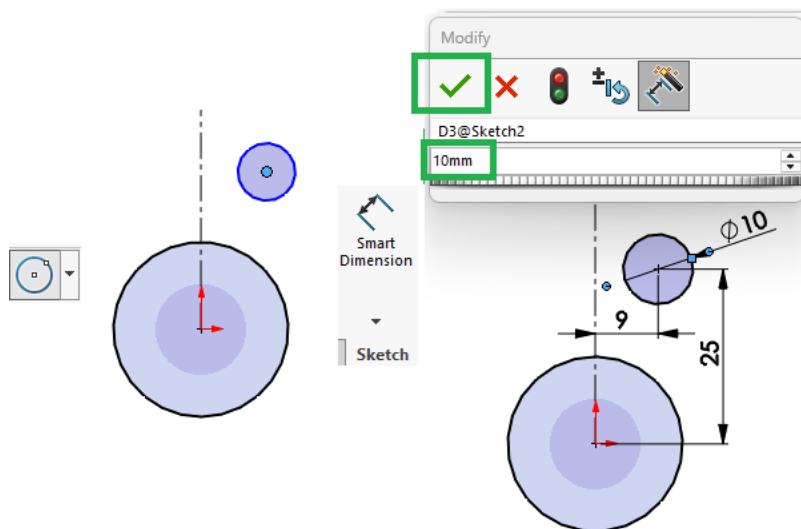
The first step consists of selecting **Right Plane** as the working plane. From the origin, a vertical symmetry axis is traced to an appropriate height using the **Centerline** command from the **Sketch** toolbar. Attention must be paid to the verticality relation: when the yellow verticality indicator appears next to the cursor, a click is made to lock the symmetry axis in the desired vertical position.



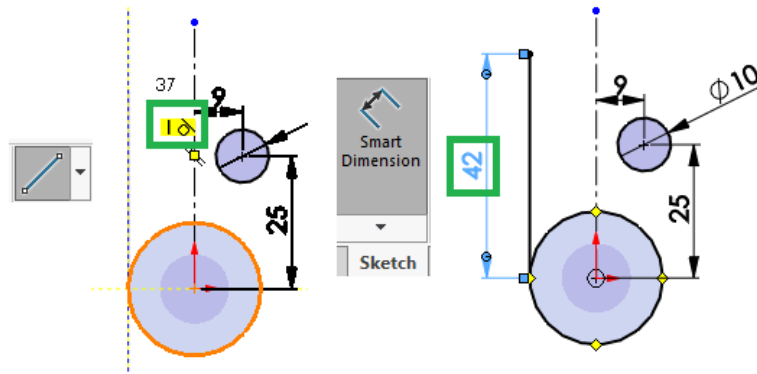
The outer edge of the cylinder is converted to begin a new sketch. The outer edge is selected, and the **Convert Entities** command is activated from the **Sketch** toolbar. This command copies the part edge into the current sketch, allowing it to be used as a reference element for subsequent modeling operations.



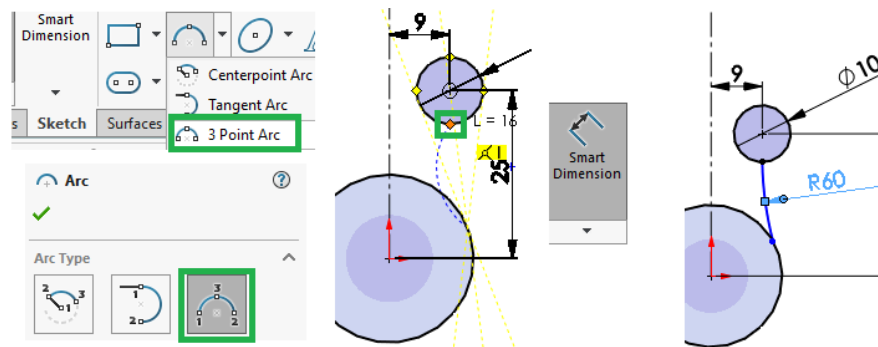
Using the **Circle** command from the **Sketch** toolbar, an additional circle with a diameter of 10 mm is constructed. This circle is positioned at a distance of 35 mm and 8 mm respectively from the symmetry axis. The precise dimensioning and positioning of the circle is carried out using the **Smart Dimension** command from the **Sketch** toolbar. The command is activated, the circle is clicked to define the 10 mm diameter, and the origin and symmetry axis are then selected, followed by the circle, to specify its position relative to the axis. This process ensures the geometric precision required in the design.



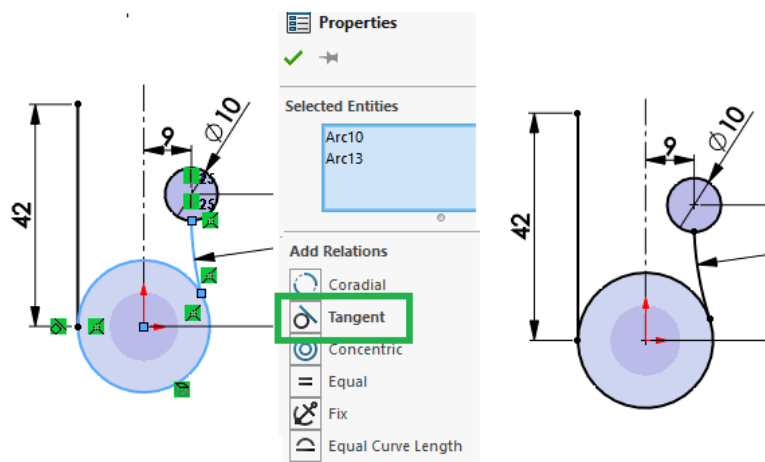
The **Line** command is activated from the **Sketch** toolbar to draw a line tangent to the large circle. A left click is made on the left side of the circle, and the tangency and verticality relations appear next to the cursor, indicated in yellow. Once the correct position is confirmed, a left click is made to create the vertical line. With the **Smart Dimension** command active, the line segment is clicked and a length of 42 mm is entered in the dimension dialog box, establishing the exact length of the line. This process ensures the geometric precision and alignment required in the design.



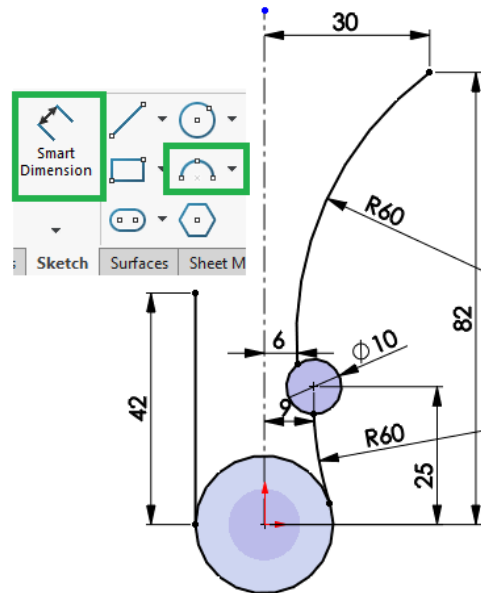
The next step involves drawing a circular arc tangent to both the base circle and the upper circle. The **3 Point Arc** command is selected from the **Sketch** toolbar. The arc is created through three clicks, in accordance with the specifications of this arc type. The first left click is made on the large circle, the second click on the lower point of the small circle, and the third click defines the vertical position of the arc relative to the origin of the small circle. The arc is positioned so that its center of origin lies on the right side. The **Smart Dimension** command is then used to dimension the arc, entering a radius of 60 mm.



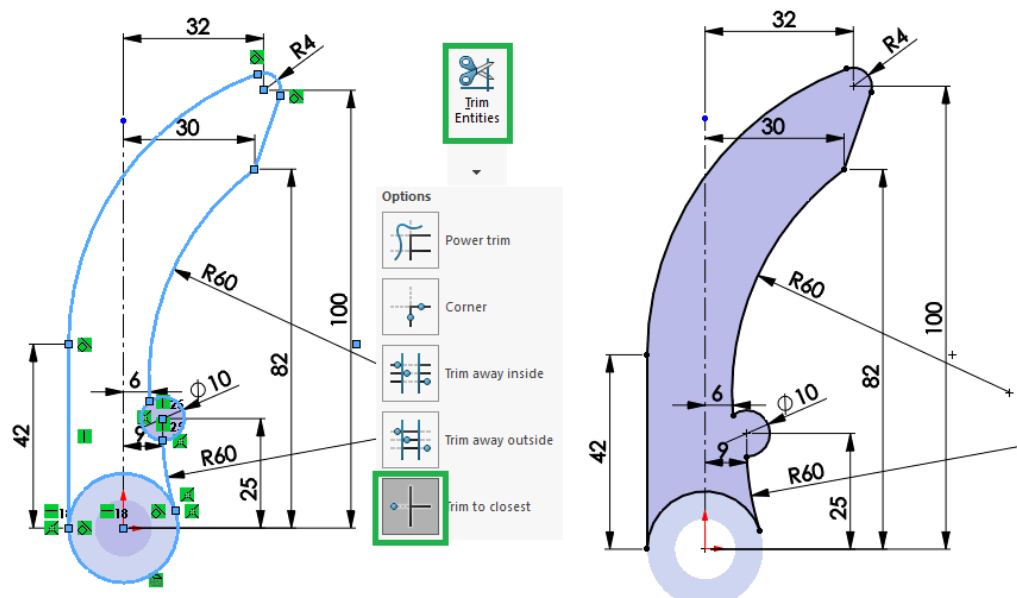
To fully define the arc segment, the base circle is first selected, followed by the arc, while holding the **Ctrl** key. In the Activity Manager, a series of geometric relations will appear in the properties panel. The **Tangent** relation is selected and confirmed by clicking **OK**.



The right side of the sketch contour is completed with a circular arc, which is initially drawn approximately using the **3 Point Arc** command, with the lower end connected to the small circle. It is essential to ensure that the center of the arc is positioned on the right side. The radius of the arc and the positions of both endpoints relative to the symmetry axis are then dimensioned using the **Smart Dimension** command, with the upper endpoint also referenced relative to the symmetry axis.

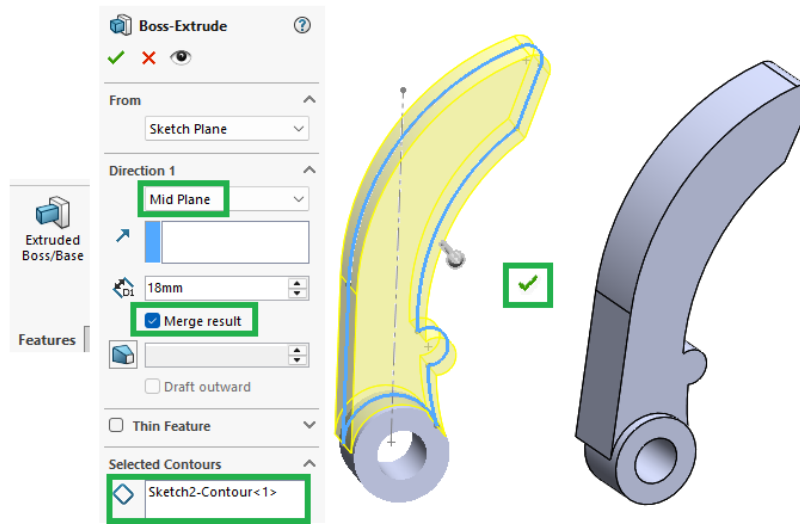


The part contour is constructed using the **Sketch** toolbar commands, incorporating the straight segment and the two circular arcs in accordance with the specified dimensions. Careful attention must be paid to the geometric relations to ensure that the entire sketch is fully defined, which is confirmed when all blue lines turn black. To obtain the final contour shape, the **Trim Entities** command is used to remove unwanted segments, resulting in a clean contour suitable for extrusion.



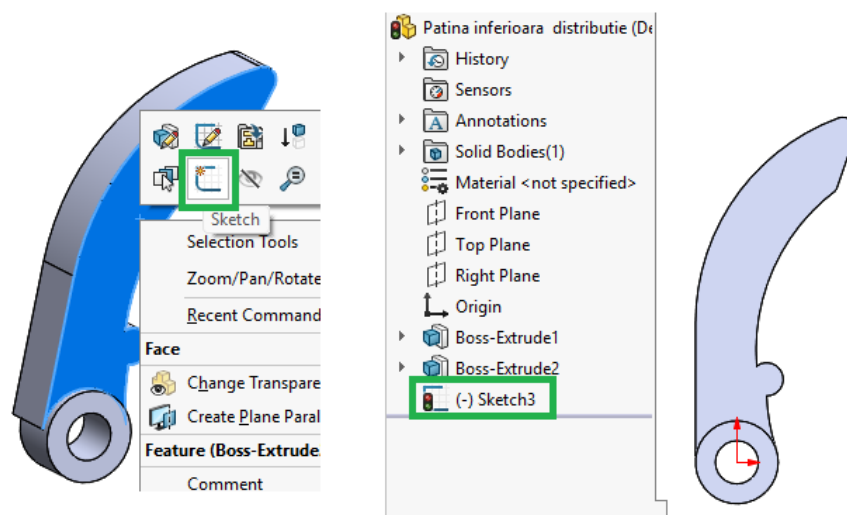
4. Extruding the Sketch to Obtain the 3D Shape

The **Extruded Boss/Base** command is accessed from the **Features** toolbar, and the extrusion condition is set to **Mid Plane** with a thickness of 18 mm. The **Merge Result** option is checked to merge the two extrusions, and the previously created sketch must be visible in the **Selected Contours** field. Once these parameters are configured, the operation is confirmed by clicking the **OK** button, generating the initial three-dimensional shape.

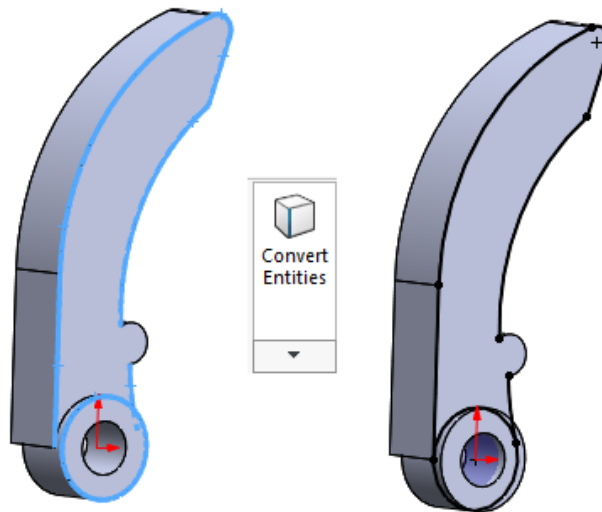


5. Creating the H-Profile Through a Cut Operation

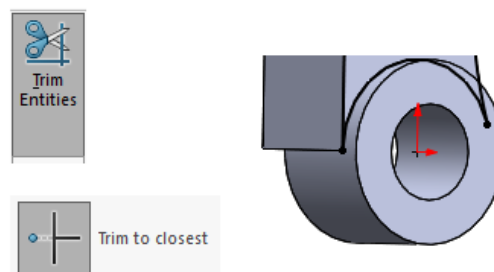
To create the lateral cutout, the corresponding surface is selected with a left click, making it the active plane on which the sketch will be defined. After selection, a right-click opens a contextual menu from which the **Sketch** command is launched, allowing a new working sketch to be created.



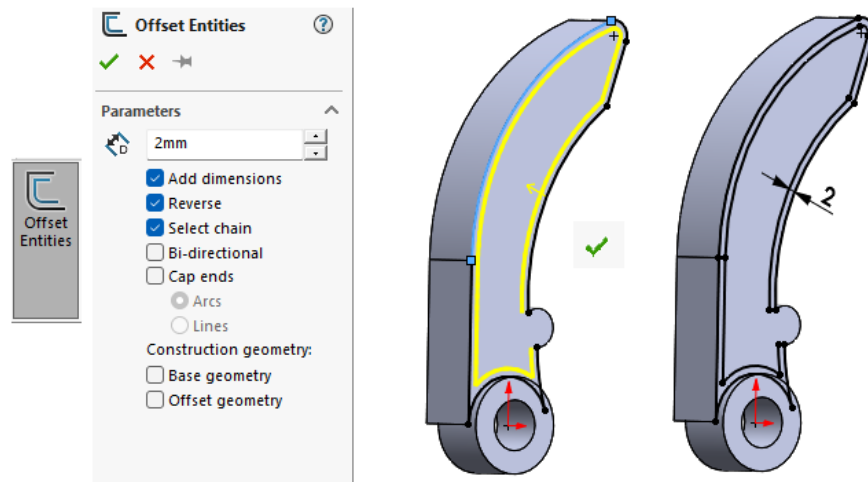
To select the part contours and convert them into the current sketch, the **Ctrl** key is held down and the desired edges are selected. This step allows the contours to be integrated into the active sketch.



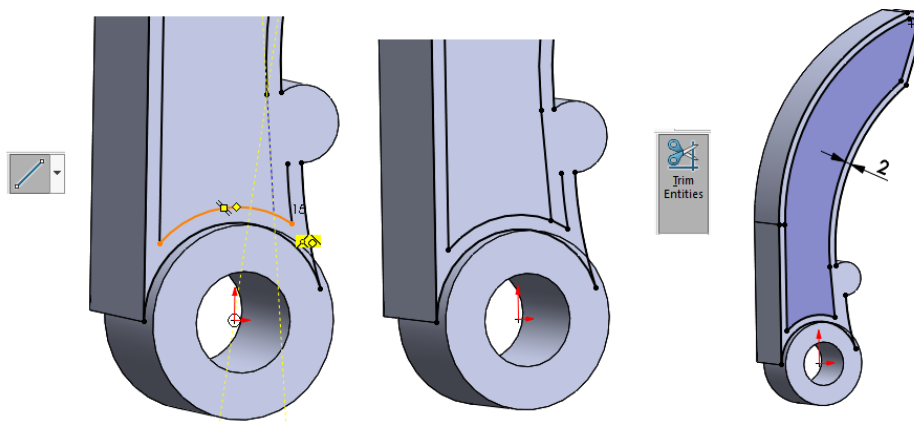
Once the sketch is converted, the lower portion of the circle can be trimmed using the **Trim** command, available in the program toolbar. This functionality allows the selective removal of geometric segments, contributing to the creation of a precise sketch adapted to the design requirements.



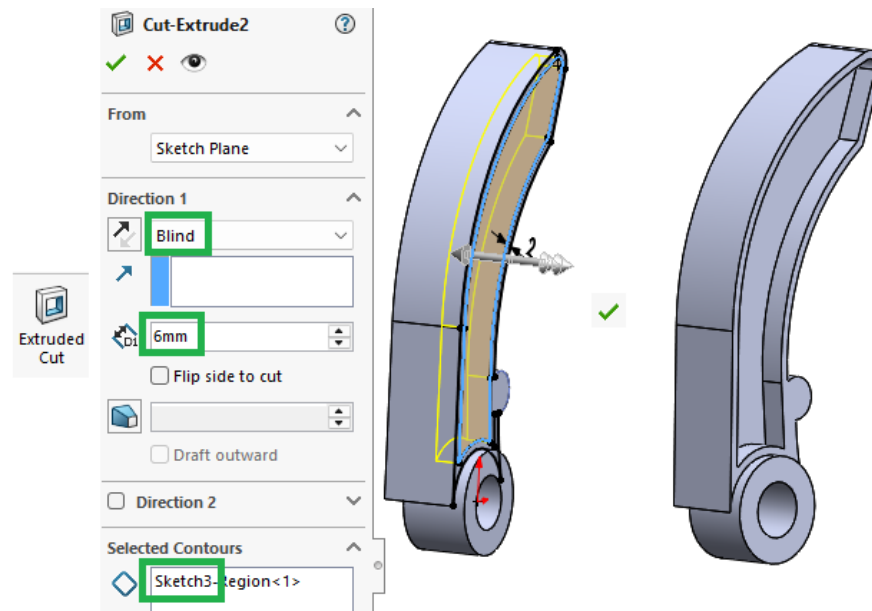
The contour is duplicated using the **Offset** command from the **Sketch** toolbar. A value of 2 mm is entered as the offset distance between the lines, and the **Reverse** option is selected to bring the sketch inside the part. The operation is completed by clicking the **OK** button.



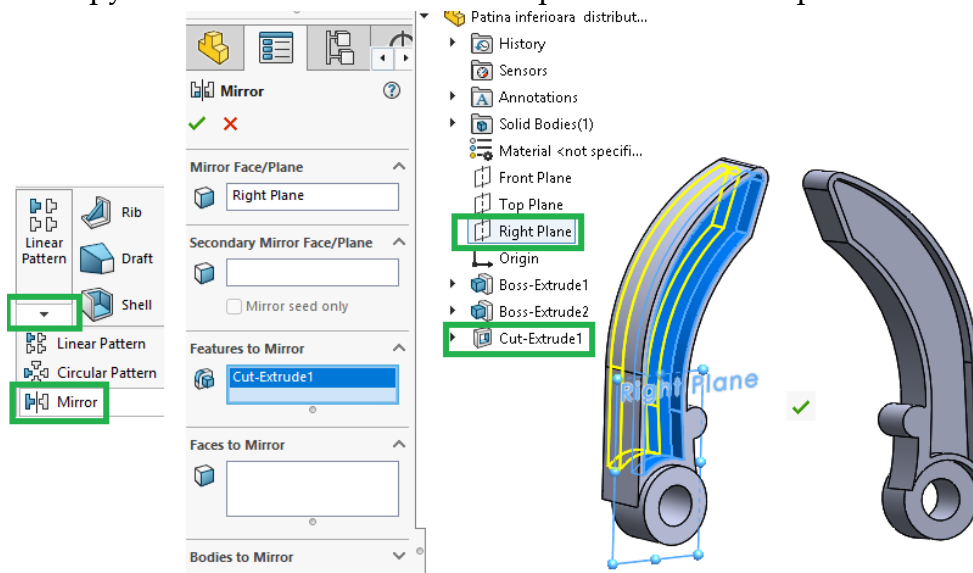
The **Line** command is accessed and a line tangent to the circular arc is drawn in place of the missing segment. The **Trim** command is then used to remove the unwanted lines from the sketch, resulting in a well-defined contour ready for the cut operation.



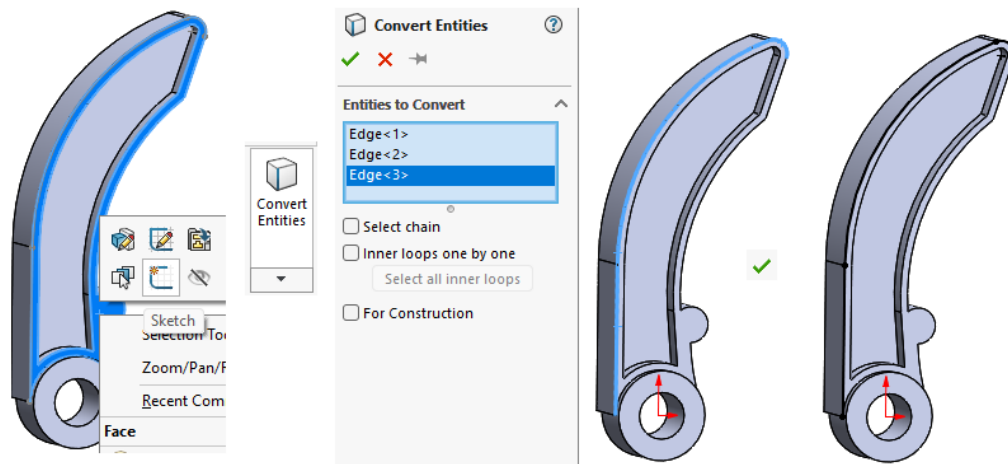
From the **Features** toolbar, the **Extruded Cut** command is used to remove material from the interior of the part, bounded by the previously created sketch. The cut is performed to a depth of 5 mm, creating a precise cutout in accordance with the geometry defined by the sketch.



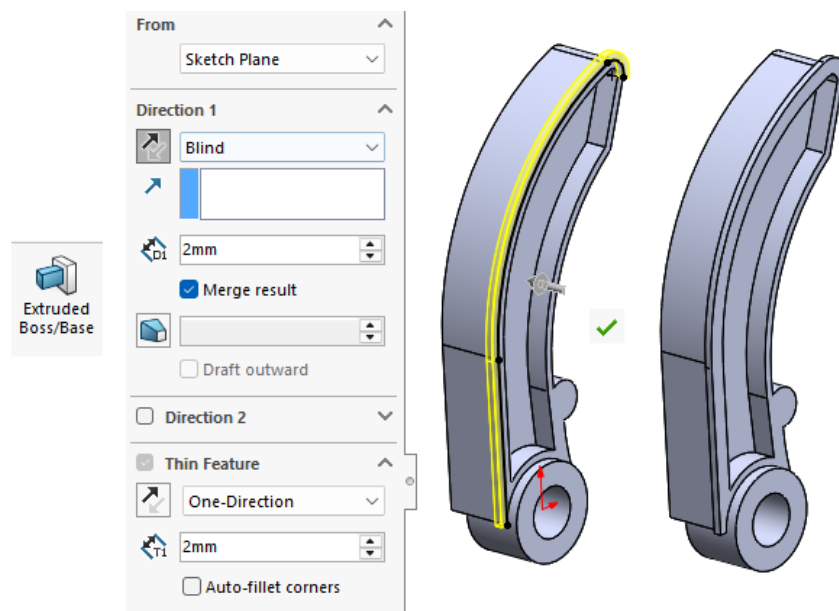
To perform this operation, **Right Plane** is selected as the reference plane and the **Cut-Extrude 1** operation is selected in the **Features to Mirror** field. This process ensures the creation of a symmetrical copy of the cutout relative to the specified reference plane.



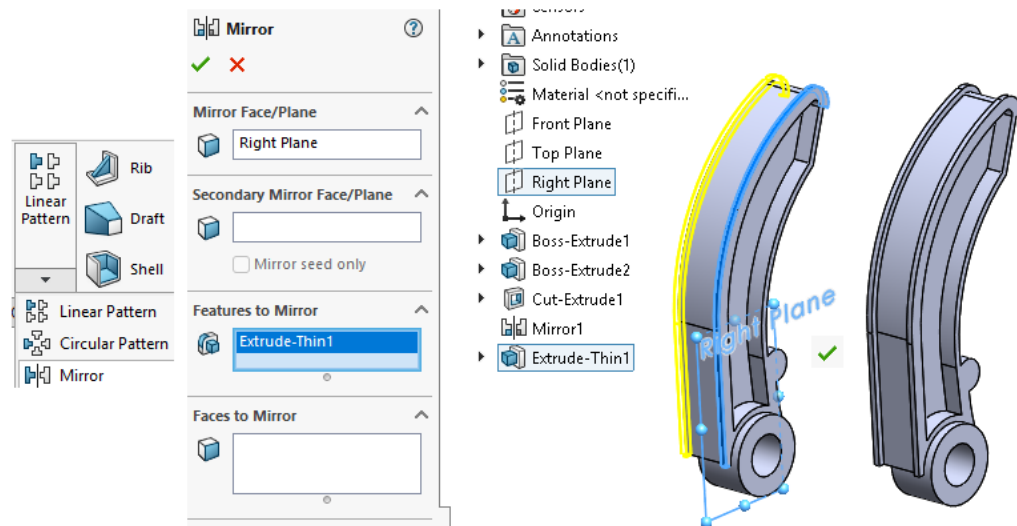
To create the lateral guide on the side of the shoe, a new sketch is constructed on the lateral surface of the part, allowing a portion of its edge to be converted. After selecting the edges as shown in the corresponding figure, the **Convert Entities** command is invoked from the toolbar. The selected edges appear in the selection window, and the process is completed by clicking **OK**, ensuring the integration of the edges into the active sketch.



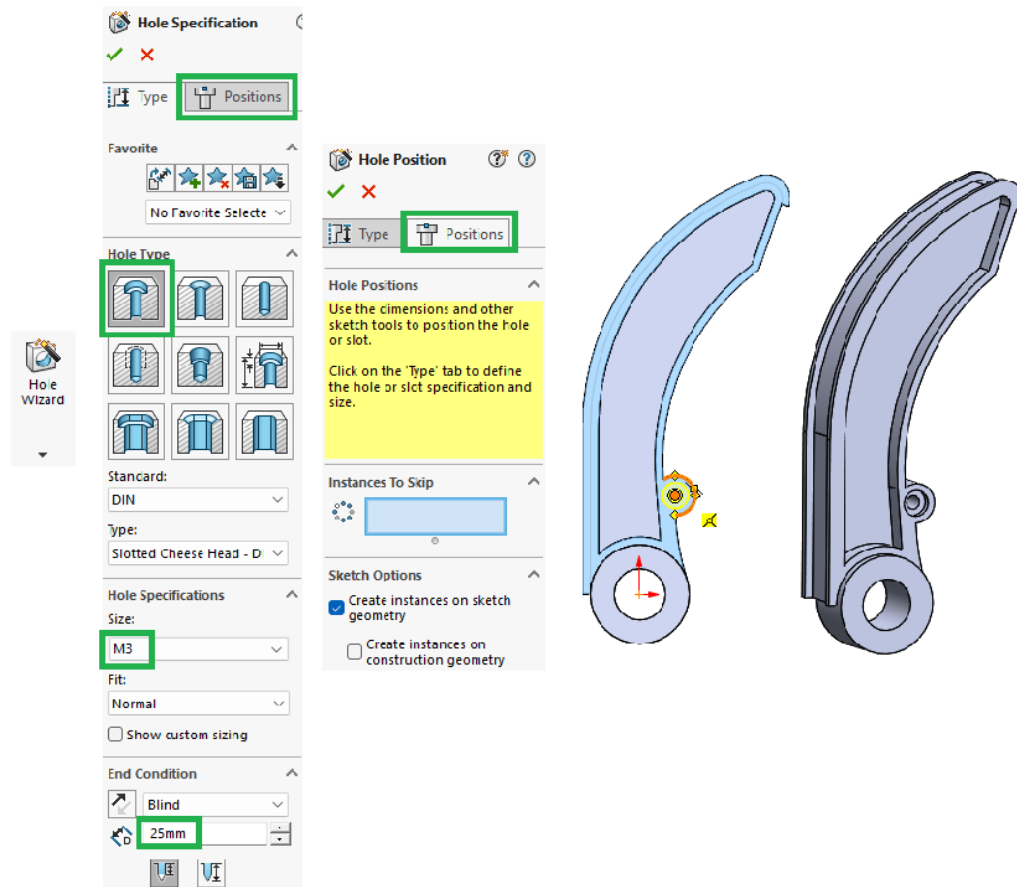
From the **Features** toolbar, the **Extruded Boss/Base** command is selected. With the **Thin Feature** option activated, the edge extrusion is performed. It is essential to verify the orientation of the arrow, which must point toward the model. The arrow direction can be reversed using the **Reverse Direction** button, marked with a double arrow. A value of 2 mm is entered in the direction field, and a thickness of 2 mm is also specified. The process is completed by clicking the **OK** button.



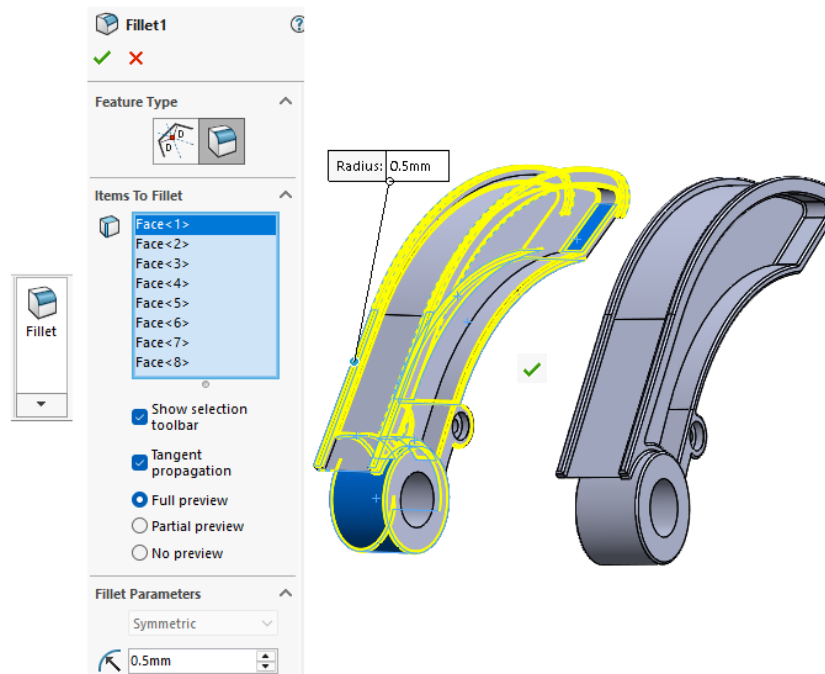
The guide is mirrored using the **Mirror** command from the **Features** toolbar. **Right Plane** is selected as the mirror plane, and the **Extrude-Thin1** operation is selected from the feature tree in the **Features to Mirror** field. The process is completed by clicking the **OK** button, ensuring the creation of a symmetrical copy of the guide relative to the specified plane.



A through hole is created in the part using the **Hole Wizard** command from the **Features** toolbar. After accessing the command, the **Counterbore** hole type is selected. A depth of 25 mm is entered for the hole dimension. The **Positions** menu is then accessed to specify the hole location. The cursor is positioned over the circular arc until its origin point appears, at which point a left click is made on the origin to place the hole concentrically with the part edge in the semicylinder region. This process ensures the precise alignment of the hole with respect to the part geometry.



Fillets are applied using the **Fillet** command, accessible from the **Features** toolbar. By selecting the edges or faces of the skate one at a time, they will appear listed in the Property Manager window. A fillet radius value of 0.5 mm is entered, and the software provides a preview option for preliminary verification. Clicking the **OK** checkmark confirms the operation, producing a final model with fillets applied according to the specified parameters.



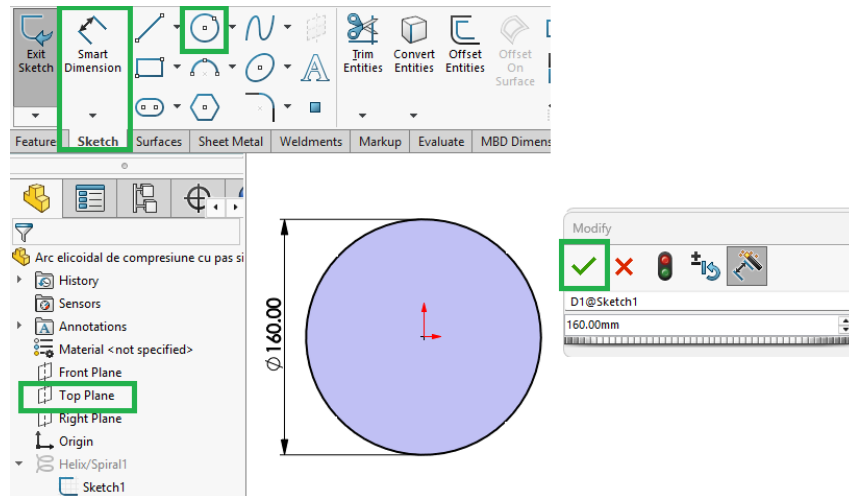
2.3.4. Modeling a Helical Compression Spring with Variable Pitch and Diameter

1. Creating a New Part File

To begin modeling, navigate to **File** → **New**, then select the **Part** document type and confirm with **OK**.

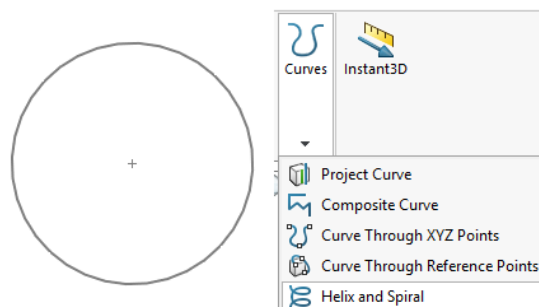
2. Creating a Sketch for the Base Diameter of the Spring

An appropriate working plane is selected, such as the horizontal plane (**Top Plane**). The **Sketch** command is then accessed from the **Sketch** menu bar to create a sketch containing a circle centered at the origin with a diameter of 160 mm, which defines the base diameter of the spring. The circle's dimension is set using the **Smart Dimension** command by selecting the circle. In the properties window that opens, the value 160 mm is entered and confirmed by clicking the **OK** checkmark.



3. Defining the Spring Profile Using the Helix Command

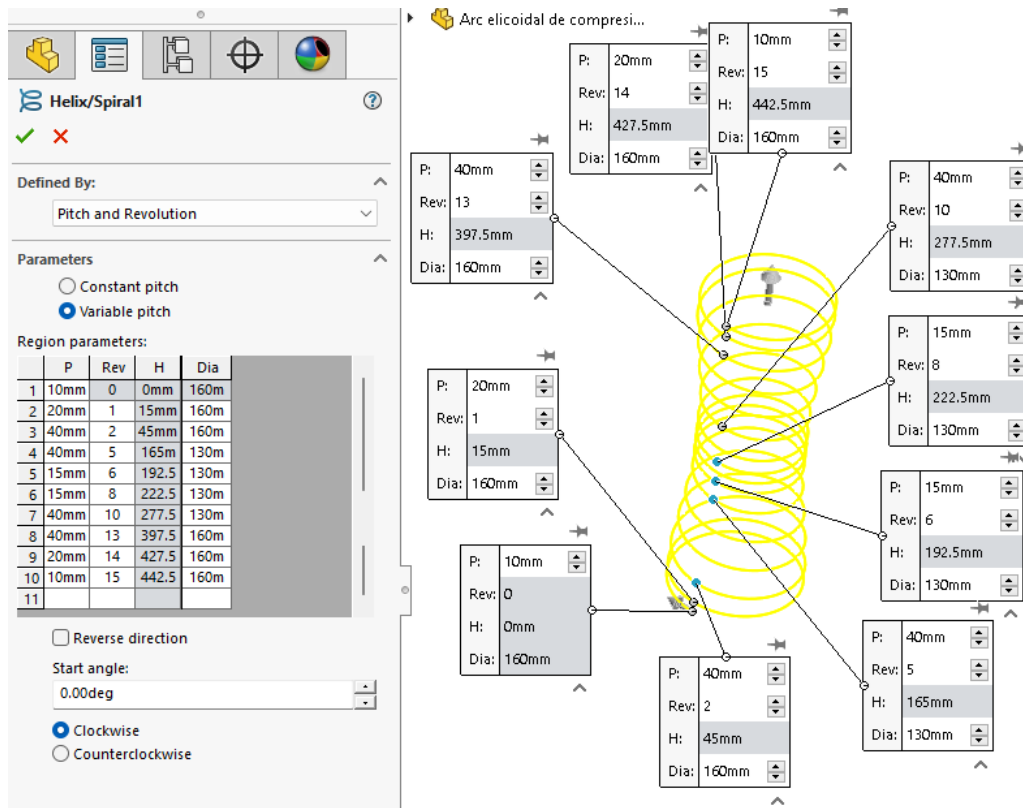
After completing the circle sketch, the **Curves** command is accessed from the **Features** toolbar, and the **Helix and Spiral** command is selected from its dropdown menu. This opens the options menu for defining a helical or spiral curve, allowing the configuration of the necessary parameters for generating the spring.



The helix parameters are set using the **PropertyManager**. As the first parameter, the **Variable Pitch** option is selected, and each entry in the corresponding table is filled in according to the reference figure, including parameters such as pitch, number of revolutions, height, and diameter.

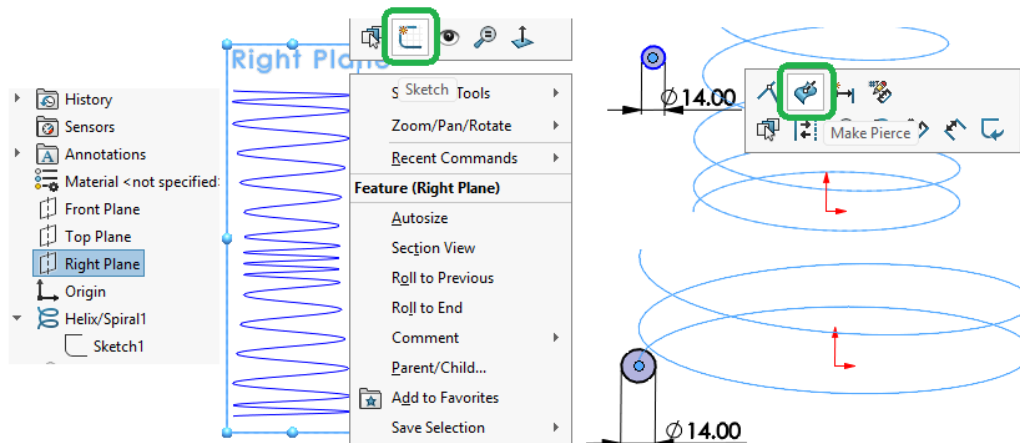
The helix will be displayed in the preview according to the entered data. If rotation is needed to verify the sketch, the middle mouse button can be held down while moving the cursor to adjust the viewing angle. Clicking the **OK** checkmark generates the helix.

These parameters influence the shape and behavior of the helical spring, ensuring the precision required in the design process.



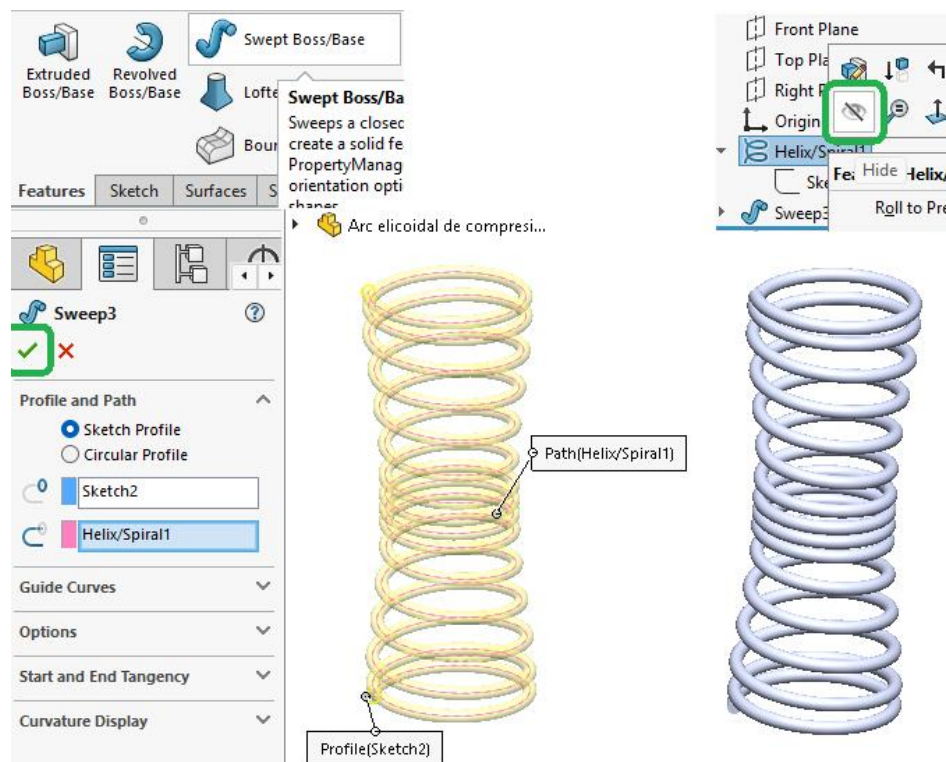
4. Creating the Cross-Sectional Profile of the Spring

A plane perpendicular to the helix path is selected, such as the **Right Plane**, and a new sketch is opened using the **Sketch** command. The cross-sectional profile of the spring is drawn as a circle with a diameter of 14 mm, placed at an arbitrary position on the plane. By holding the **Ctrl** key and simultaneously selecting the center of the circle and the first coil, a series of geometric relations becomes available, among which the **Make Pierce** option appears. Selecting this relation links the circle to the origin of the helix, ensuring correct alignment of the cross-sectional profile with the helical path.



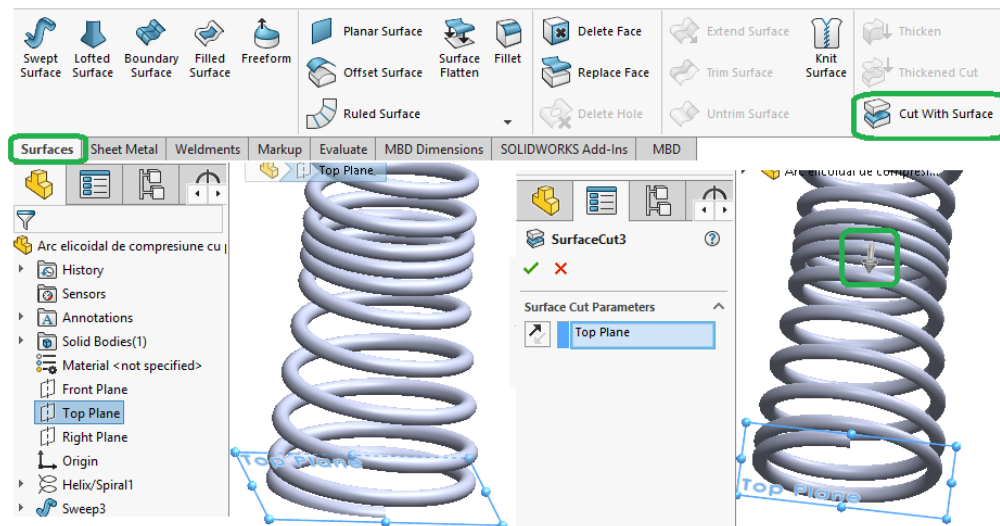
5. Obtaining the Spring Volume Using SWEEP

Using the **Sweep** command from the **Features** menu bar, the drawn profile is selected as the cross-section and the previously defined helical path is selected as the trajectory. SolidWorks automatically generates the helical spring based on these settings, maintaining a spring diameter of 160 mm and a coil diameter of 14 mm, after which the **OK** checkmark is clicked to complete the operation. To hide the helix, right-click on **Helix/Spiral1** in the feature tree and select the **Hide** option from the context menu. This action removes the helix from view, retaining only the geometry of the generated spring.



6. Trimming the Lower Base of the Spring Using the Cut Command

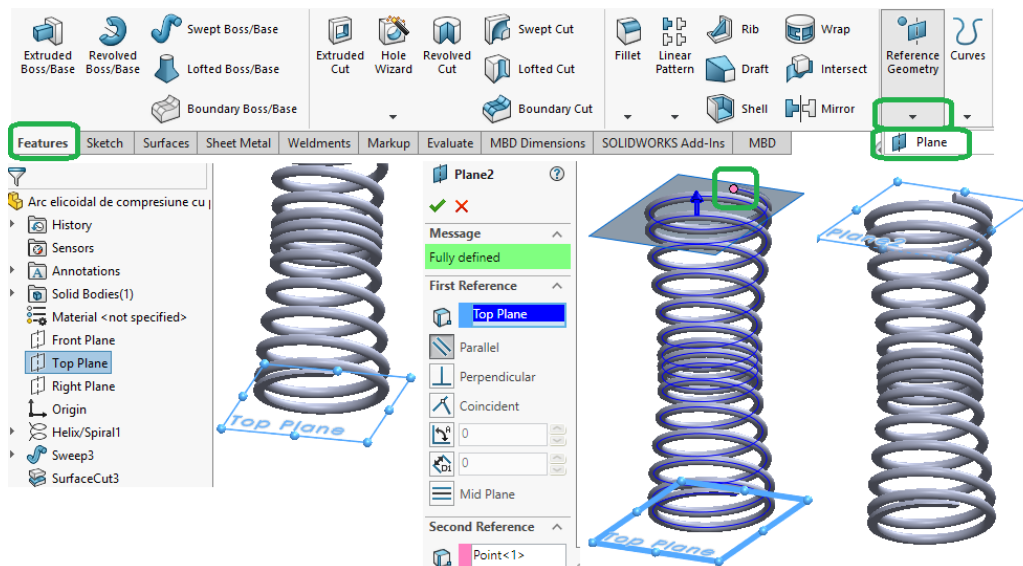
The **Cut With Surface** command from the **Surface** menu bar is used to remove material from the ends of the spring. **Top Plane** is clicked to select the working plane, after which the **Cut With Surface** command is accessed. The arrow direction must be oriented downward, indicating the direction in which material is cut from the model. Clicking the **OK** checkmark then confirms the operation, permanently removing the material and completing the cut.



7. Trimming the Upper Base of the Spring Using the Cut Command and Finalizing the Model

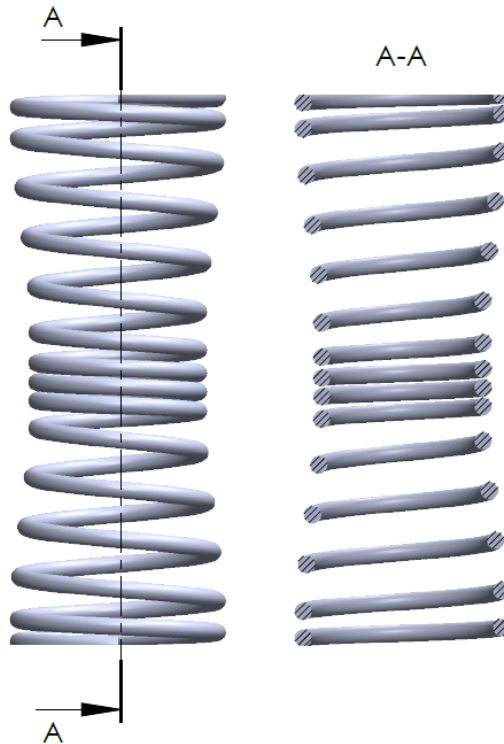
To repeat the cutting procedure at the upper base, a new cutting plane must be constructed. **Top Plane** is clicked, then from the **Features** menu bar the arrow next to **Reference Geometry** is expanded, where the **Plane** command is located. In the **PropertyManager**, the correct position of the new plane is configured: **Top Plane** will appear at **First Reference**, while the upper end of the helix must be selected at **Second Reference**, thereby defining the plane's position. The command is completed by clicking the green **OK** checkmark.

Finally, the previous step (5) is repeated to trim the upper end, paying attention to the arrow orientation, which must point upward at this stage of the trimming operation.



The completed model is presented below, and as can be observed, both the pitch and diameter are variable in this compression spring. To create a section view, the **Section View** command is accessed from the **View** toolbar. In the command's **PropertyManager**, **Right Plane** is selected in the **Section 1** field as the cutting plane, this being the axial plane passing through

the axis of the spring. After confirming with the **OK** checkmark, the model is displayed in section view, and the visualization can be toggled between sectioned and full view using the **Section View** button in the toolbar.

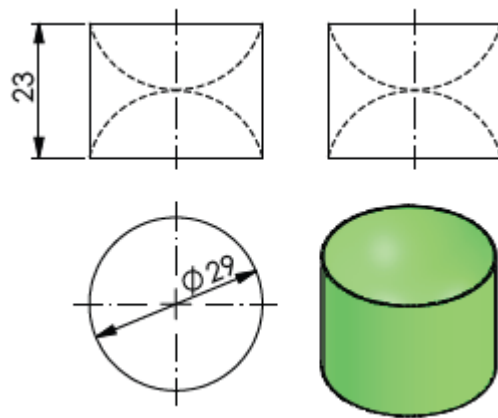


2.3.5. Independent Practice Exercises

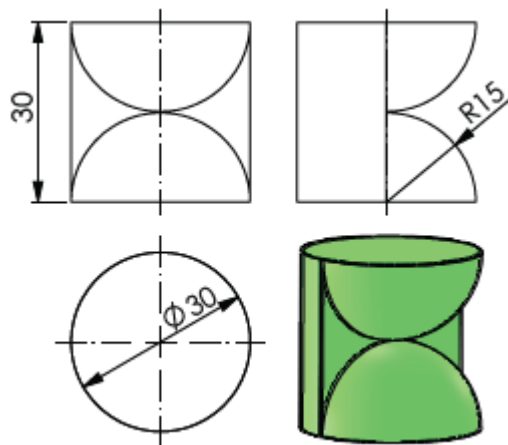
The exercises proposed below are primarily didactic in nature, designed to consolidate and deepen the use of the commands presented throughout the chapter. Working through them will provide practice with both the **Sketch** module commands and the specific operations of the **Features** module, thereby developing the ability to approach parametric modeling of parts with varied geometry.

Dimensions not specified in the reference figures are to be determined constructively, respecting the visual proportions of the part and the general principles of technical representation in engineering drawings.

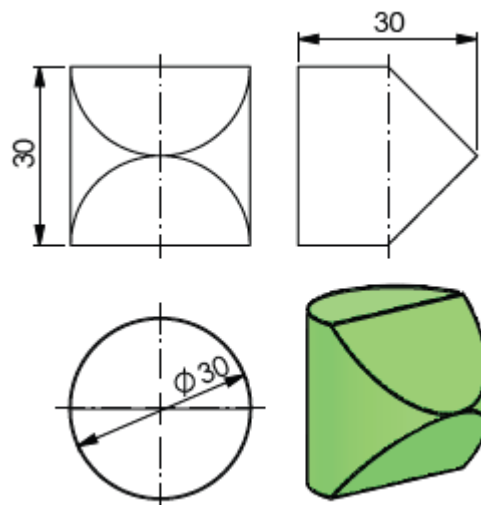
2.3.5.1. Modeling a Cylindrical Body with Opposing Spherical Interior Cavities



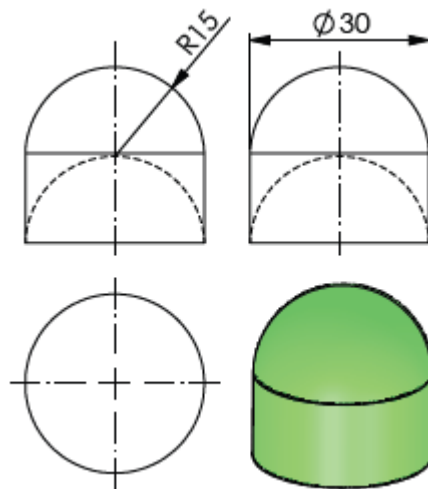
2.3.5.2. Cylindrical Body with Opposing Exterior Domes



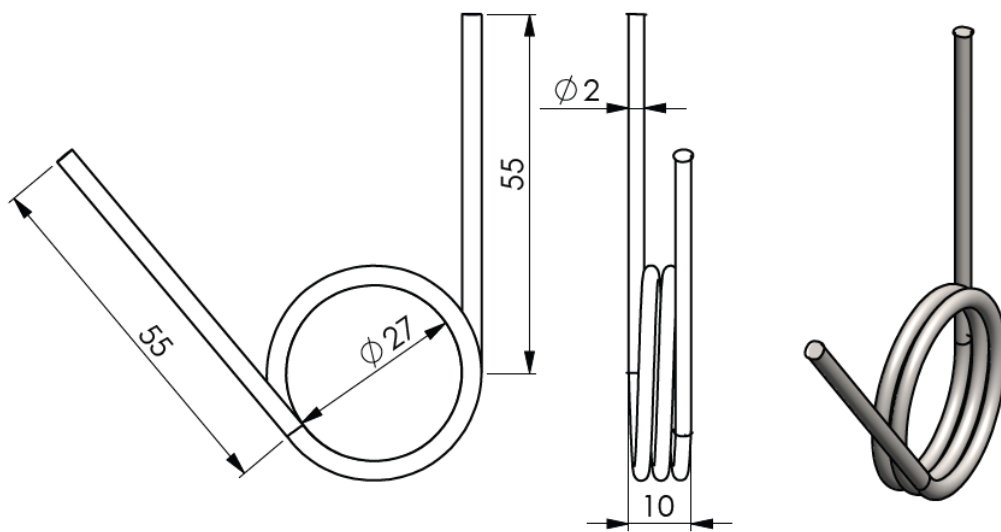
2.3.5.3. Cylindrical Part with Orthogonal Conical Cutouts



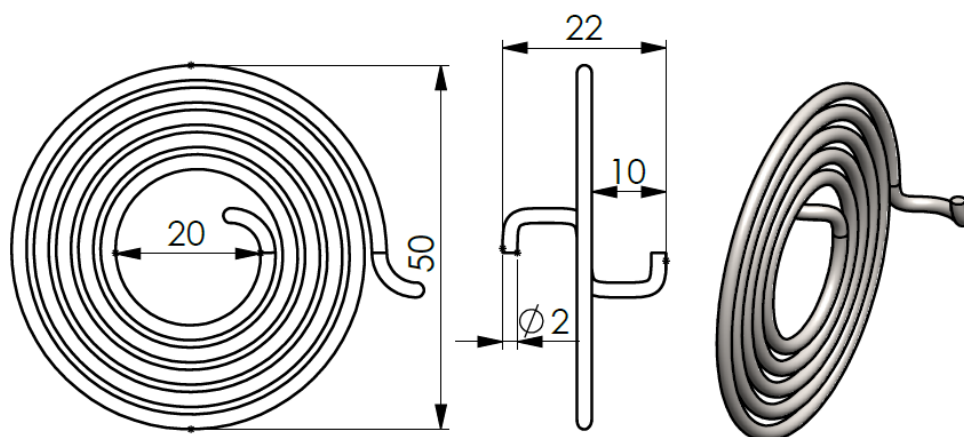
2.3.5.4. Modeling a Dome-Shaped Body



2.3.5.5. Torsion Spring with Straight Arms



2.3.5.6. Flat Spiral Spring with Mounting Hooks



2.3.6. Assessment Questions

1. In what situations is the use of the Hole Wizard command recommended?

2. What is the role of the Helix/Spiral curve in spring modeling?

3. Why does the order of operations affect the quality of the parametric model?

4. What are the main steps in establishing a modeling strategy?

3. MODELING THE COMPONENTS OF A DIESEL FUEL INJECTOR

3.1. Chapter Objectives

This chapter aims to develop an integrated applied approach through the individual modeling of the components of a fuel injector. The case study allows the correlation of parametric modeling concepts with the functional analysis of parts and with an understanding of the role of each component within a real mechanical subassembly (Figure 3.1).

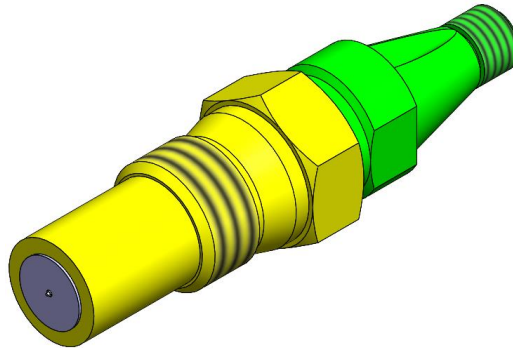


Figura 3.1 3D model of the fuel injector created in SolidWorks

Modeling the components of a fuel injector is an essential process in the development and optimization of injection systems for internal combustion engines. Through the use of a modeling application, precise 3D models of individual parts can be created such as the injector body, injector needle, pressure spring, and nozzles, enabling functional analysis, compatibility verification, and simulation of behavior under real operating conditions. This exercise not only strengthens computer-aided design (CAD) skills, but also familiarizes students with the technical specifications and functional requirements of a complex injection system. Modeling the injector in a dedicated application therefore serves as a valuable tool for developing a thorough understanding of design and manufacturing processes, as well as for the technical preparation of future engineers.

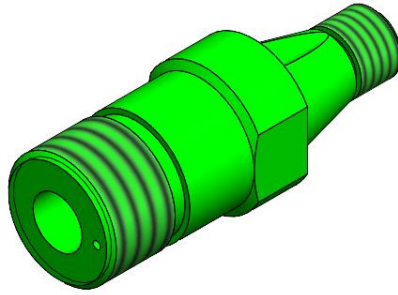
3.2. Learning Outcomes

By the end of this chapter, the student will be able to:

- to identify the main components of a fuel injector;
- to correlate the constructive shape of components with their functional requirements and assembly conditions;
- to model individual parts with varying degrees of geometric complexity;
- to correlate the constructive shape of components with their functional requirements.

3.3. Theoretical and Applied Content

3.3.1. Injector Body Modeling

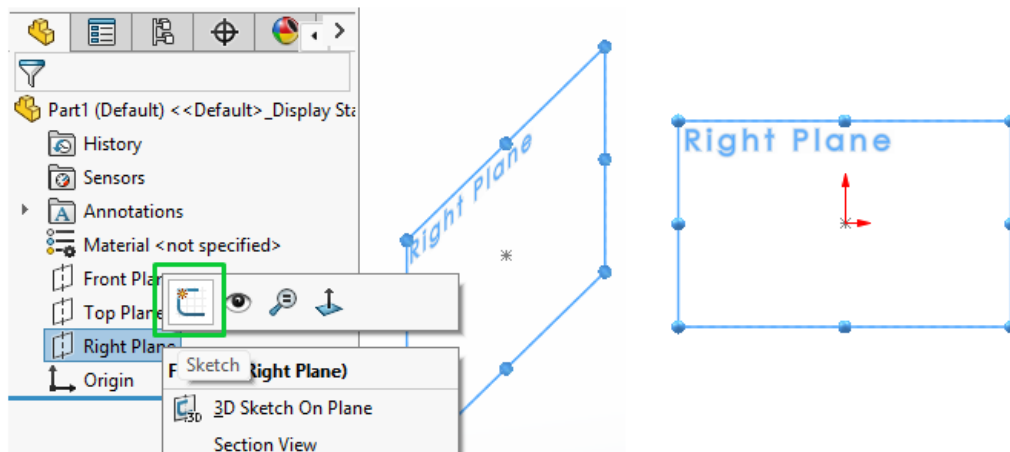


1. Creating a New Part File

The first step in the part modeling process consists of opening the SolidWorks application and selecting **File** → **New** → **Part** to create a new part file.

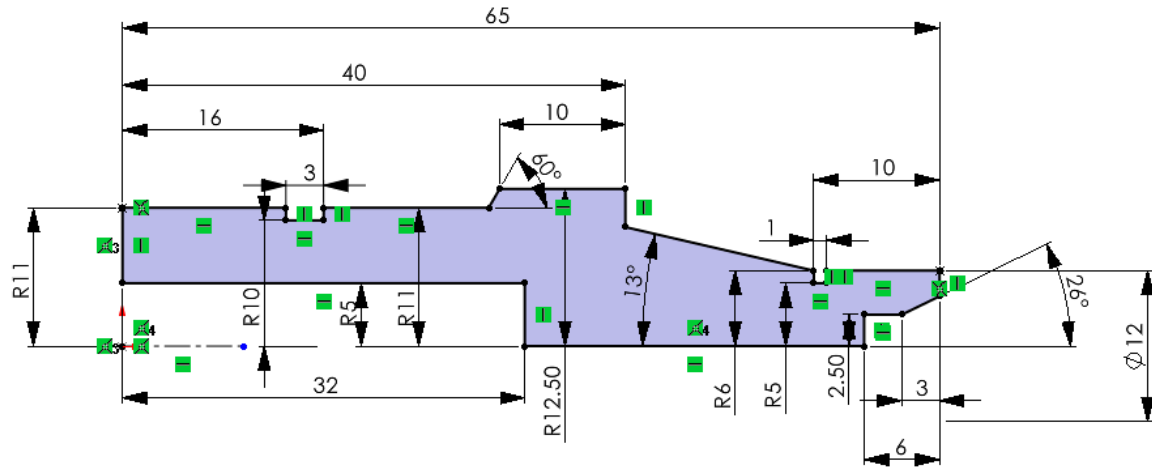
2. Sketching the Base Profile

To begin modeling the injector body, the **Right Plane** is selected as the working plane by right-clicking on it, and sketch mode is opened using the **Sketch** command.



3. Creating and Dimensioning the Profile of the Injector Body

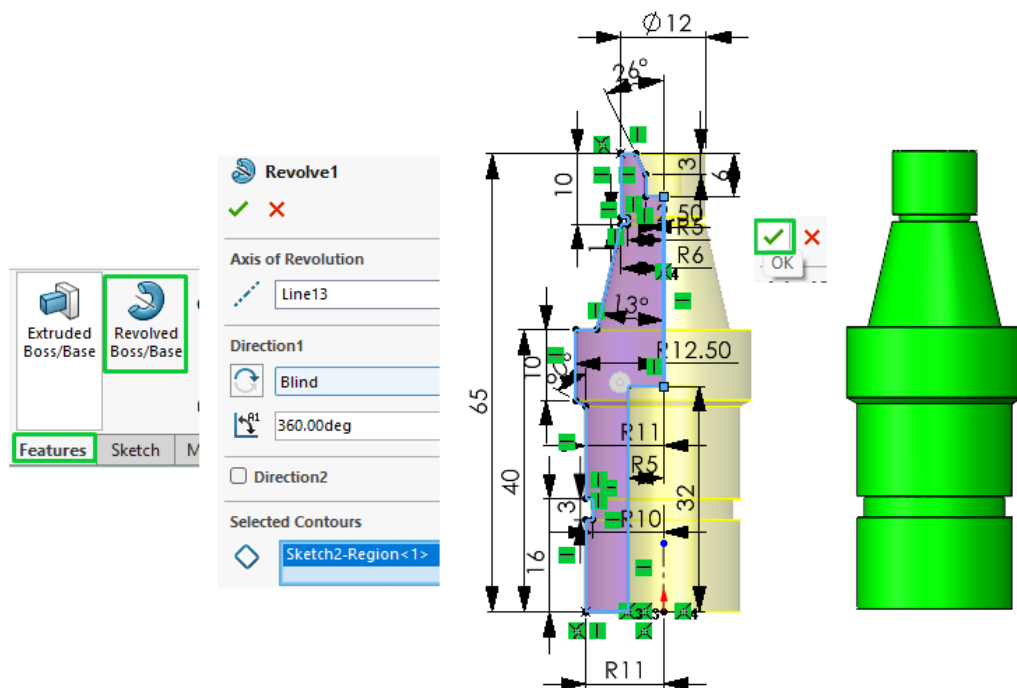
In the first phase, using the commands from the **Sketch** toolbar, the **Centerline** command is used to draw a symmetry axis from the origin along the horizontal. This axis serves as a reference element for the alignment and symmetric construction of the profile. The outer contour of the injector body profile is then drawn, paying close attention to geometric details and compliance with the technical specifications. The **Smart Dimension** command is used to define the geometric dimensions, respecting the measurements shown in the sketch below.



4. Generating the Body by Revolution

After completing the sketch, the **Revolve Boss/Base** command is accessed from the **Features** toolbar. In the **PropertyManager**, the **Axis of Revolution** box is clicked, and the symmetry axis around which the rotation is to be performed is then selected. In the **Direction** window, the **Blind** option is chosen, and the rotation angle is set to 360 degrees.

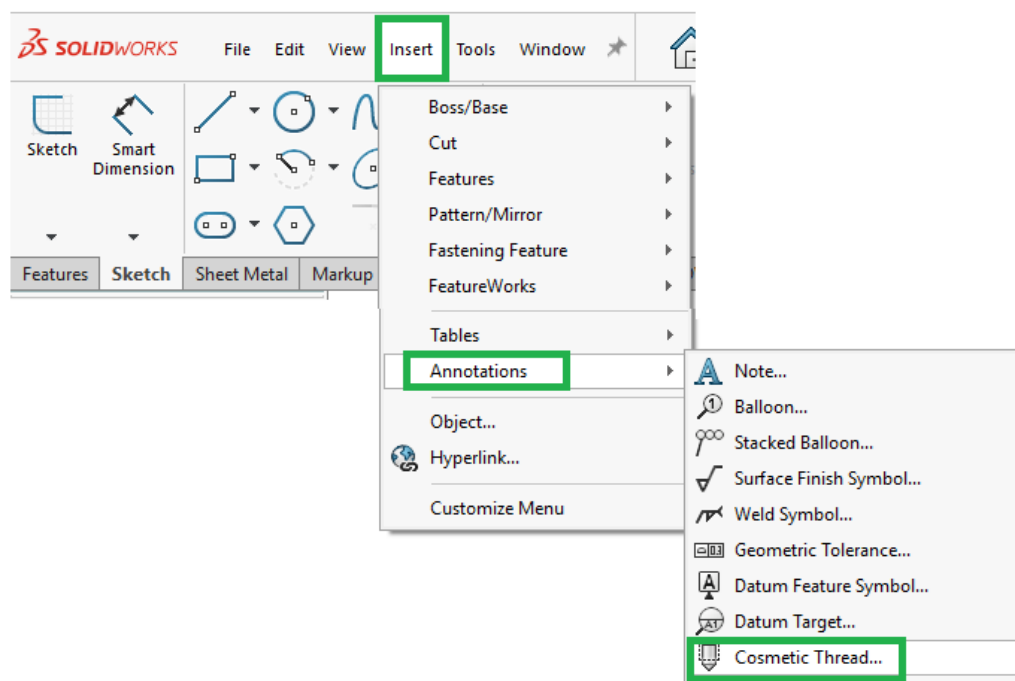
In the **Selected Contours** section, the sketch created in the previous step is selected. The software provides an instant preview of the volume to be generated, displayed in a semi-transparent yellow color. After confirming the settings by clicking the green **OK** checkmark, the 3D volume is generated automatically, producing the injector body in its final form. This process highlights the efficiency and precision of the **Revolve** command in transforming a 2D profile into a three-dimensional model.



5. Representing Threads Using the Cosmetic Thread Command

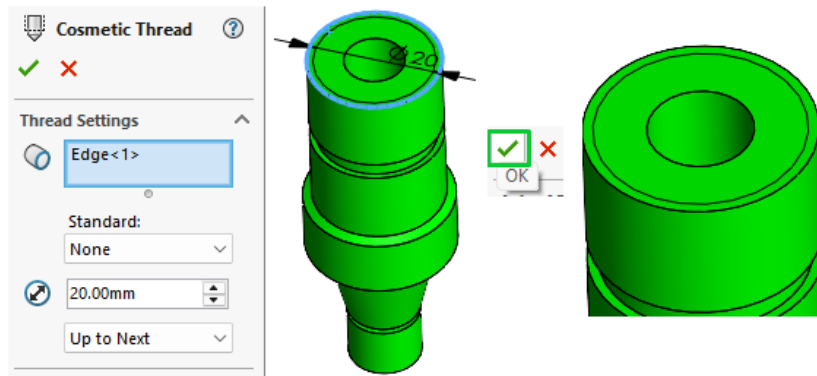
Since components such as the nozzle cap and fuel line are mounted on the injector body, it is necessary to represent threads on both ends. For this purpose, the **Cosmetic Thread** option is used, which allows the visual representation of the thread without geometrically modifying the 3D model.

From the menu bar, the **Insert** section is accessed, where the **Annotations** submenu is opened. The **Cosmetic Thread** command is then selected. This functionality adds a symbolic representation of the thread, complying with technical standards and specifications, without increasing the complexity of the model.

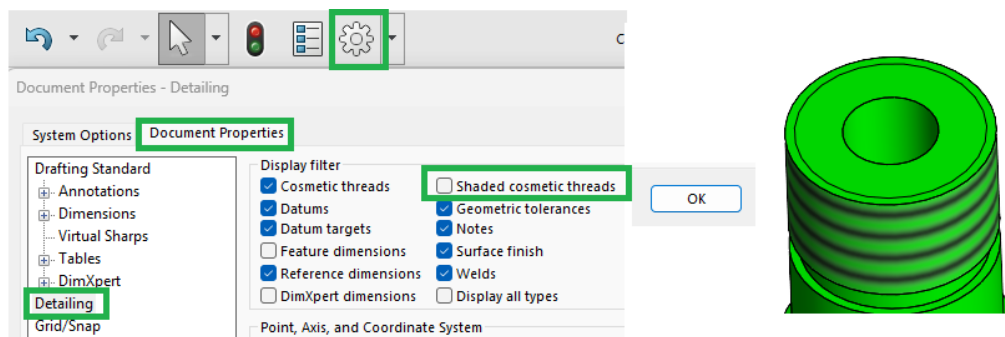


The **Cosmetic Thread** command is launched with a left click. In the **PropertyManager**, a series of input fields appear which, when correctly configured, allow a thread to be obtained according to the specified requirements. In the first phase, the **Thread Settings** box is clicked, and the circular edge on the model where the thread representation is desired is then selected.

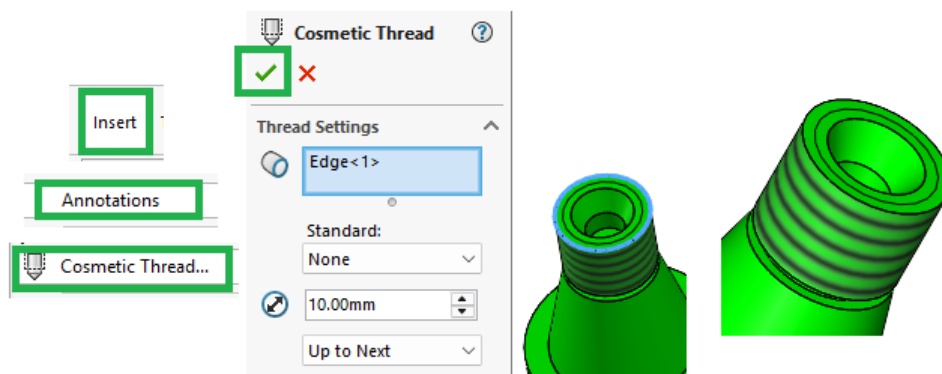
Since the sketch also includes a relief groove, the **Up to Next** option is chosen instead of **Blind**, ensuring that the thread extends to the next surface. The software automatically recognizes the diameter of the selected edge, and the corresponding field displays a value of 20 mm for the diameter. After confirming the settings by clicking **OK**, a circle is created representing the minor diameter of the external thread. This process ensures an accurate and compliant thread representation, meeting the technical requirements of the model.



Since the thread visualization feature is not active by default, the software does not display the configured thread. To enable this option, the general document settings are accessed. In the **Document Properties** menu, the **Shaded Cosmetic Threads** option is checked. After selecting this option, **OK** is clicked to apply the changes and display the thread correctly. This step ensures a clear and realistic visual representation of the thread, facilitating model verification and validation.

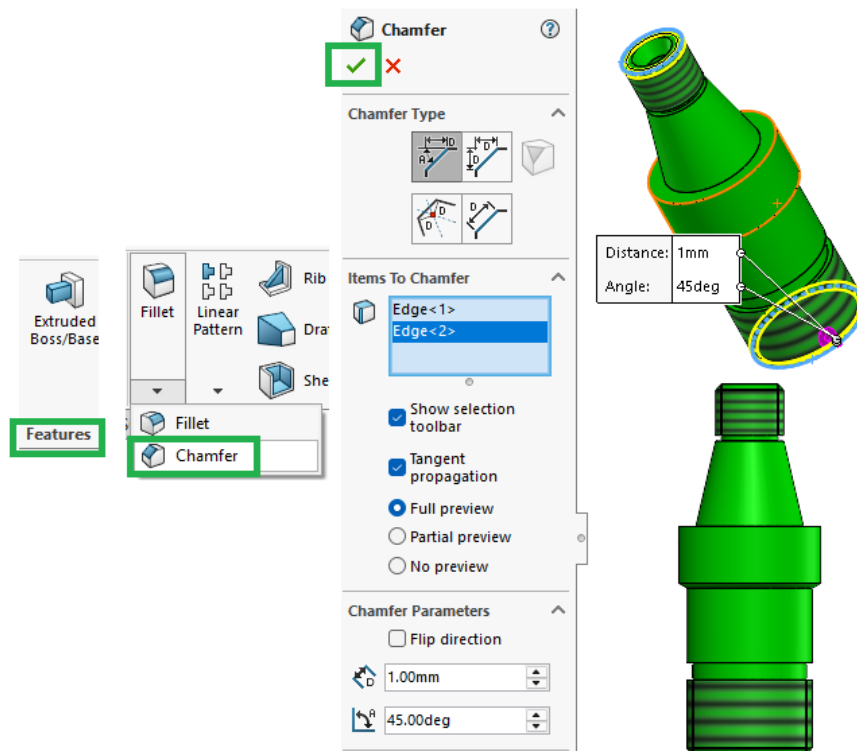


Following the same steps as before, an M12 thread is added on the side where the fuel line is connected. Using the **Cosmetic Thread** command, the corresponding circular edge is selected. Since the visualization option is already active, the thread appears immediately on the model after clicking the **OK** checkmark.



6. Applying a Chamfer to the Thread Start

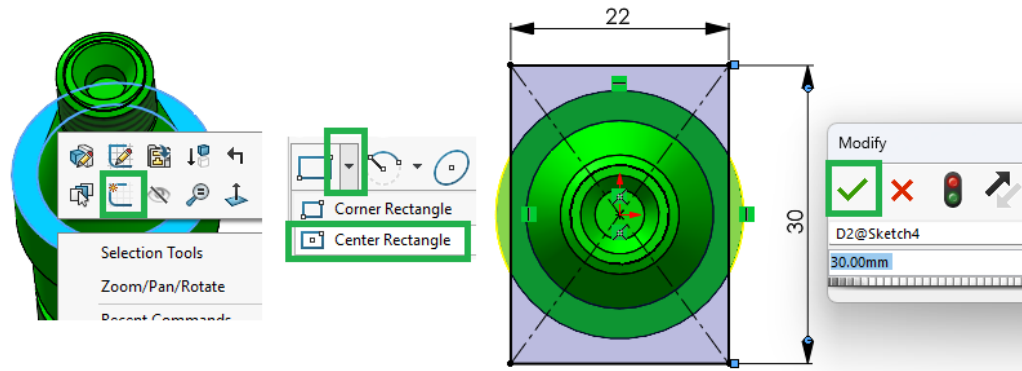
To represent the thread start, the **Chamfer** command is clicked from the **Features** toolbar. In the **PropertyManager**, the two edges corresponding to the thread ends are selected in the **Items to Chamfer** box. In the **Chamfer Parameters** section, a value of 1 mm at an angle of 45 degrees is entered. After completing the settings, the **OK** checkmark is clicked and the chamfer operation is finalized.



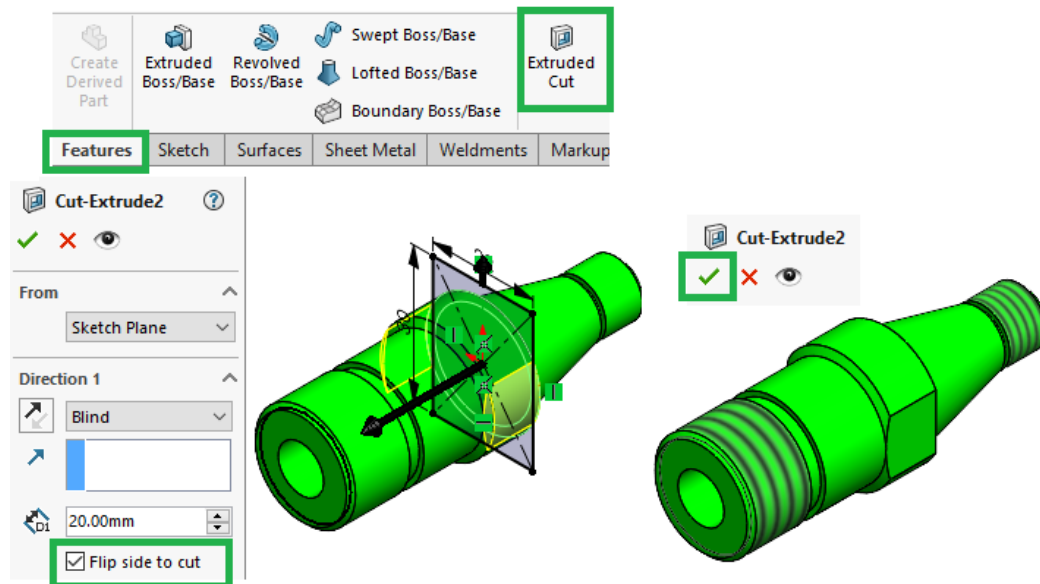
7. Creating the Wrench Flat on the Injector Body

The injector body must also be provided with a wrench flat. To create the two flat surfaces, a 22 mm section is cut from the portion with the maximum diameter of the body.

As a first step, a new sketch is created on the circular surface highlighted in blue in the figure below, by right-clicking and selecting **Sketch**. From the **Sketch** toolbar, the **Center Rectangle** command is accessed to draw a rectangle centered with respect to the origin. Using the **Smart Dimension** command, each side of the rectangle is selected and the dimensions are defined: a width of 22 mm and a length of 30 mm. The values are entered in the dimensioning window and confirmed by clicking the **OK** checkmark.

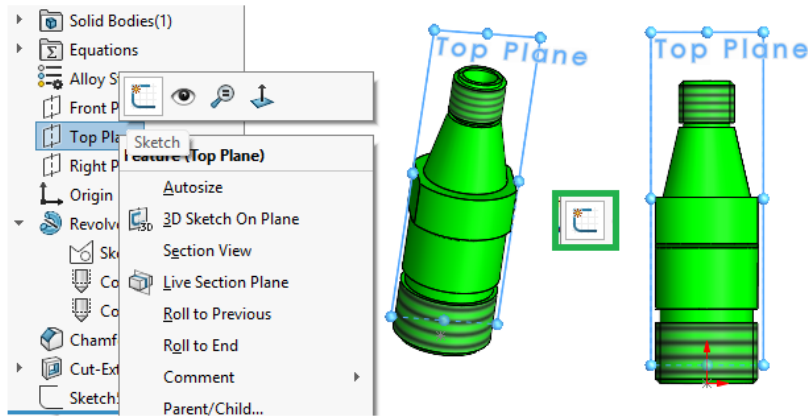


After completing the sketch, the **Extruded Cut** command is accessed from the **Features** toolbar. In the command's **PropertyManager**, the execution direction is set by selecting the **Blind** option with a depth of 20 mm. It is essential that the **Flip Side to Cut** option is activated, as this ensures material removal on the outside of the sketch profile. After configuring all parameters, the command is completed by clicking the **OK** checkmark.

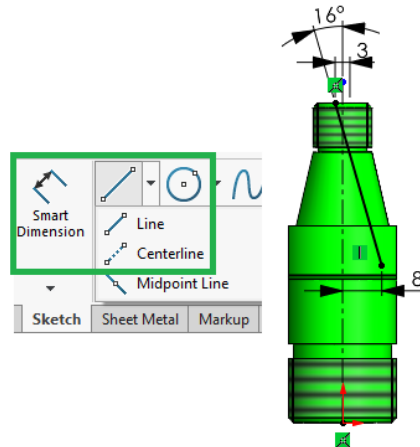


8. Creating the Fuel Inlet Hole

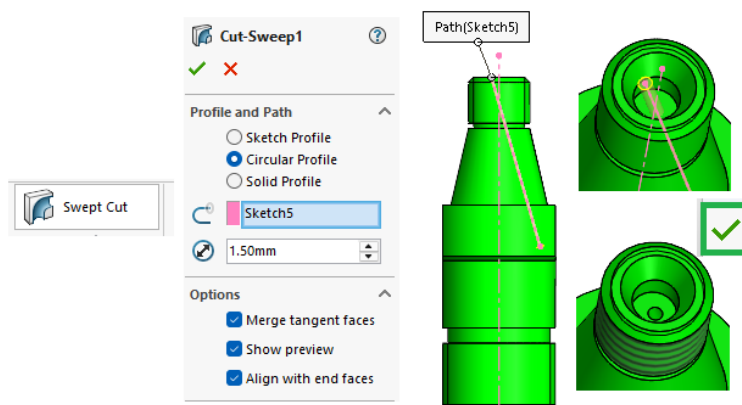
The axial high-pressure bore of the injector body consists of two coaxial sections with different diameters, which from a manufacturing standpoint requires two distinct drilling operations performed successively along the same axis. In SolidWorks, each section is modeled through a separate **Cut-Extrude** operation. For the first section, **Top Plane** is right-clicked, **Sketch** is selected from the context menu to open a new sketch, and the circle corresponding to the first section of the bore is then drawn.



Using the **Centerline** and **Line** commands from the **Sketch** toolbar, a vertical symmetry axis is first drawn. A segment is then drawn from the upper contour, respecting the indicated geometric constraints. The **Smart Dimension** command is used to apply angular and linear constraints, enabling precise definition of distances and angles. This process ensures the geometric accuracy of the sketch and prepares the model for subsequent operations.



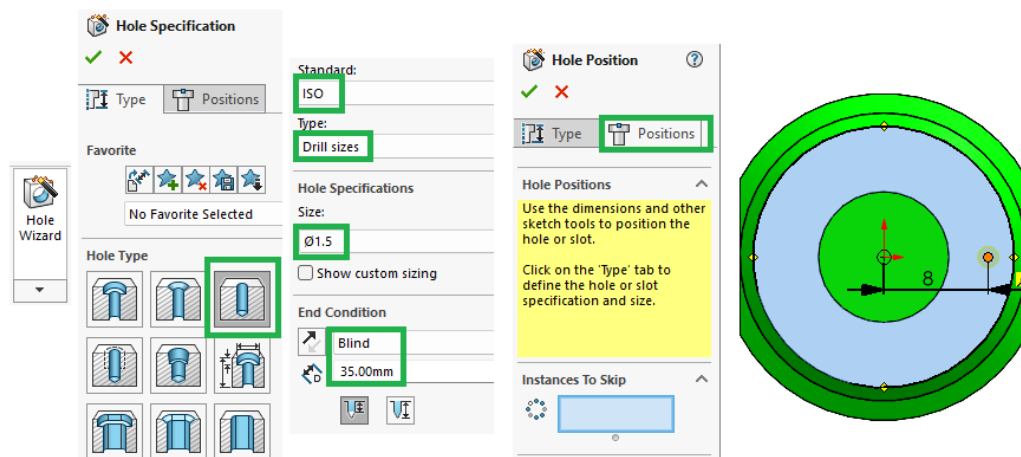
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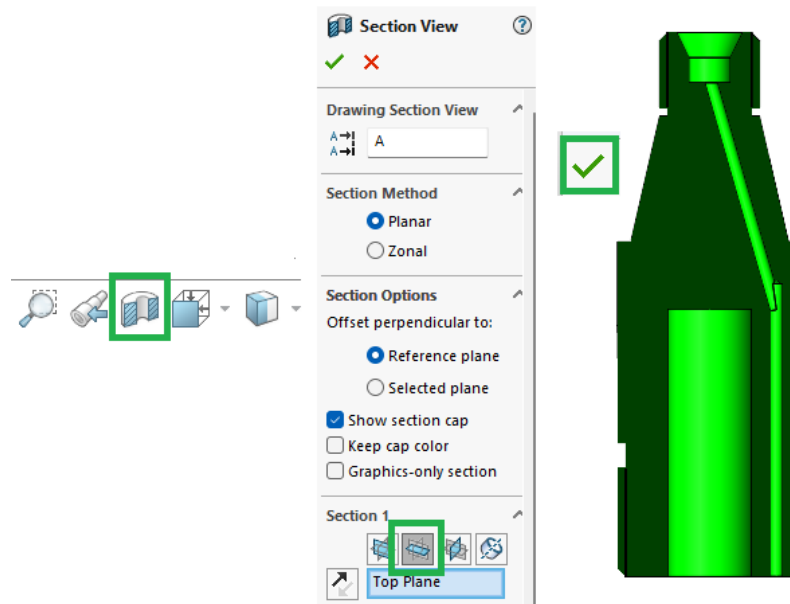
The software allows the bore to be created in a single step; however, from a manufacturing standpoint this is not feasible. The bore is therefore created in stages, respecting the actual fabrication process.

The model is rotated by holding down the middle mouse button until a perspective is obtained that allows visual access to its lower base. After repositioning, the lower base surface is selected with a left click, and the **CTRL+8** key combination is then pressed to orient the view perpendicularly to the selected surface, bringing the model into a convenient working position. To complete the bore entirely, the **Hole Wizard** command is selected from the **Features** toolbar. In the **PropertyManager**, the hole type is set to **Hole**, with the **ISO** standard and **Drill Size** type selected.

In the **Hole Specifications** section, a diameter of 1.5 mm is selected from the available list, and the **End Condition** is set to **Blind** with a depth of 35 mm. In the **Position** menu, the hole position is established by ensuring it is aligned horizontally with the origin, and a left click is used to place the reference point. Using the **Smart Dimension** command, the position of the hole is dimensioned at 8 mm from the origin, after which the **OK** checkmark is clicked to complete the operation.

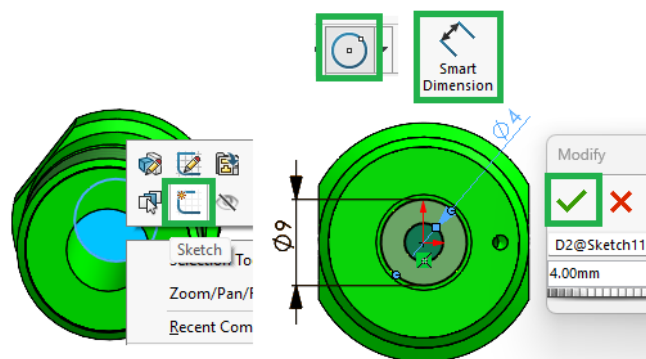


Using the **Section View** command, the bore created through the injector body can be examined. **Section View** is clicked, and **Top Plane** is then selected from the menu that appears as the cutting plane. If the two bores intersect according to the specifications, this confirms that the previous steps were carried out correctly.

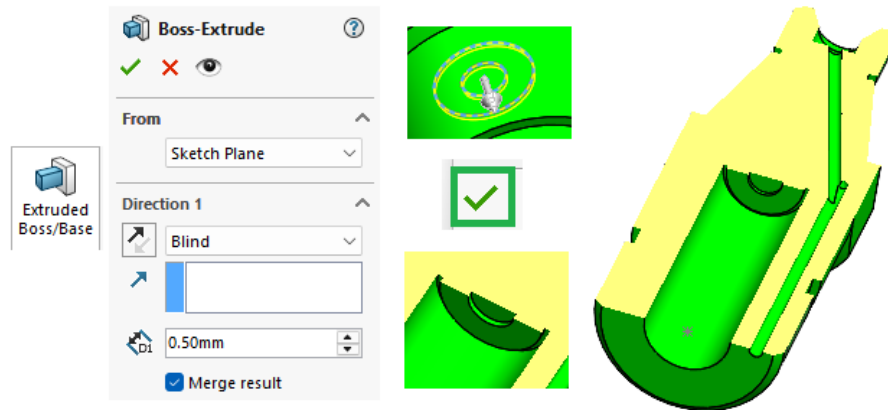


9. Creating the Bearing Surface for the Spring

Another essential element that must be modeled is the bearing surface for the spring. Since this is located inside the 10 mm diameter bore, the model is rotated to a convenient position that allows visual access to the flat inner circular surface, which is then selected with a left click and the sketch is opened using the **Sketch** command from the context menu. Using the **Circle** command, two concentric circles are drawn with diameters of 4 mm and 9 mm respectively. Dimensional accuracy is ensured through the use of the **Smart Dimension** command, which allows the exact definition of the circle diameters. After entering the values, each dimension is confirmed by clicking the **OK** checkmark.

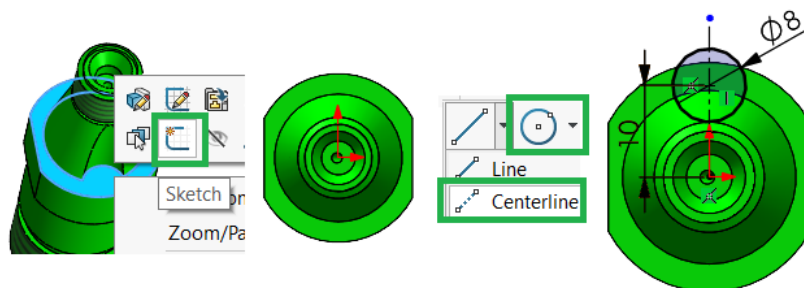


To extrude the sketch, the **Extruded Boss/Base** command is accessed. In the **PropertyManager**, the **Blind** option is selected in the **Direction** section, and a value of 0.5 mm is entered in the dimension field. To ensure the extrusion is integrated into the existing solid, the **Merge Results** option is checked, thus avoiding the creation of a separate solid. Finally, the **OK** checkmark is clicked to complete the extrusion process.

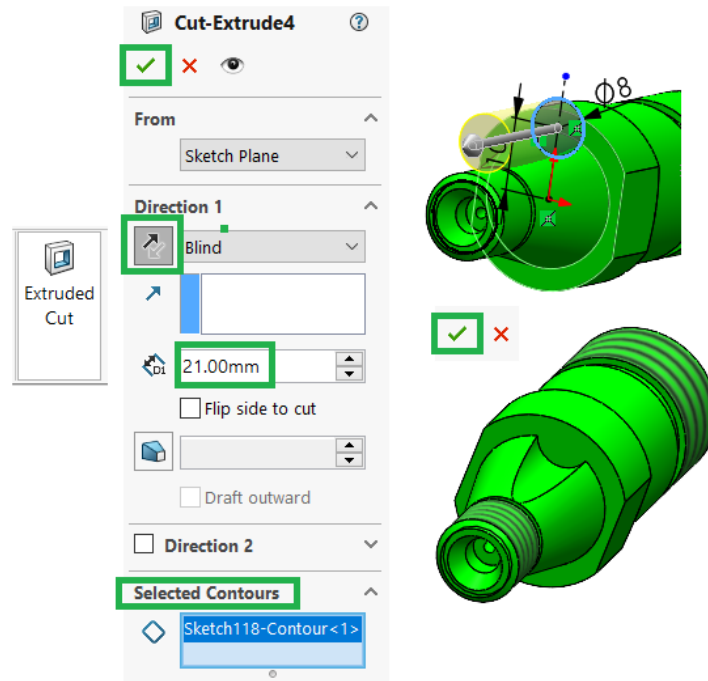


10. Creating the Cutout for the Fuel Return Circuit

To allow the attachment of hoses for the fuel return circuit on the injector body, a cutout must be created. The process begins by selecting the upper flat cylindrical face of the injector according to the blue highlighted area, where a new sketch is created using the **Sketch** command. Within this sketch, a vertical symmetry axis is drawn using the **Centerline** command, after which the cutout profile is outlined using the **Circle** command, respecting the dimensions and geometry required for optimal fastening. The dimensioning of the circle, with a diameter of 8 mm and a specific distance from the origin along the vertical direction, is carried out using the **Smart Dimension** command from the **Sketch** toolbar.

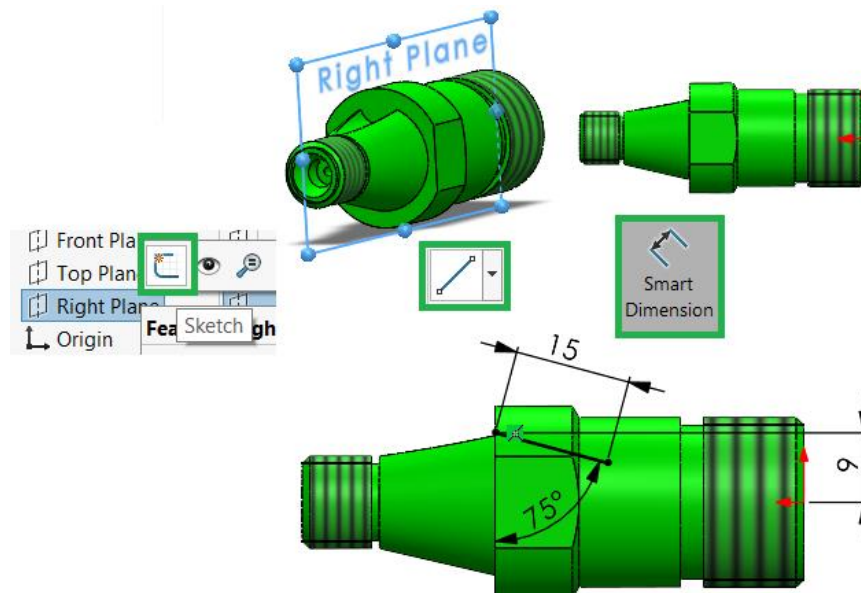


The cutout is created using the **Extruded Cut** command from the **Features** toolbar. After activating this command, the **Blind** option is selected in the **PropertyManager** for the cut direction, and a depth of 16 mm is set. The circle with a diameter of 8 mm is then chosen in the **Selected Contours** field, and the **OK** checkmark is clicked to confirm. It is important to pay attention to the cut direction, which must be oriented toward the outside of the part body. If reversing the cut direction is necessary, this is done using the **Reverse Direction** button, represented by a double arrow, available in the **PropertyManager**.



11. Creating the Sketch for the Fuel Return Bore

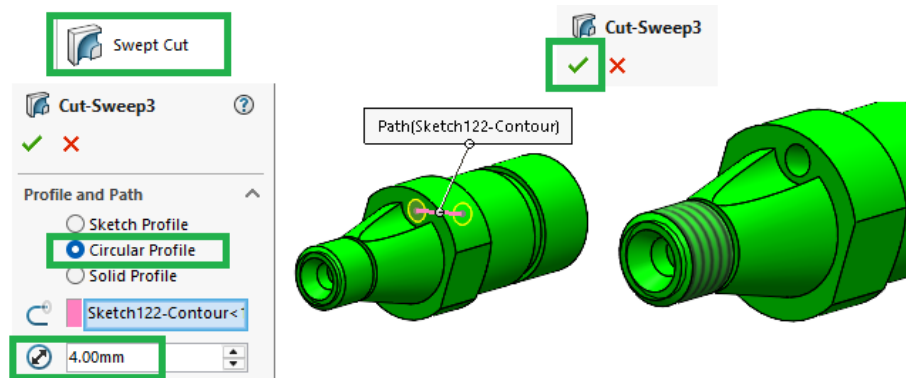
The fuel return bore in the injector body serves to allow excess fuel to reach the return circuit, from where it is directed back to the fuel tank. To create this configuration, the **Right Plane** is selected and a new sketch is opened using the **Sketch** command. Using the **Line** command, a straight line is drawn, positioned and dimensioned according to the figure below. After completing the drawing, the operation is confirmed by clicking the **OK** checkmark.



12. Creating the Fuel Return Bore

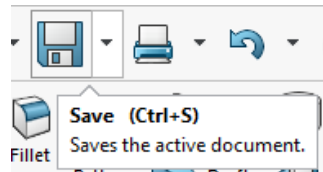
To create the fuel return bore, the **Swept Cut** command is used from the **Features** toolbar. In the configuration window within the **PropertyManager**, the **Circular Profile** option is selected

in the **Profile and Path** section, and the line from the previous sketch is then chosen as the path. The cut diameter is set to 4 mm, and the operation is completed by clicking the **OK** checkmark.



13. Saving the Document

From the menu bar, the model is saved to the predefined directory using the **Save** button.



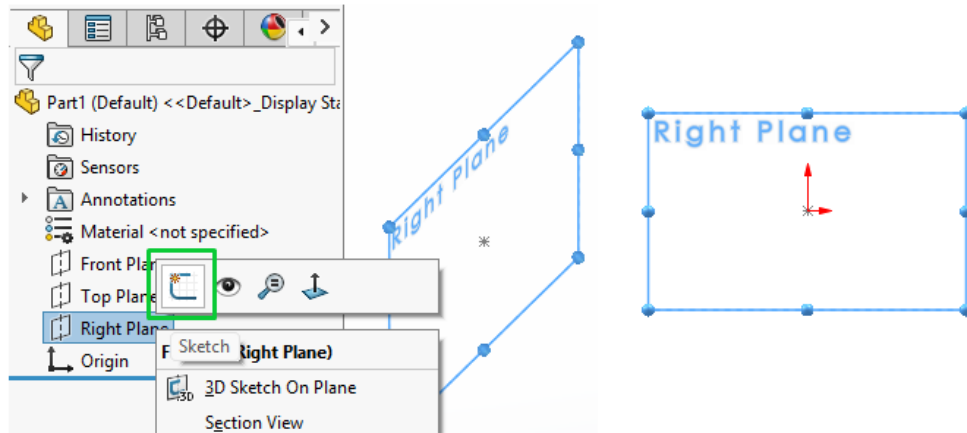
3.3.2. Modeling the Nozzle Cap

1. Creating a New Part File

The first step in the part modeling process consists of opening the SolidWorks application and selecting **File** → **New** → **Part** to create a new part file.

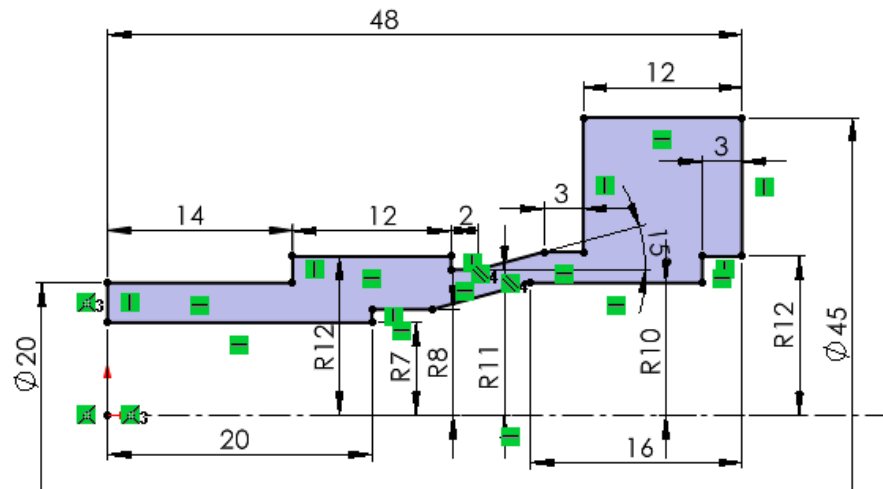
2. Sketching the Base Profile

To begin modeling the cap, the **Right Plane** is selected as the reference plane by right-clicking on it, and sketch mode is opened using the **Sketch** command. To ensure dimensional accuracy, the **Smart Dimension** command is used.



3. Creating and Dimensioning the Profile of the Nozzle Cap

In the first phase, using the commands from the **Sketch** toolbar, the **Centerline** command is used to draw a symmetry axis from the origin along the horizontal. This axis serves as a reference element for the alignment and symmetric construction of the profile. The outer contour of the nozzle cap profile is then drawn, paying close attention to the geometric details shown in the figure below.

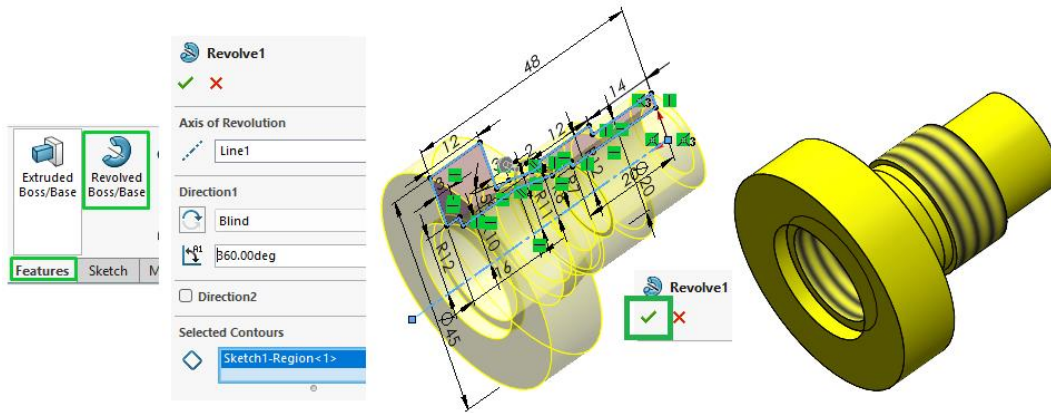


4. Generating the Body by Revolution

After completing the sketch, the **Revolve Boss/Base** command is accessed from the **Features** toolbar. In the **PropertyManager**, the **Axis of Revolution** box is clicked, and the symmetry axis around which the rotation is to be performed is then selected. In the **Direction** window, the **Blind** option is chosen, and the rotation angle is set to 360 degrees.

In the **Selected Contours** section, the sketch created in the previous step is selected. The software provides an instant preview of the volume to be generated, displayed in a semi-transparent yellow color. After confirming the settings by clicking the green **OK** button, the 3D volume is generated automatically, producing the nozzle cap body in its final form. This process

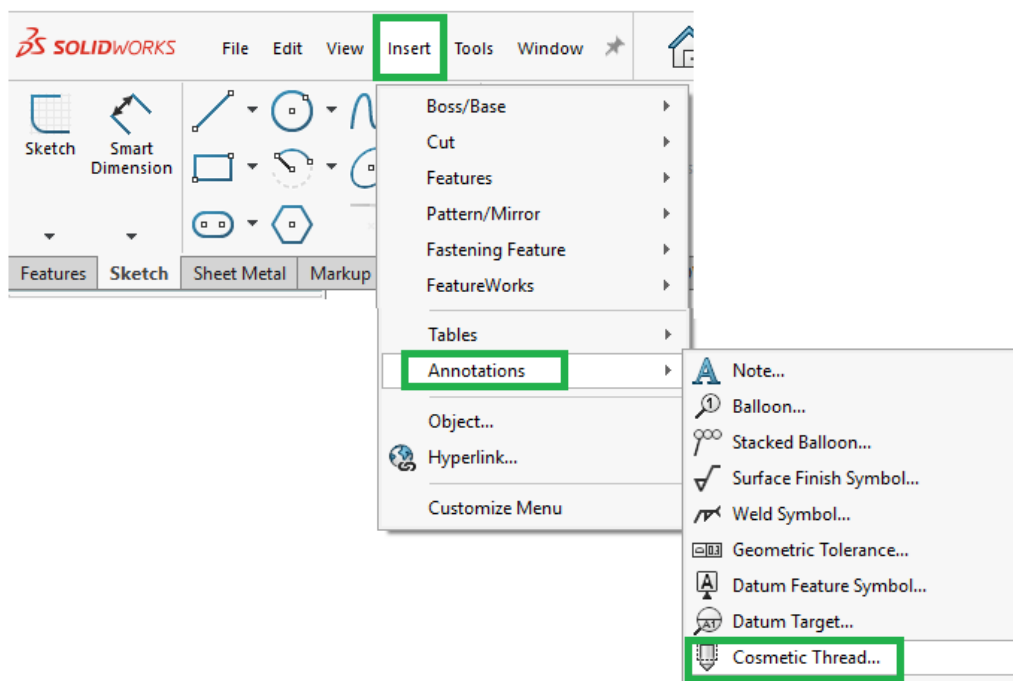
highlights the efficiency and precision of the **Revolve** command in transforming a 2D profile into a three-dimensional model.



5. Representing Threads Using the Cosmetic Thread Command

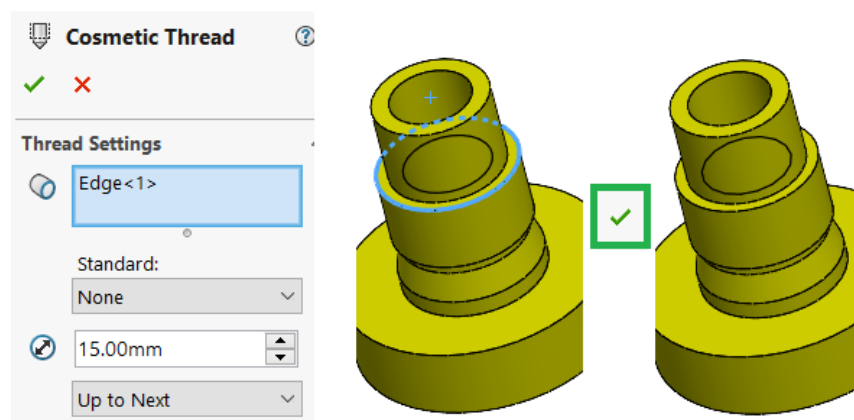
Since components such as the nozzle cap and fuel line are mounted on the injector body, it is necessary to represent threads on both ends. For this purpose, the **Cosmetic Thread** option is used, which allows the visual representation of the thread without geometrically modifying the 3D model.

From the menu bar, the **Insert** section is accessed, where the **Annotations** submenu is opened. The **Cosmetic Thread** command is then selected. This functionality adds a symbolic representation of the thread, complying with technical standards and specifications, without increasing the complexity of the model.

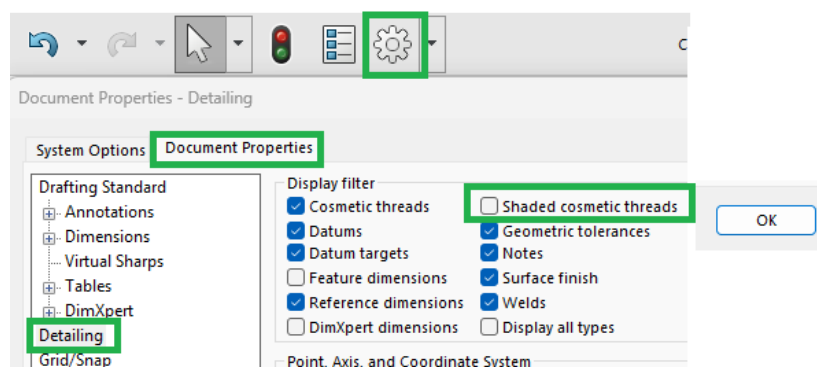


The **Cosmetic Thread** command is launched with a left click. In the **PropertyManager**, a series of input fields appear which, when correctly configured, allow a thread to be obtained according to the specified requirements. In the first phase, the **Thread Settings** box is clicked, and the circular edge on the model where the thread representation is desired is then selected.

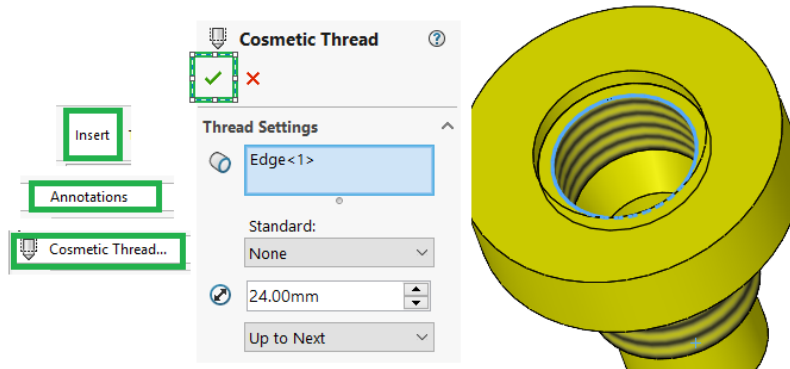
Since the sketch also includes a relief groove, the **Up to Next** option is chosen instead of **Blind**, ensuring that the thread extends to the next surface. The software automatically recognizes the diameter of the selected edge, and the corresponding field displays a value of 15 mm for the diameter. After confirming the settings by clicking the **OK** checkmark, a circle is created representing the minor diameter of the external thread. This process ensures an accurate and compliant thread representation, meeting the technical requirements of the model.



Since the thread visualization feature is not active by default, the software does not display the configured thread. To enable this option, the general document settings are accessed. In the **Document Properties** menu, the **Shaded Cosmetic Threads** option is checked. After selecting this option, **OK** is clicked to apply the changes and display the thread correctly. This step ensures a clear and realistic visual representation of the thread, facilitating model verification and validation.

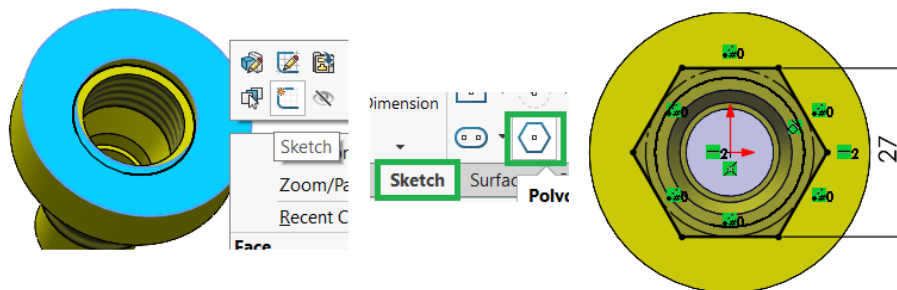


Following the same steps as before, an internal M24 thread is added. Using the **Cosmetic Thread** command, the corresponding circular edge is selected. Since the visualization option is already active, the thread appears immediately on the model after clicking the **OK** checkmark.



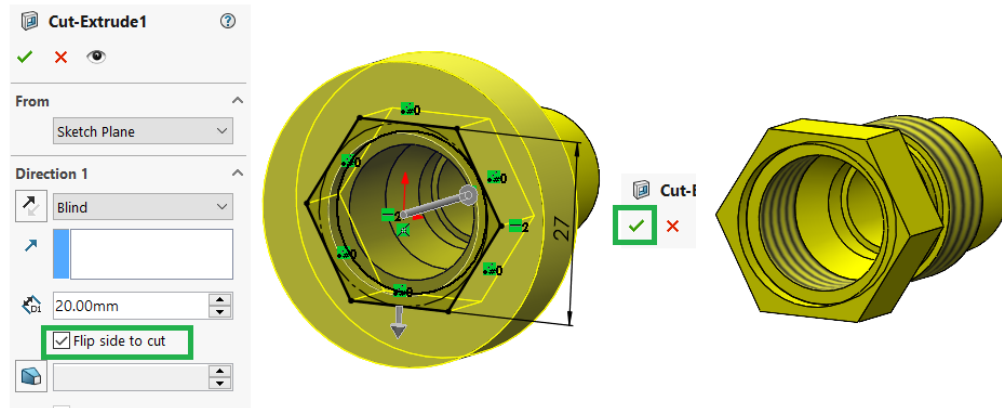
6. Creating the Hexagon Sketch for the Wrench Opening

The upper flat surface is selected as shown in the figure below, highlighted in blue, to create a new sketch. To ensure an adequate wrench opening required for tightening or loosening the nozzle cap, the **Polygon** command is used from the **Sketch** toolbar. After activating this command, the origin of the hexagon is positioned at the center of the model. The **Smart Dimension** command is then used to set its dimension to 27 mm, ensuring conformity with the figure below.



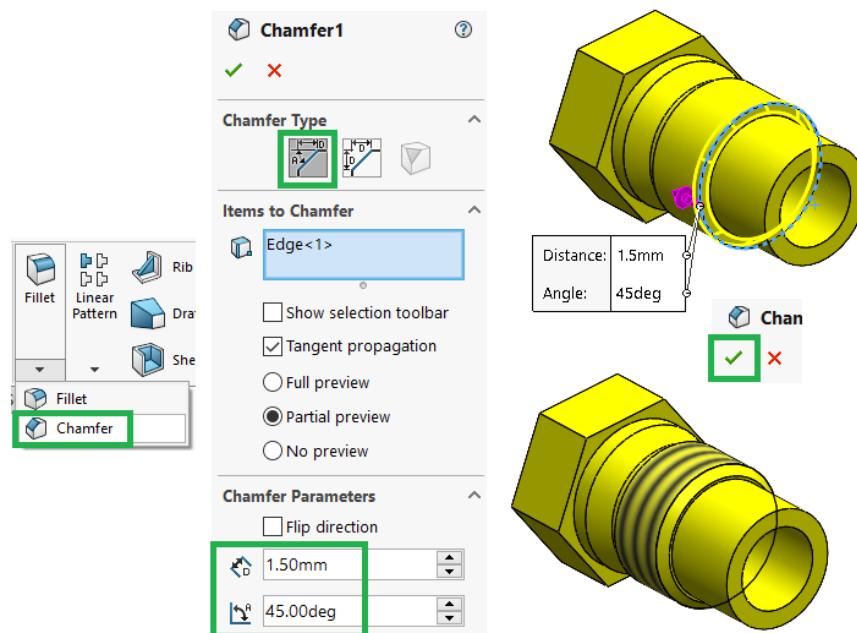
7. Creating the Hexagonal Cutout

To create the necessary hexagon, the **Extruded Cut** command is used. After activating it, the **Blind** option is selected in the direction section of the **PropertyManager**, and the cut depth is set to 20 mm. To ensure the correct orientation of the cutout and generate the hexagon on the model, the **Flip Side to Cut** option is activated.



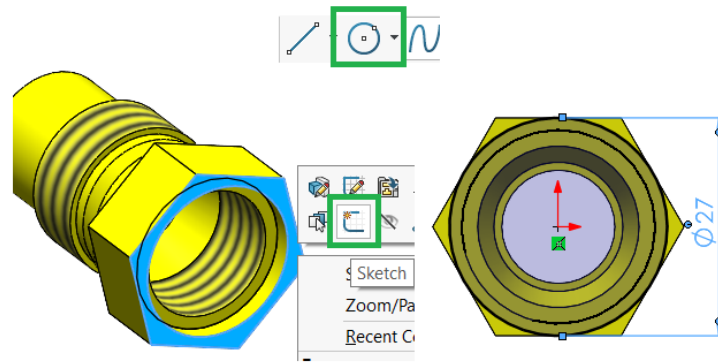
8. Applying a Chamfer to the External Thread Start

To create a chamfer at the start of the external thread, the **Chamfer** command is used from the **Features** toolbar. After activating the command, the starting edge of the external thread is selected. In the **Chamfer Parameters** section, a value of 1.5 mm at an angle of 45 degrees is set, ensuring a correct transition that meets the technical requirements.

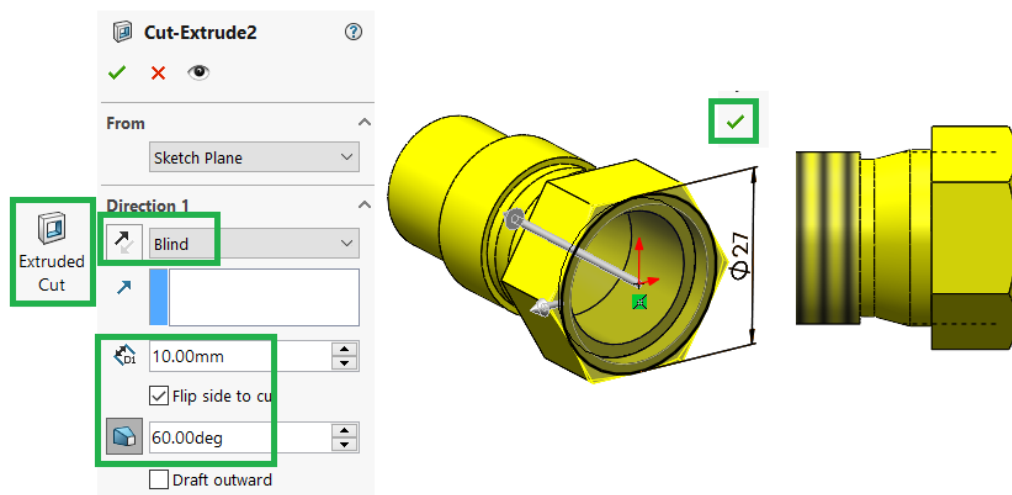


9. Applying a Chamfer to the Hexagon Faces

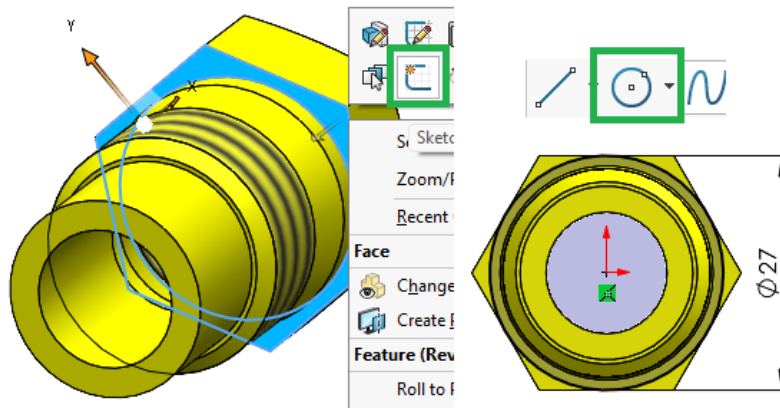
To create chamfers on both faces of the hexagon, the **Chamfer** command cannot be used, as it operates exclusively on linear or circular edges of a solid and is unable to generate a uniform chamfer along the hexagonal contour with the desired angle and depth. In this sketch, a circle with a diameter of 27 mm is drawn, which will serve as the basis for creating the chamfer according to the technical specifications.



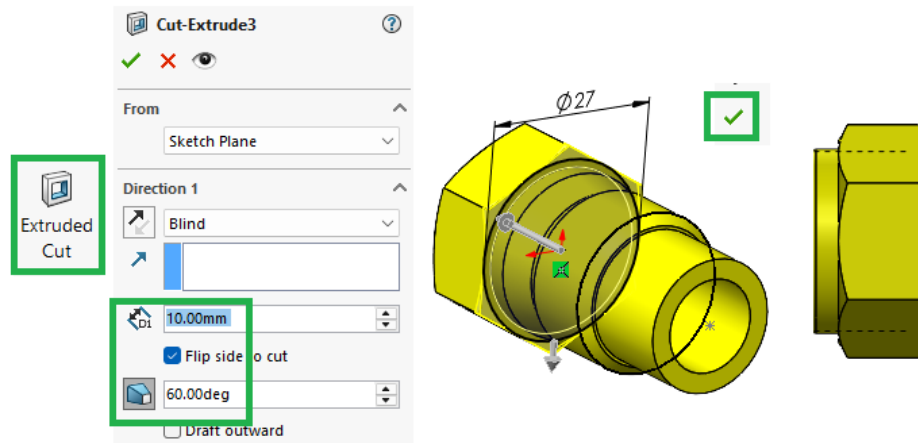
To perform the chamfer, the **Extruded Cut** command is used. After activating it, the cut depth is set to 10 mm in the direction section of the **PropertyManager**. The **Flip Side to Cut** option is then activated to reverse the cut direction, and the **Draft** option is selected, where an angle of 60 degrees is entered. After correctly configuring all parameters, the operation is confirmed by clicking the **OK** checkmark.



Following the same approach as the previous step, a new sketch is created on the other face of the hexagon by right-clicking, after which the **Circle** command is selected from the toolbar. A click is made at the origin to begin the circle, then a second click is made at an approximate diameter corresponding to the wrench opening. The **Smart Dimension** command is activated from the **Sketch** toolbar, and the circle is clicked to enter a diameter of 27 mm in the **Modify** window.

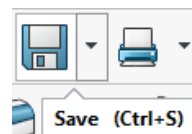


To perform the chamfer, the **Extruded Cut** command is used. After activating it, the cut depth is set to 10 mm in the direction section of the **PropertyManager**. The **Flip Side to Cut** option is then activated to reverse the cut direction, and the **Draft** option is selected, where an angle of 60 degrees is entered. After correctly configuring all parameters, the operation is confirmed by clicking the **OK** checkmark.



10. Saving the Document

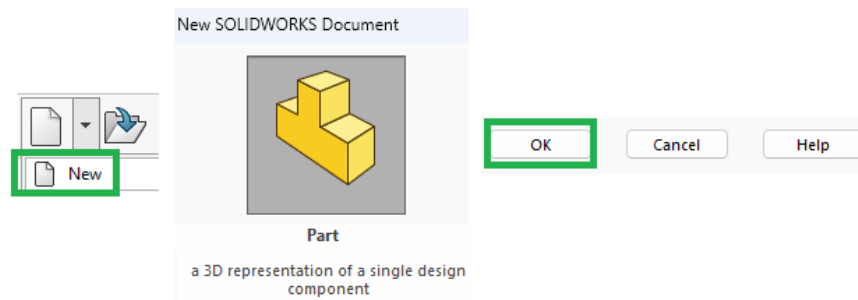
From the menu bar, the model is saved using the **Save** command, assigning it the name *capac_pulverizator*. After saving, the current window is closed, and a new **Part** file is opened via **File** → **New** → **Part** to begin modeling the next component.



3.3.3. Modeling the Helical Compression Spring

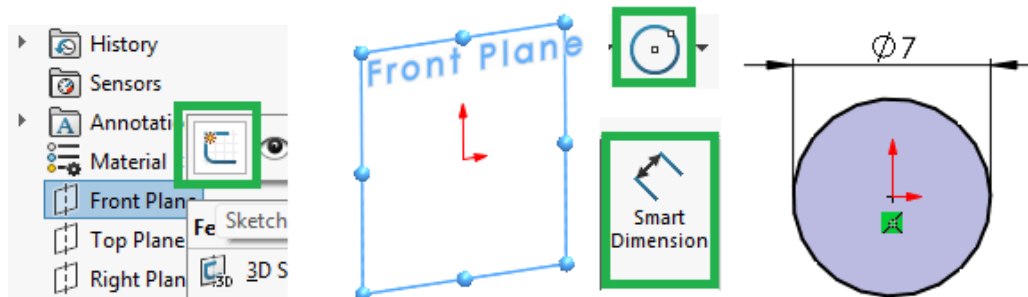
1. Creating a New Part File

To initiate the 3D modeling process, the main menu bar is accessed and the **New** command is selected. In the dialog window, the **Part** category is chosen, and the selection is confirmed by clicking the **OK** checkmark.



2. Creating the Base Sketch

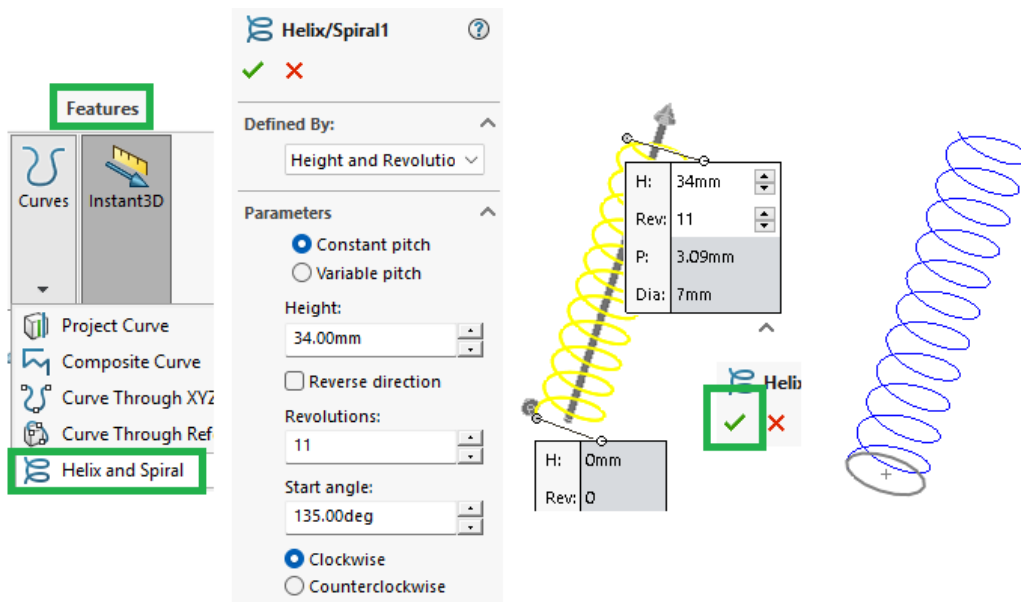
To initiate the modeling process, a new sketch is created by activating the **Sketch** command. The **Top Plane** is then selected as the working plane. Within this sketch, a circle is drawn that defines the diameter of the helical spring. Its dimension is set using the **Smart Dimension** command, ensuring conformity with the technical specifications of the fuel injector.



3. Generating the Helical Path

To define the helical path of the spring, the **Helix** command is used from the **Features** toolbar. The previously created circle is selected, and the essential parameters are configured in the **PropertyManager**, including pitch, number of coils, and helix direction. Depending on the technical requirements, either a constant or variable pitch can be chosen.

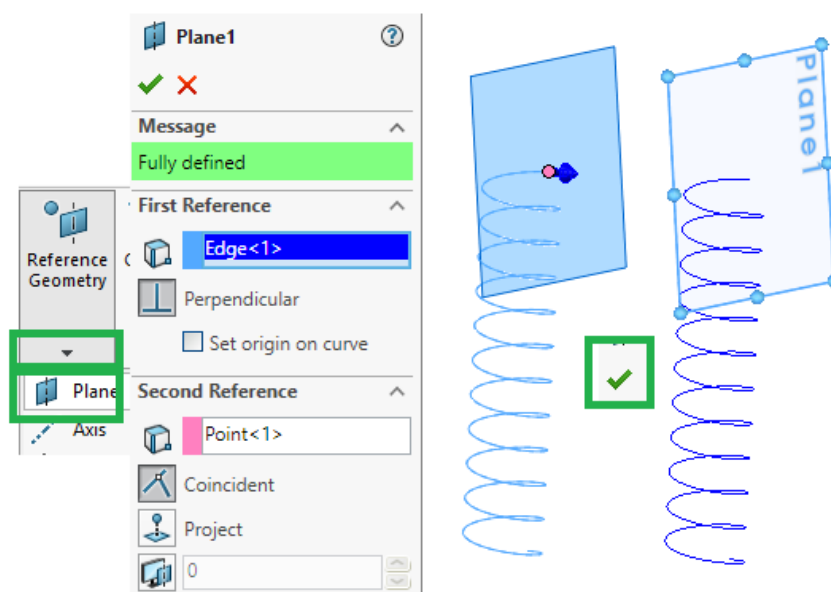
In this case, the constant pitch option is selected by activating the **Pitch** option. A height of 34 mm is then set, and a value of 11 mm is entered in the **Revolution** box. After configuring these parameters, the **OK** checkmark is clicked, generating the helix.



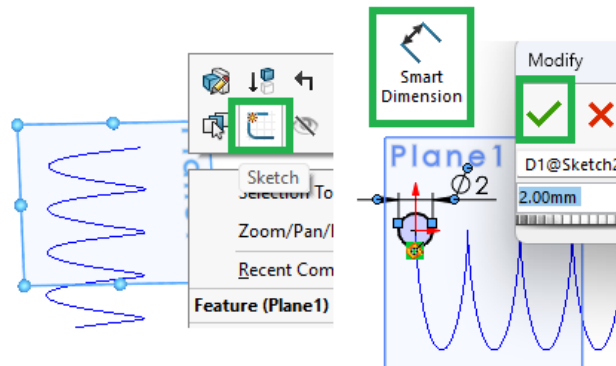
4. Creating the Cross-Section and Generating the Spring

After defining the path, a new sketch is created on a perpendicular plane using the **Sketch** command. To generate a plane perpendicular to the end of the helix, the **Plane** command is accessed from the toolbar, and the arrow in the **Reference Geometry** section is clicked to display the available subcommands.

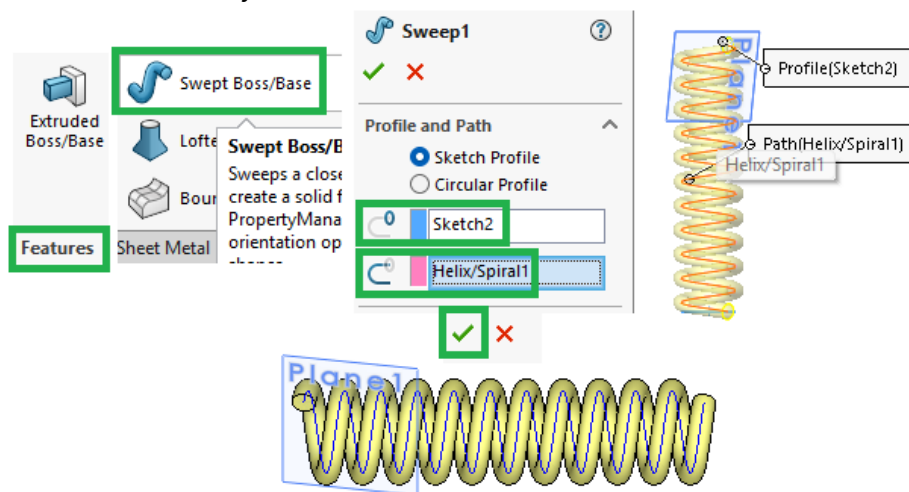
In the **PropertyManager**, the **First Reference** window is clicked and the helix is selected. The corresponding box is then clicked to set the second reference, and the point located at the upper end of the helix is selected. The **Perpendicular** and **Coincident** relations must be active to ensure correct alignment of the plane with the helical path.



The newly created plane is right-clicked and a sketch is initiated using the **Sketch** command. Within this sketch, a circle is drawn that will define the cross-sectional profile of the spring wire. Its diameter of 2 mm is set using the **Smart Dimension** command, ensuring conformity with the technical requirements of the model.

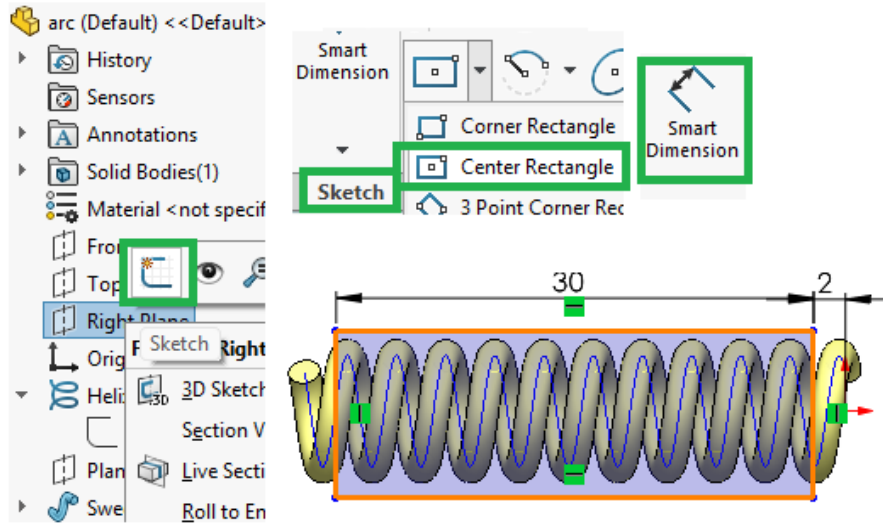


To generate the helical spring, the **Sweep** command is used from the **Features** toolbar. In the **PropertyManager**, the previously defined circle is selected as the cross-sectional profile and the created helical path is selected as the trajectory. After correctly configuring the parameters and confirming the settings by clicking the **OK** checkmark, the three-dimensional model of the spring is generated automatically.



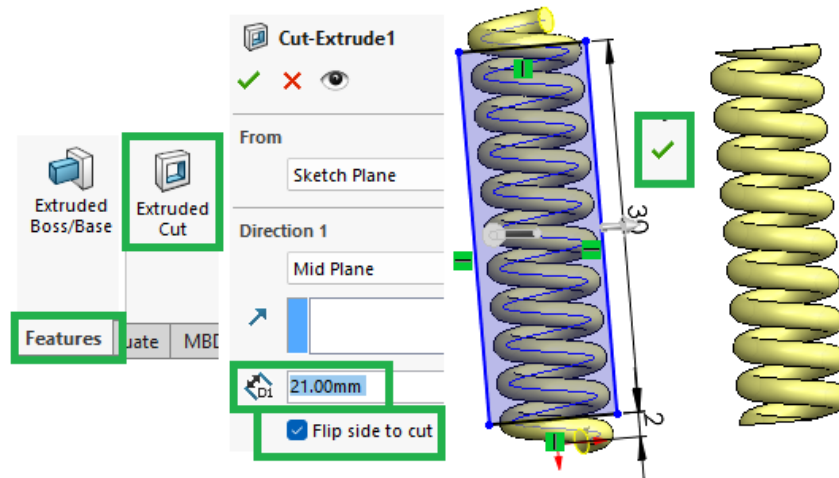
5. Trimming the Spring Ends

To perform the trimming operation, a new sketch is created on an appropriate plane such as the **Right Plane**. Within this sketch, the necessary geometry is outlined using the **Rectangle** command from the **Sketch** toolbar. This step allows the area to be removed to be defined, ensuring precision and conformity with the technical requirements of the model.



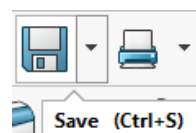
To finalize the model, the ends of the spring must be removed to ensure an optimal contact surface. This is accomplished using the **Extruded Cut** command. In the **PropertyManager**, **Mid Plane** is selected for the direction, and the cut depth is set to 21 mm. An essential aspect of this step is checking the **Flip Side to Cut** option, which ensures correct trimming of the spring ends.

After confirming the operation, the model of the helical compression spring intended for the fuel injector is complete, meeting the technical and functional requirements imposed by the design.



6. Saving the Document

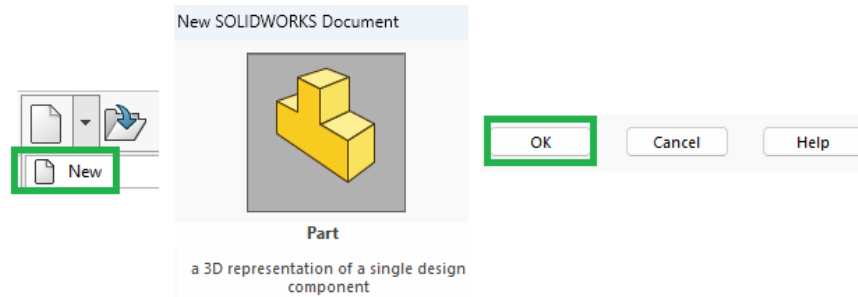
The model is saved using the **Save** command from the menu bar, assigning it the name *arc* in the dialog window. After saving, the current window is closed.



3.3.4. Modeling the Spacer Washer

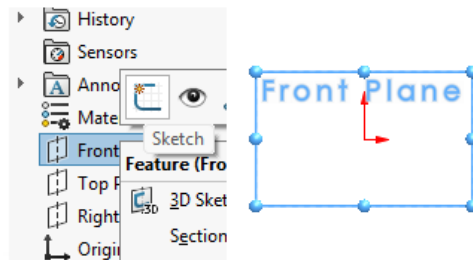
1. Creating a New Part File

To begin modeling, a new part file is created using the **New Part** option from the menu bar. This step establishes an appropriate working environment for the design process.

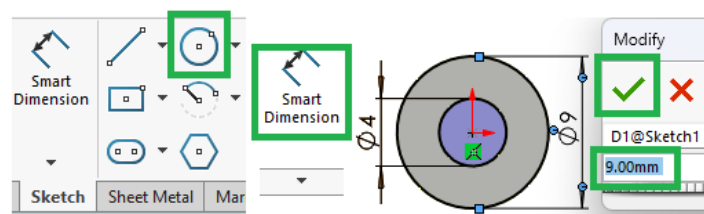


2. Defining the Initial Contour

An appropriate working plane is selected, **Front Plane**, and a new sketch is created using the **Sketch** command.

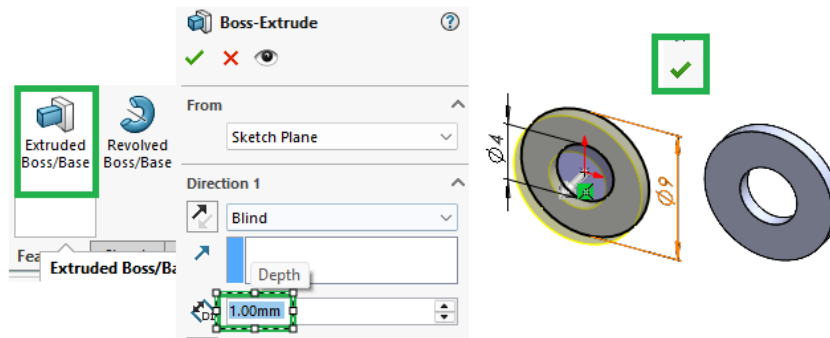


Within this sketch, two concentric circles are drawn to define the outer and inner contours of the spacer washer, with diameters of 4 mm and 9 mm respectively. Their dimensions are set using the **Smart Dimension** command from the **Sketch** toolbar.



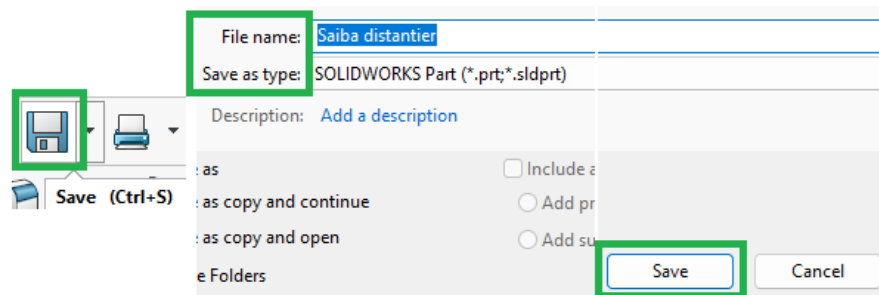
3. Generating the Volume

To convert the sketch into a three-dimensional model, the **Extruded Boss/Base** command is used from the **Features** toolbar. In the **PropertyManager**, the extrusion direction is configured and a thickness of 1 mm is entered. After confirming the settings, the 3D model of the washer is generated.



4. Saving the Model

To complete the process, the model is saved using the **Save** button from the menu bar. In the save dialog window, the part is named *Saiba_distantier* and stored in the predefined directory.

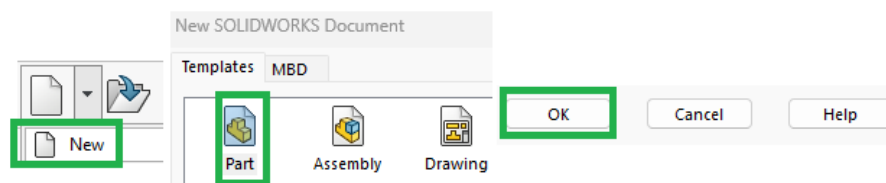


3.3.5. Modeling the Adjustment Washer

Modeling an adjustment washer in SolidWorks involves a series of essential steps, making use of the software's functionality to ensure geometric precision and conformity with technical requirements. The spacer washer maintains a fixed distance between components, preventing direct contact, while the adjustment washer allows fine-tuning of their position. The former ensures alignment, while the latter provides adaptability for precise adjustments.

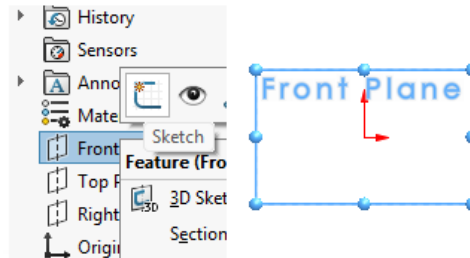
1. Creating a New Part File

To initiate the modeling process, the menu bar is accessed and the **New** command is used. In the selection window, the **Part** option is chosen, and the selection is confirmed by clicking the **OK** checkmark, thereby generating a new file for part modeling.

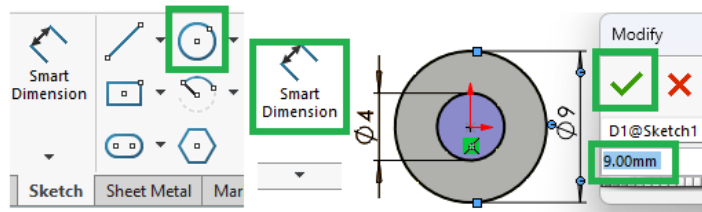


2. Creating a Base Sketch

The **Sketch** command is used from the **Sketch** toolbar, and **Top Plane** is selected as the working plane.

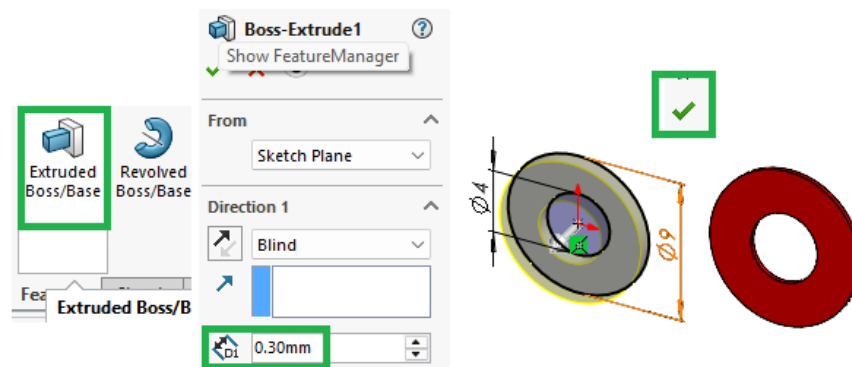


Within this sketch, two concentric circles are drawn using the **Circle** command. The outer and inner diameter dimensions are set using the **Smart Dimension** command, respecting the technical specifications of the injector.



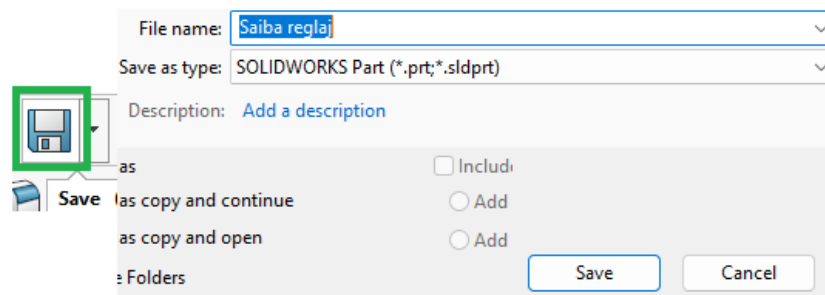
3. Generating the Three-Dimensional Model

To transform the sketch into a 3D model, the **Extruded Boss/Base** command is used from the **Features** toolbar. In the **PropertyManager**, the extrusion direction is set and a thickness of 0.3 mm is entered. After confirming the parameters by clicking the **OK** checkmark, the three-dimensional model of the washer is generated.



4. Saving the Model

To complete the process, the model is saved using the **Save** button from the menu bar. In the save dialog window, the part is named *saiba_de_reglaj*, and the storage location is selected according to the user's preferences.

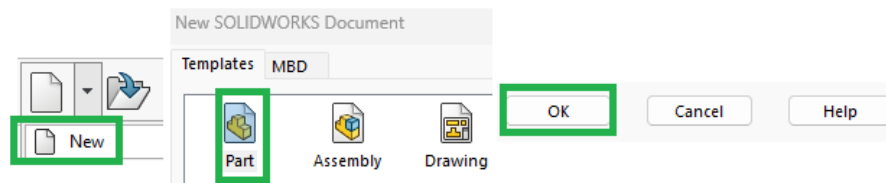


3.3.6. Modeling the Spring Retainer

The spring retainer within a fuel injector serves to ensure uniform distribution of the spring force onto the valve, contributing to the sealing and optimal operation of the injector.

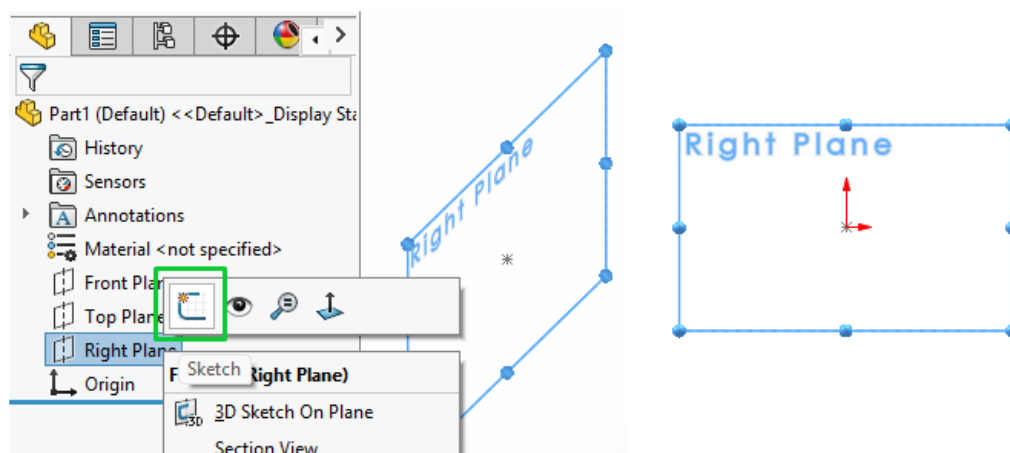
1. Creating a New Part File

To initiate the modeling process, the menu bar is accessed and the **New** command is used. In the selection window, the **Part** option is chosen, and the selection is confirmed by clicking the **OK** checkmark, thereby generating a new file for part modeling.



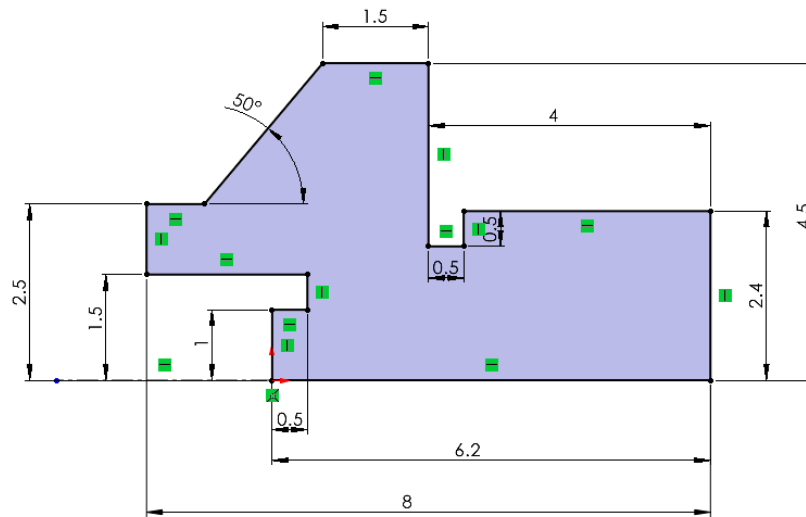
2. Creating the Initial Sketch of the Spring Retainer

To begin modeling the spring retainer, the **Right Plane** is selected as the reference plane by right-clicking on it, and sketch mode is opened using the **Sketch** command.



3. Defining the Geometric Profile of the Spring Retainer

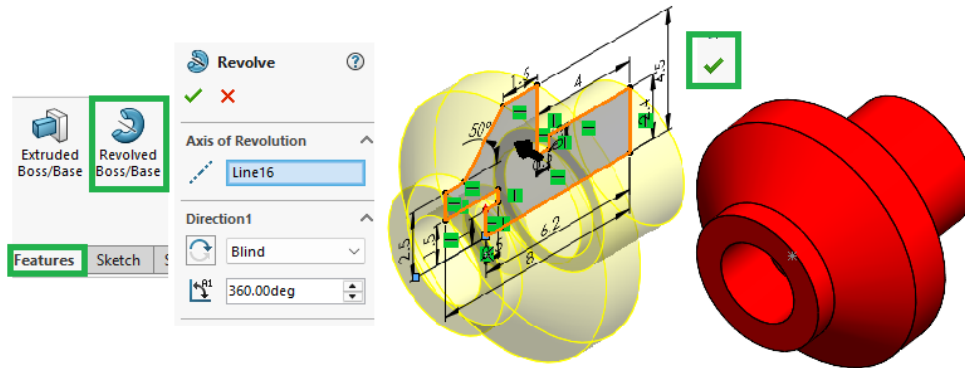
In the first phase, using the commands from the **Sketch** toolbar, the **Centerline** command is used to draw a symmetry axis from the origin along the horizontal. This axis serves as a reference element for the alignment and symmetric construction of the profile. The outer contour of the spring retainer profile is then drawn, paying close attention to geometric details and conformity with the technical specifications. The **Smart Dimension** command is used to ensure dimensional accuracy.



4. Generating the Body by Revolution

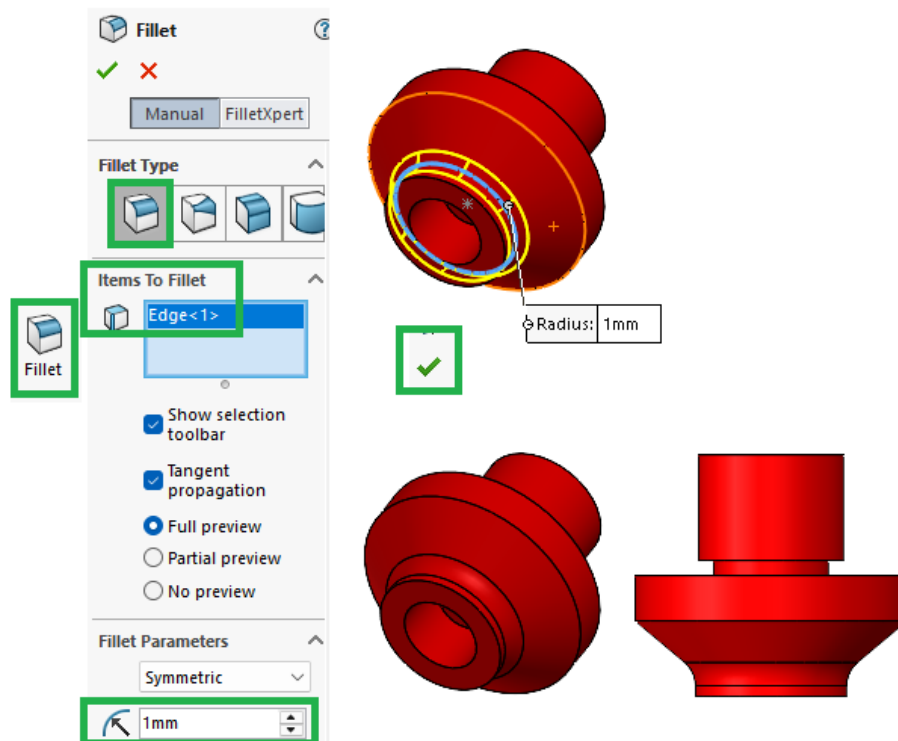
After completing the sketch, the **Revolve Boss/Base** command is accessed from the **Features** toolbar. In the **PropertyManager**, the **Axis of Revolution** box is clicked, and the symmetry axis around which the rotation is to be performed is then selected. In the **Direction** window, the **Blind** option is chosen, and the rotation angle is set to 360 degrees.

In the **Selected Contours** section, the sketch created in the previous step is selected. The software provides an instant preview of the volume to be generated, displayed in a semi-transparent yellow color. After confirming the settings by clicking the green **OK** checkmark, the 3D volume is generated automatically, producing the spring retainer in its final form. This process highlights the efficiency and precision of the **Revolve** command in transforming a 2D profile into a three-dimensional model.



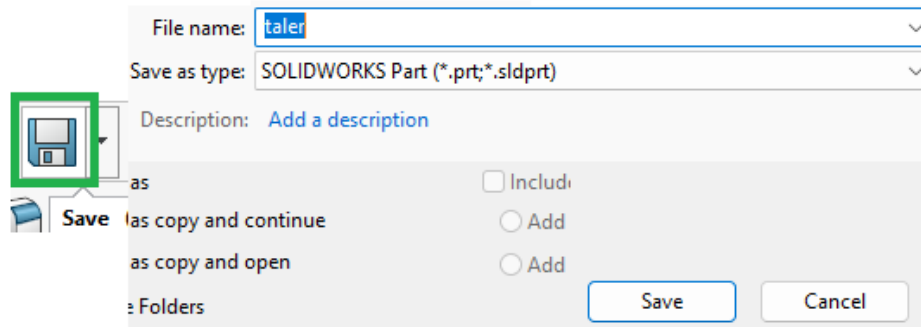
5. Applying Fillets

To optimize the geometry and reduce stress concentrations, the **Fillet** command is used from the **Features** toolbar. After accessing it, the **Constant Size Fillet** type is selected in the **PropertyManager**, and the desired edge for filleting is chosen in the **Items to Fillet** box. A value of 1 mm is then entered in the **Fillet Parameters** section. After confirming the settings by clicking the **OK** checkmark, the fillet radius is generated automatically.



6. Saving the Model

To complete the process, the model is saved using the **Save** button from the menu bar. In the save dialog window, the part is named *taler*, and the storage location is selected according to the user's preferences.

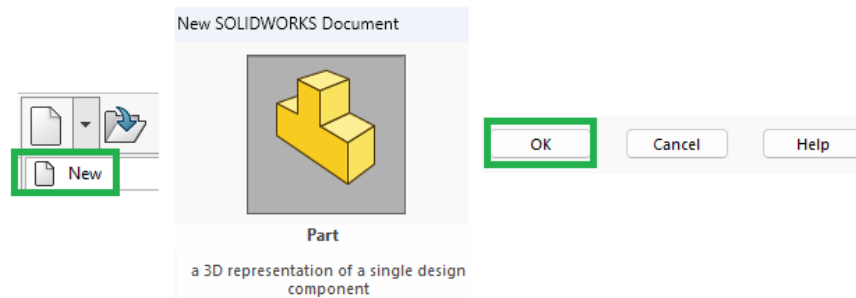


3.3.7. Modeling the Centering Element

The centering element ensures precise positioning of the internal components of the fuel injector, contributing to its optimal operation by preventing unwanted displacements.

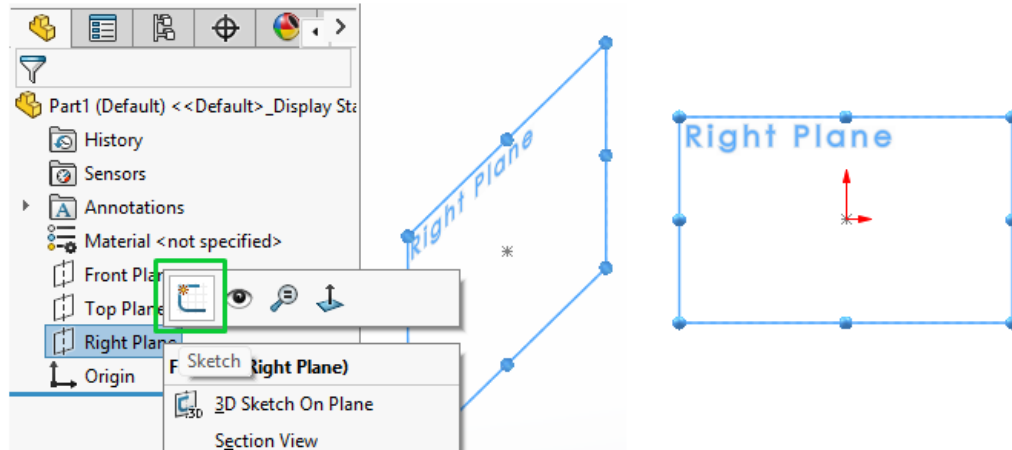
1. Creating a New Part File

To initiate the three-dimensional modeling process, the main menu bar of the design environment is accessed and the **New** command is selected. In the dialog window that appears, the **Part** category is identified and chosen, representing a base document for modeling individual components. This selection is confirmed by clicking the **OK** checkmark, which automatically generates a new working file configured specifically for the parametric design process of the part.



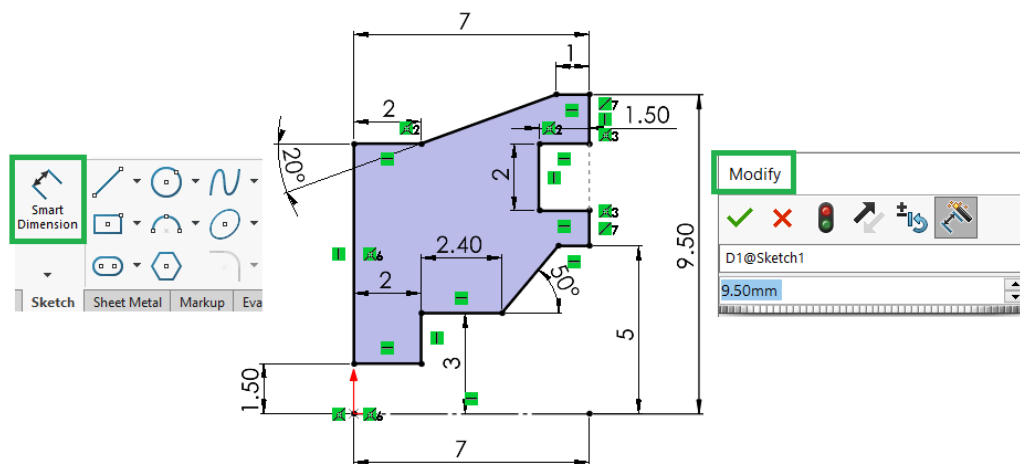
2. Creating the Initial Sketch

To initiate the modeling process, the **Right Plane** is selected as the working plane by right-clicking on it, and sketch mode is then activated using the **Sketch** command from the toolbar.



3. Creating and Dimensioning the Profile of the Centering Element

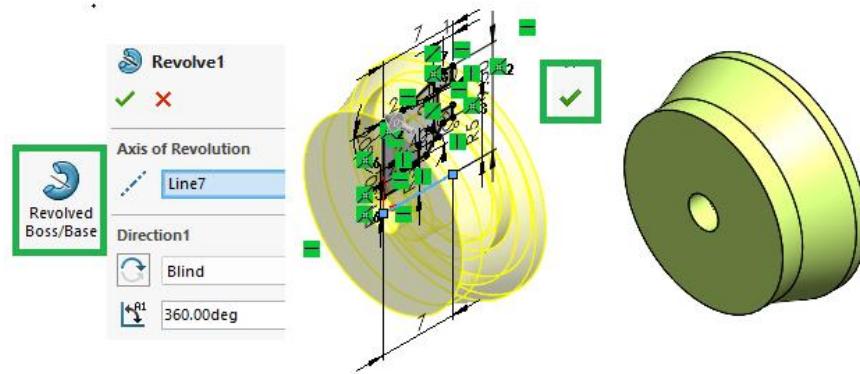
In the first phase, using the commands from the **Sketch** toolbar, a horizontal symmetry axis is drawn from the origin point using the **Centerline** command. This axis serves as a reference element for the alignment and symmetric construction of the profile. The outer contour of the part profile is then drawn according to the figure below.



4. Generating the Centering Element by Revolution

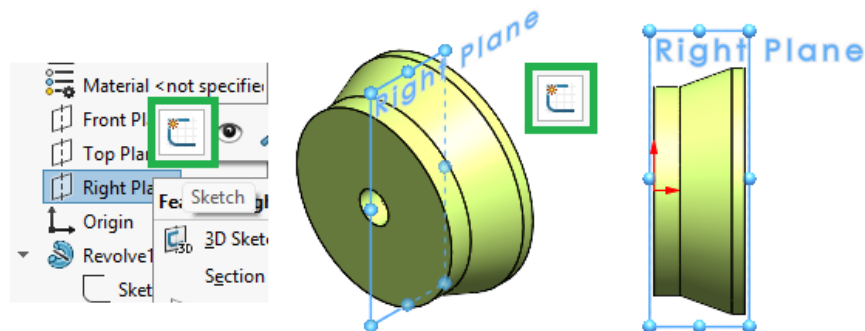
After completing the sketch, the **Revolve Boss/Base** command is accessed from the **Features** toolbar. In the **PropertyManager**, the **Axis of Revolution** box is clicked, and the symmetry axis around which the rotation is to be performed is then selected. In the **Direction** window, the **Blind** option is chosen, and the rotation angle is set to 360 degrees.

If the generated volume is not automatically previewed in yellow, the sketch created in the previous step is manually selected in the **Selected Contours** section. Following this action, the software displays a semi-transparent preview of the volume to be created. After confirming the settings by clicking the **OK** checkmark, the 3D volume is generated automatically, producing the centering element in its final form.

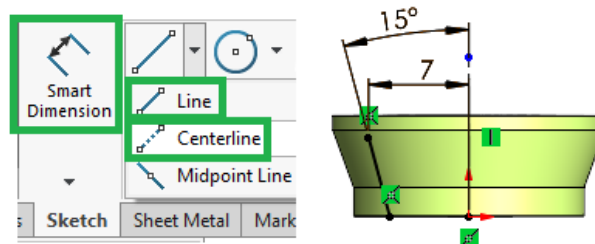


5. Creating the Bores

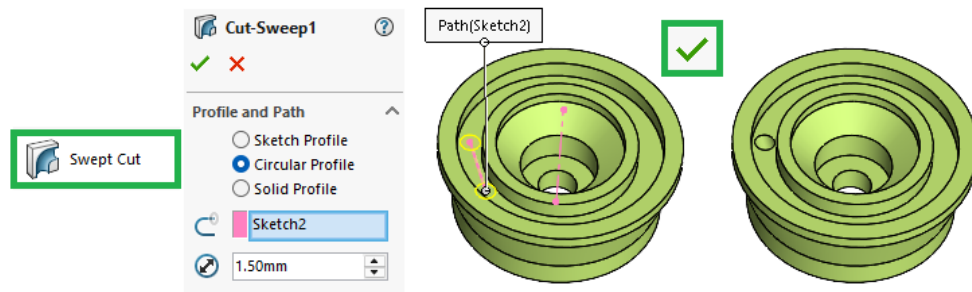
To create the high-pressure bore running through the part, **Top Plane** is right-clicked and the **Sketch** command is used to initiate a new sketch.



In the first phase, the **Centerline** and **Line** commands from the **Sketch** toolbar are used to draw a vertical symmetry axis. A segment is then drawn on the upper contour, respecting the imposed geometric constraints. The **Smart Dimension** command is used to apply angular and linear constraints, enabling precise definition of distances and angles..



To create a circular cross-section channel of 1.5 mm, the **Swept Cut** command is used from the **Features** toolbar. In the **PropertyManager**, the **Circular Profile** option is activated in the **Profile and Path** configuration menu. In the **Path Sketch** window, sketch no. 2 is selected, which contains the lines drawn in the previous step. A value of 1.5 mm is entered for the circular profile diameter, and the operation is completed by confirming the settings with the **OK** checkmark.

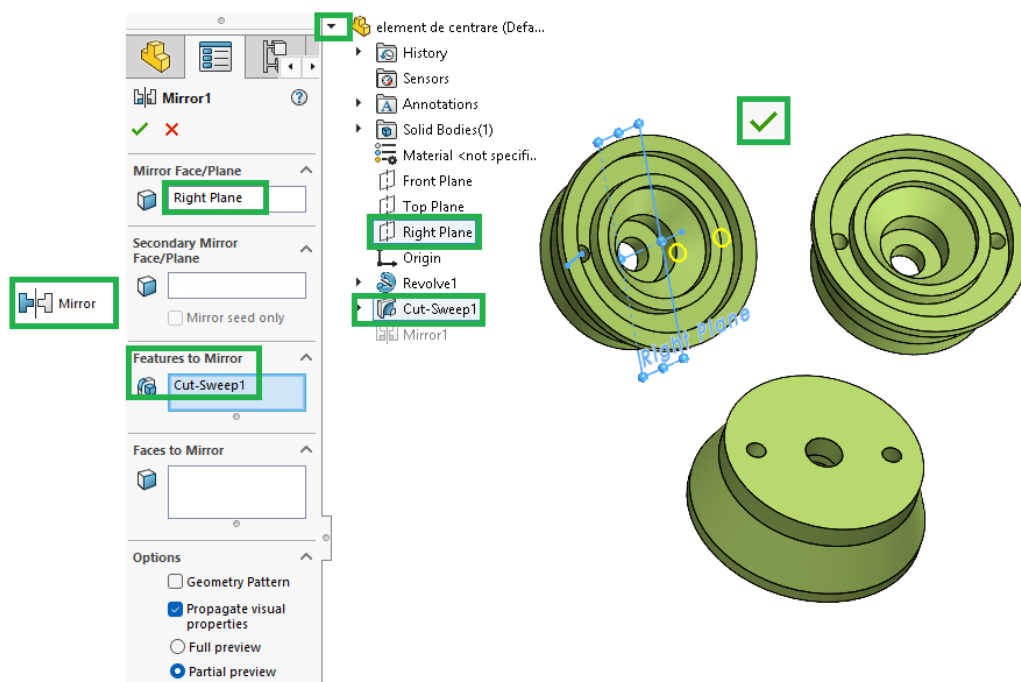


6. Mirroring the Bore

To mirror the bore created in the previous step, the **Mirror** command is used from the **Features** toolbar. After activating the command, the options necessary for correctly configuring the mirroring operation are displayed in the **PropertyManager**.

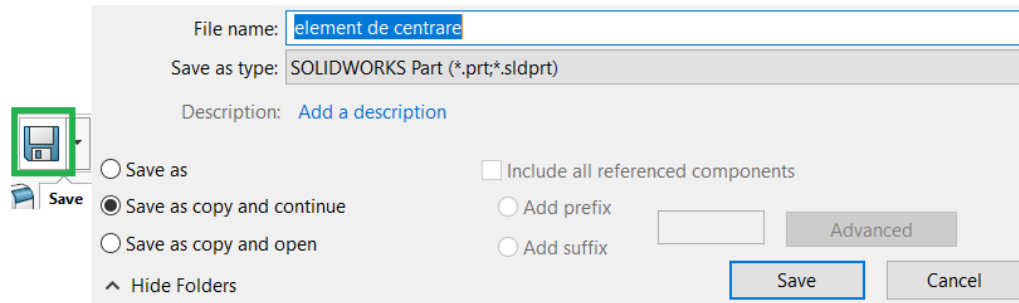
The **Mirror Face/Plane** box is clicked, and the arrow next to the part name is then pressed in the working window to open the model's feature tree, where **Right Plane** is selected. In the next step, the **Features to Mirror** window is accessed, and the **Cut-Sweep1** operation is selected from the part tree. The software also provides a preview of the mirrored bore if this option is active.

The operation is completed by confirming the settings with the **OK** checkmark.



7. Saving the Model

To complete the process, the model is saved using the **Save** button from the menu bar. In the save dialog window, the part is named *element_de_centrare*, and the storage location is selected according to the user's preferences.

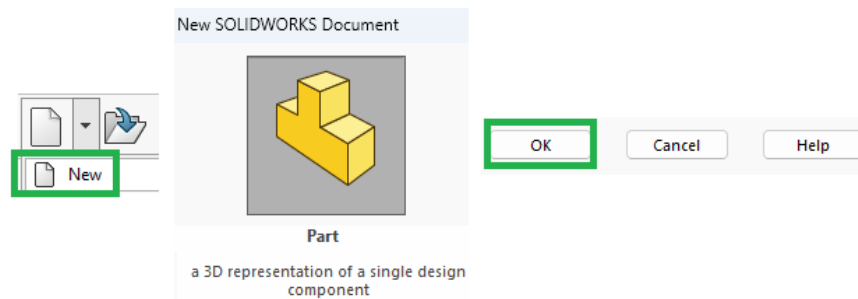


3.3.8. Modeling the Injector Needle

The injector needle within a fuel injector serves to control the flow rate and precise atomization of fuel into the combustion chamber. It moves axially to open and close the injection orifices, ensuring optimal fuel atomization.

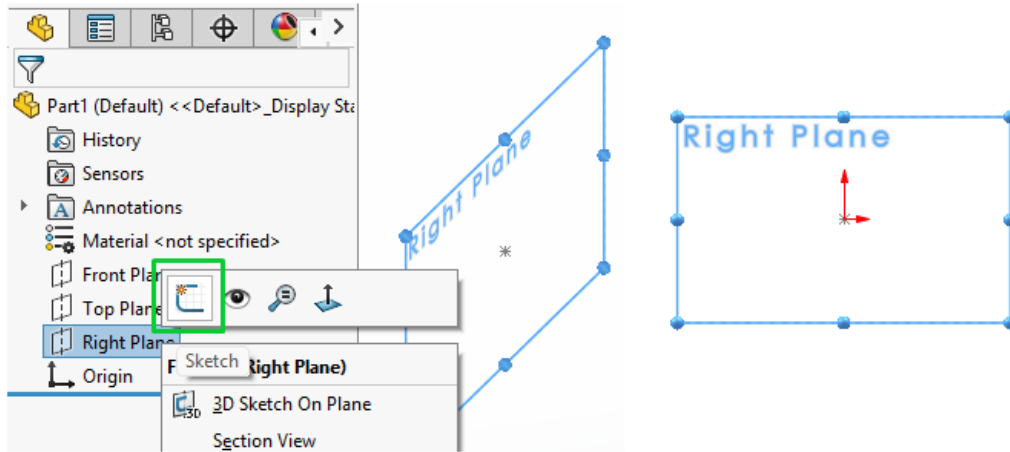
1. Creating a New Part File

To initiate the 3D modeling process, the main menu bar is accessed and the **New** command is selected. In the dialog window, the **Part** category is chosen, and the selection is confirmed by clicking the **OK** checkmark.



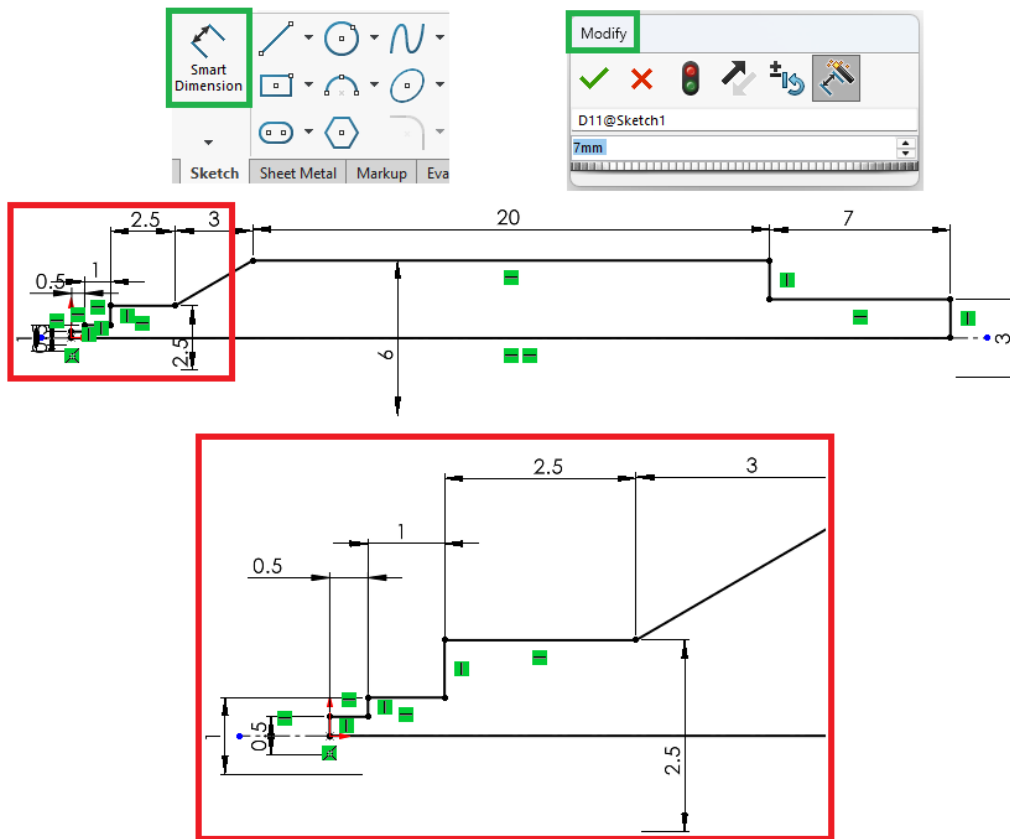
2. Creating the Initial Sketch

To initiate the modeling process, the **Right Plane** is selected as the reference plane by right-clicking on it, and sketch mode is then activated using the **Sketch** command from the toolbar.



3. Creating and Dimensioning the Profile of the Injector Needle

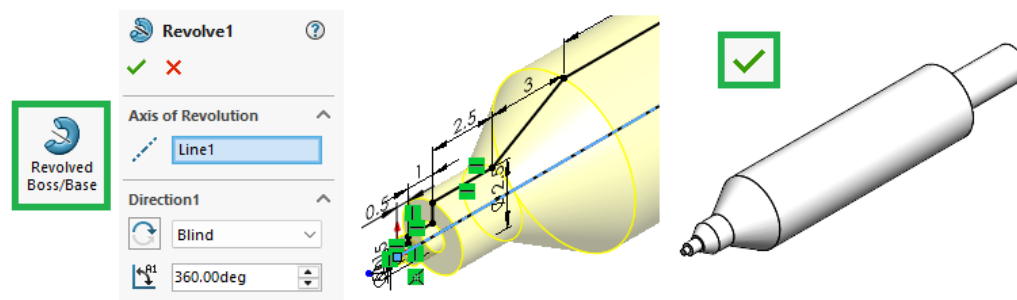
In the first phase, using the commands from the **Sketch** toolbar, a horizontal symmetry axis is drawn from the origin point using the **Centerline** command. This axis serves as a reference element for the alignment and symmetric construction of the profile. The outer contour of the part profile is then drawn. To ensure dimensional accuracy, exact dimensions are applied and distances and angles are verified according to the figure below, using the **Smart Dimension** command.



4. Generating the Body by Revolution

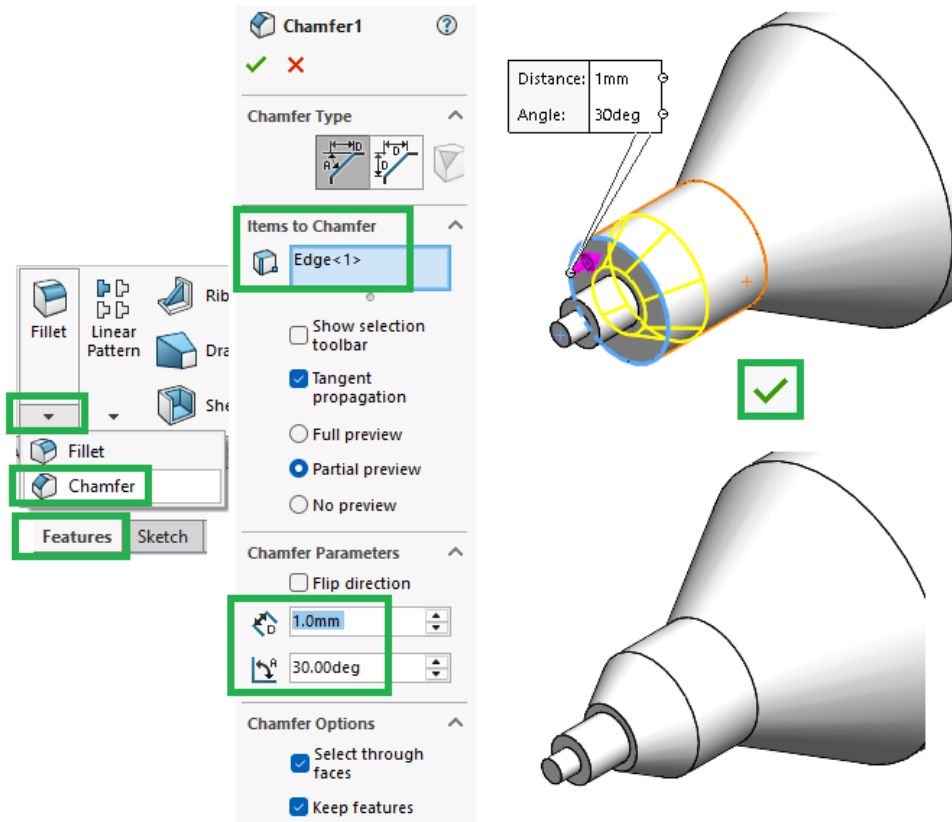
After completing the sketch, the **Revolve Boss/Base** command is accessed from the **Features** toolbar. In the **PropertyManager**, the **Axis of Revolution** box is clicked, and the symmetry axis around which the rotation is to be performed is then selected. In the **Direction** window, the **Blind** option is chosen, and the rotation angle is set to 360 degrees.

If the generated volume is not automatically previewed in yellow, the sketch created in the previous step is manually selected in the **Selected Contours** section. Following this action, the software displays a semi-transparent preview of the volume to be created. After confirming the settings by clicking the **OK** checkmark, the 3D volume is generated automatically, producing the injector needle in its final form.



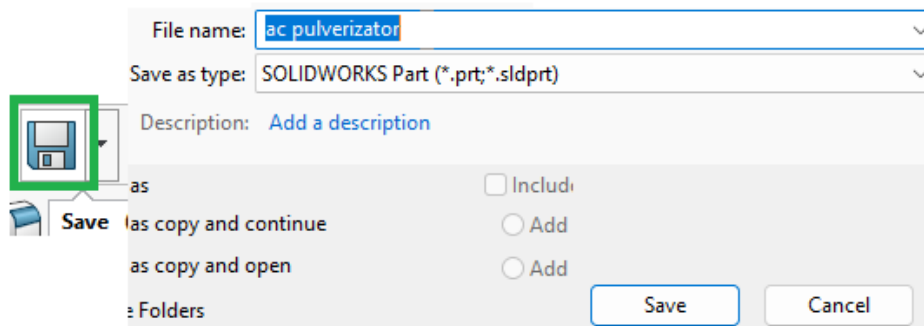
5. Applying a Chamfer to the Closing Cone of the Spray Orifice

To apply a chamfer, the **Chamfer** command is used from the **Features** toolbar. After activating the command, the configuration menu opens in the **PropertyManager**. In the **Chamfer Type** section, the desired chamfer type is selected, such as **Angle-Distance**. The edge of the part to be chamfered is then chosen in the **Items to Chamfer** box by selecting it directly from the three-dimensional model. The chamfer dimensions are set by entering the corresponding values in the **Chamfer Parameters** section, specifying 1 mm with a 30 degree inclination. After previewing the modification, the operation is confirmed by clicking the **OK** checkmark, and the chamfer is automatically generated on the selected edge.



6. Saving the Model

To complete the process, the model is saved using the **Save** button from the menu bar. In the save dialog window, the part is named *ac_pulverizator*, and the storage location is selected according to the user's preferences.

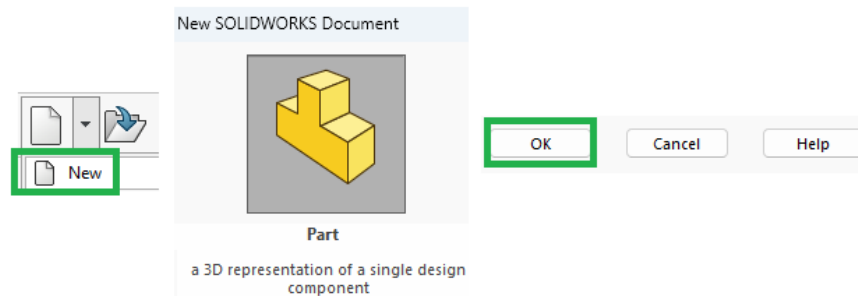


3.3.9. Modeling the Nozzle

The nozzle within a fuel injector ensures fine atomization of fuel for efficient combustion in the combustion chamber. It contributes to the uniform distribution of fuel, optimizing engine performance and reducing pollutant emissions.

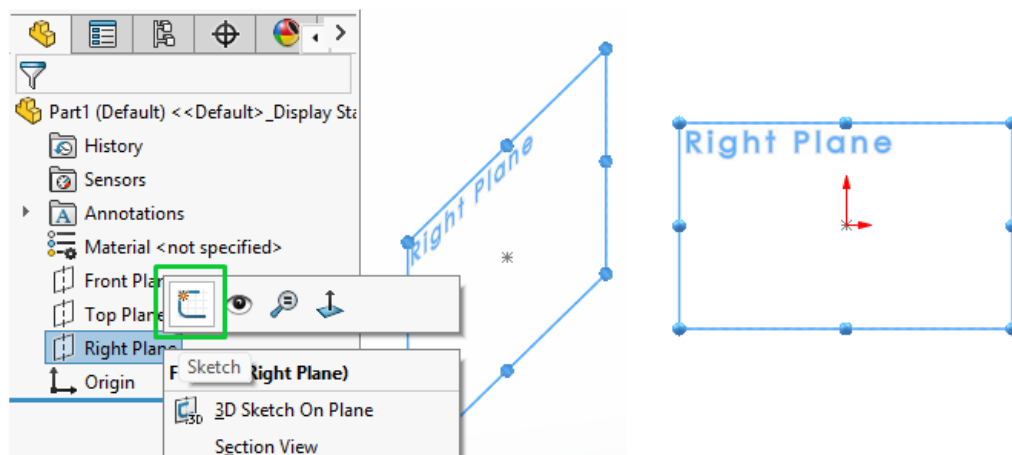
1. Creating a New Part File

To initiate the modeling process, the main menu bar is accessed and the **New** command is selected. In the selection window that appears, the **Part** option is chosen. This selection is confirmed by clicking the **OK** checkmark, thereby generating a new working file configured for the parametric design of the part.



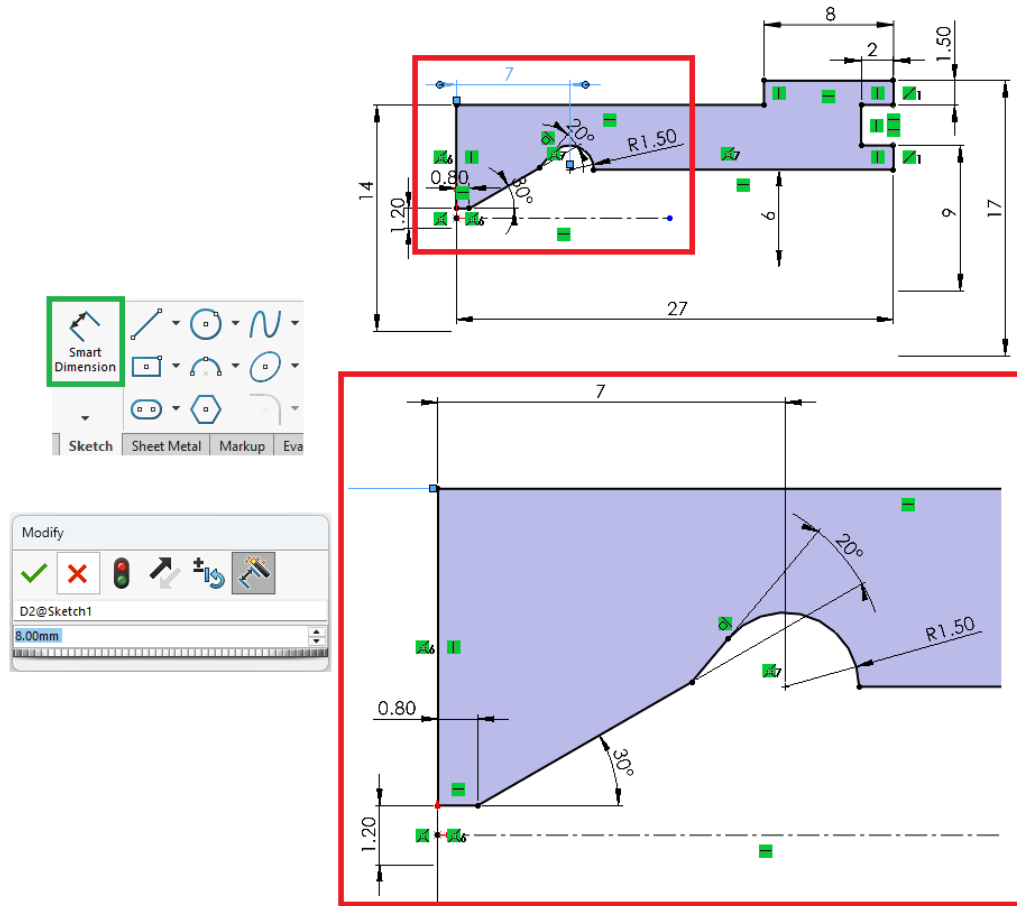
2. Creating the Initial Sketch

To initiate the modeling process, the **Right Plane** is selected as the working plane by right-clicking on it, and sketch mode is then activated using the **Sketch** command from the toolbar.



3. Creating and Dimensioning the Nozzle Sketch

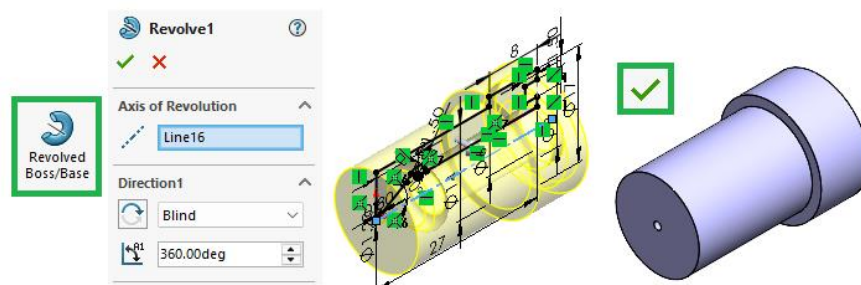
In the first phase, using the commands from the **Sketch** toolbar, a horizontal symmetry axis is drawn from the origin point using the **Centerline** command. The outer contour of the part profile is then drawn according to the figure below, respecting the dimensioning and element sizes using the **Smart Dimension** command.



4. Generating the Body by Revolution

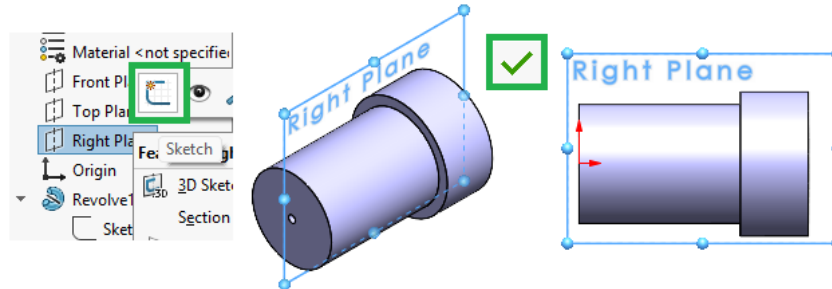
After completing the sketch, the **Revolve Boss/Base** command is accessed from the **Features** toolbar. In the **PropertyManager**, the **Axis of Revolution** box is clicked, and the symmetry axis around which the rotation is to be performed is then selected. In the **Direction** window, the **Blind** option is chosen, and the rotation angle is set to 360 degrees.

If the generated volume is not automatically previewed in yellow, the sketch created in the previous step is manually selected in the **Selected Contours** section. Following this action, the software displays a semi-transparent preview of the volume to be created. After confirming the settings by clicking the **OK** checkmark, the 3D volume is generated automatically, producing the nozzle in its final form.

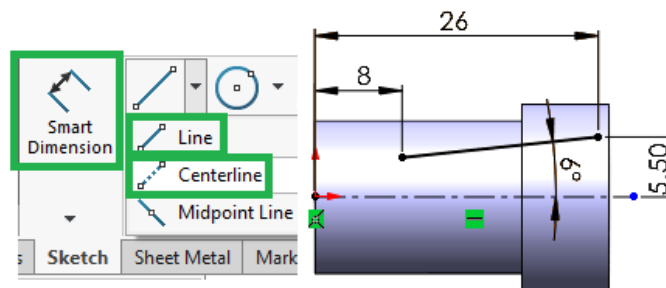


5. Creating the Bores

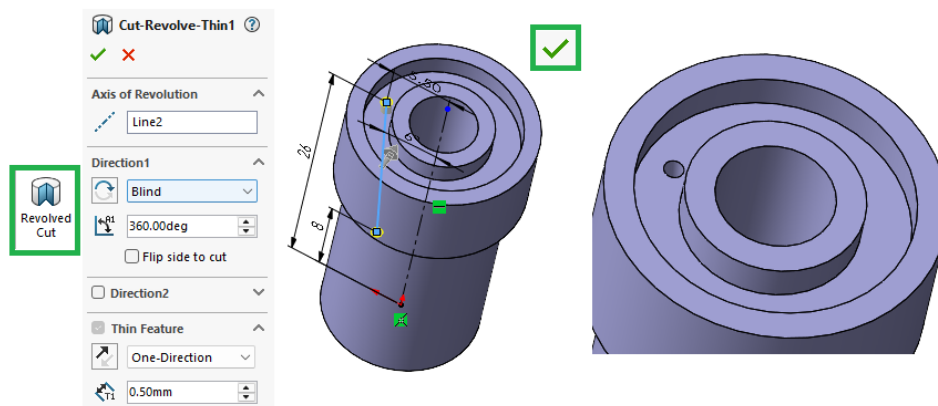
To create the high-pressure bore running through the part, **Top Plane** is right-clicked and the **Sketch** command is used to initiate a new sketch.



In the first phase, the **Centerline** and **Line** commands from the **Sketch** toolbar are used to draw a vertical symmetry axis. A segment is then drawn on the upper contour, respecting the geometric constraints imposed according to the figure below.



To generate a circular cross-section channel of 1.5 mm, the **Cut-Revolve** command is used from the **Features** toolbar. In the **PropertyManager**, the **Thin Feature** option is activated in the configuration menu. In the **Axis of Revolution** window, **Line 2**, drawn in the previous step, is selected as the axis of rotation. The circular profile dimension is defined by entering a value of 0.5 mm. After configuring the necessary parameters, the operation is completed by confirming the settings with the **OK** checkmark, thereby generating the desired channel in the part model.



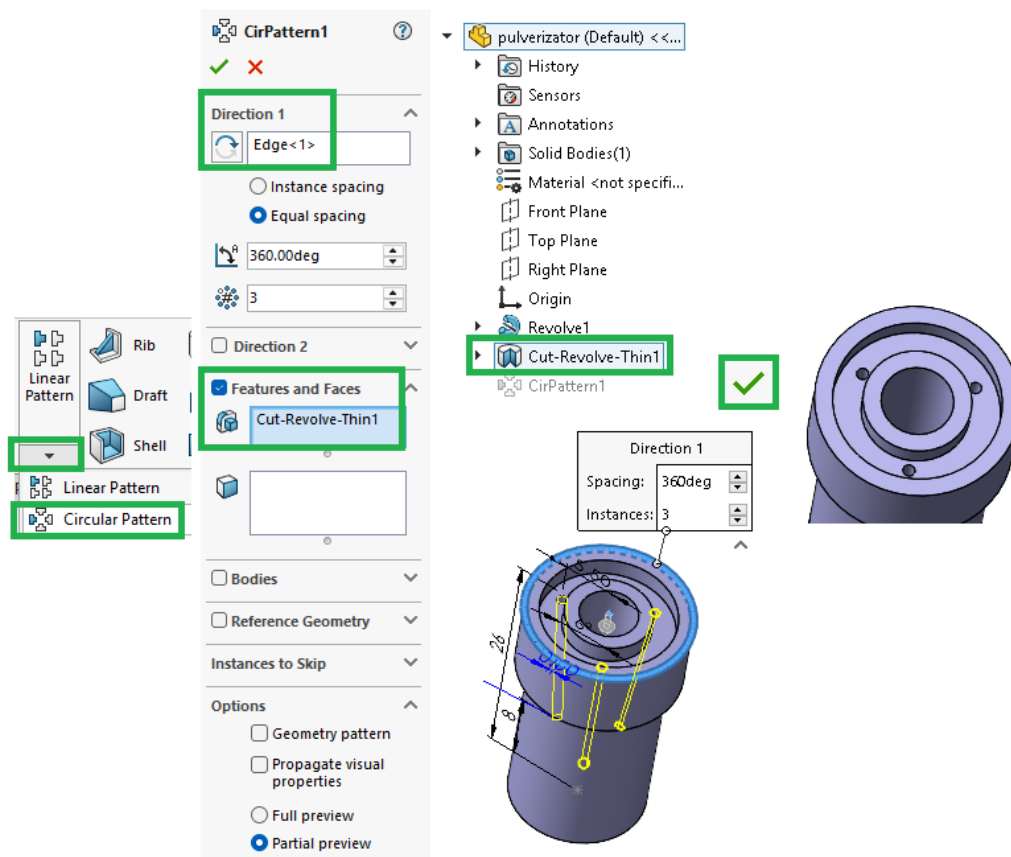
6. Circular Pattern of the Bore

To distribute three bores uniformly around the nozzle body, the **Circular Pattern** command is used from the **Features** toolbar.

After activating the command, the **Direction 1** box is accessed in the **PropertyManager**, where a circular edge or the central axis of the model is selected as the distribution reference. The **Equal Spacing** option is activated, and a value of 3 is entered in the **Number of Instances** field, ensuring uniform distribution at 120° from one another over a total angle of 360 degrees.

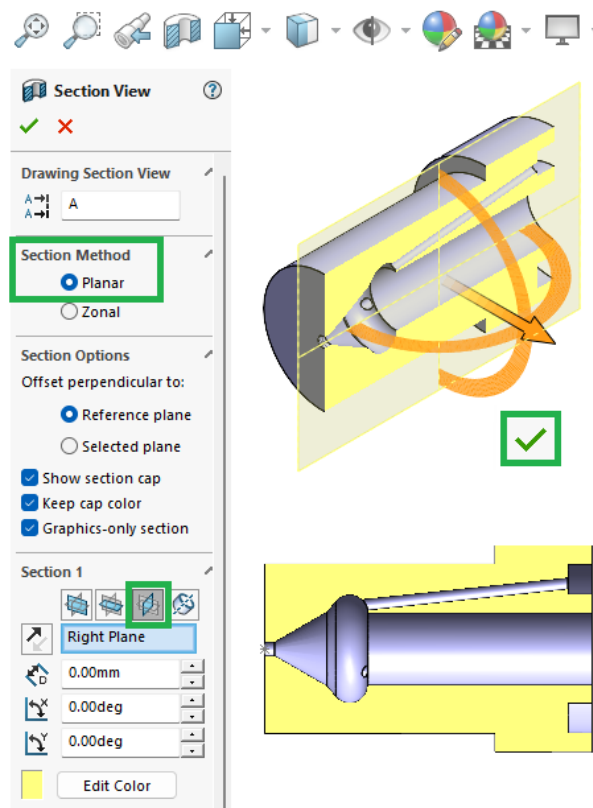
The **Circular Pattern** command is accessed from the **Features** toolbar via the dropdown menu of the **Linear Pattern** command. In the command's **PropertyManager**, the circular edge of the part is selected in the **Direction 1** field to define the pattern axis, and the number of instances is set to 3 with the **Equal Spacing** option activated. In the **Features and Faces** section, the corresponding field is clicked and the **Cut-Revolve-Thin1** operation is selected from the **FeatureManager** tree, this being the previously created bore that is to be patterned circumferentially.

After verifying the preview and confirming the parameters by clicking the **OK** checkmark, SolidWorks automatically generates the two additional bores, completing the desired configuration of the nozzle.



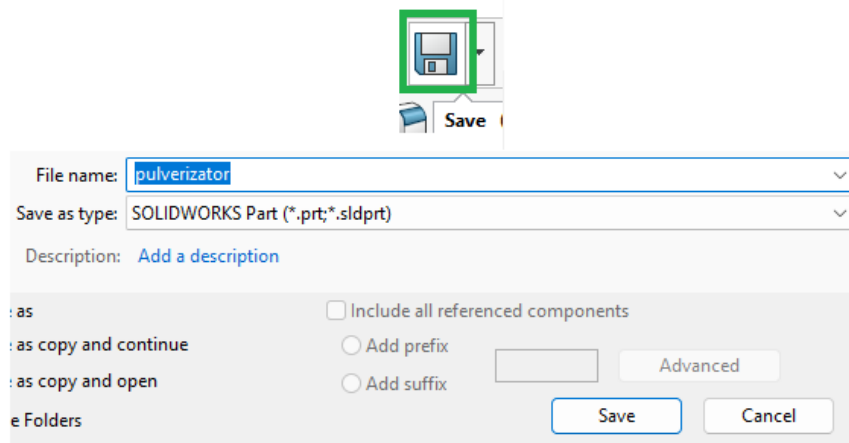
7. Visualizing the Interior Cross-Section of the Nozzle

To examine the interior of the nozzle, the **Section View** command is used from the 3D visualization area. After activating the command, the **Planar** option is selected in the **Section Method** section of the **PropertyManager**. **Right Plane** is chosen in the **Section 1** field to section the model through the first created bore. After configuring the settings, the section view is generated by clicking the **OK** checkmark. For optimal model alignment, the **Ctrl+4** key combination is used, which automatically rotates the model to the front view, facilitating detailed analysis of the internal geometry.



8. Saving the Model

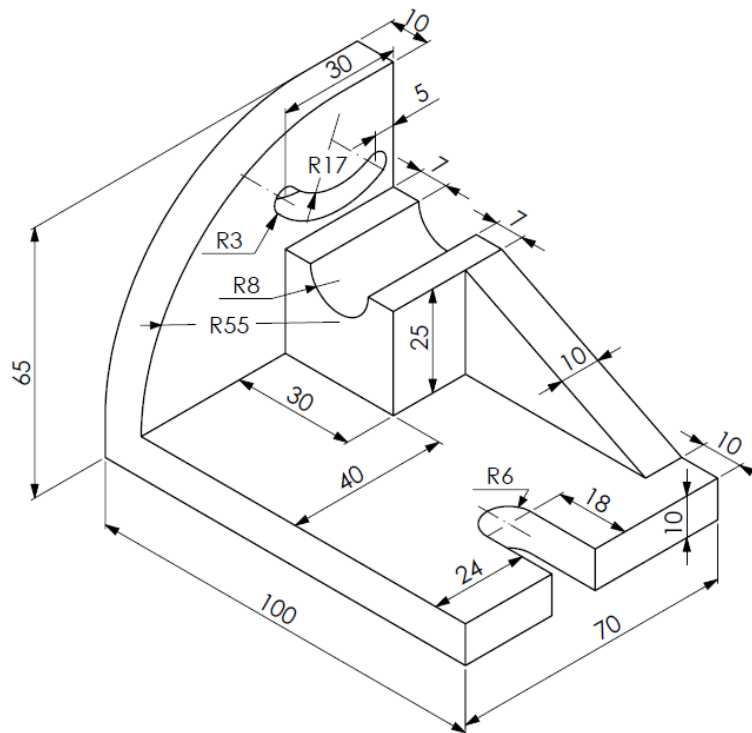
To complete the process, the model is saved using the **Save** button from the menu bar. In the save dialog window, the part is named *pulverizator*, and the storage location is selected according to the user's preferences.



3.3.10. Independent Practice Exercises

The proposed exercises are structured around the specific themes of the chapter, presenting for modeling parts with an increased degree of geometric complexity, representative of the fuel injection systems domain. Approaching them requires the integrated application of knowledge accumulated throughout the entire chapter, as well as the ability to adapt the workflow to the constructive particularities of each individual part. Unspecified dimensions are to be determined constructively, respecting the geometric proportions of the part.

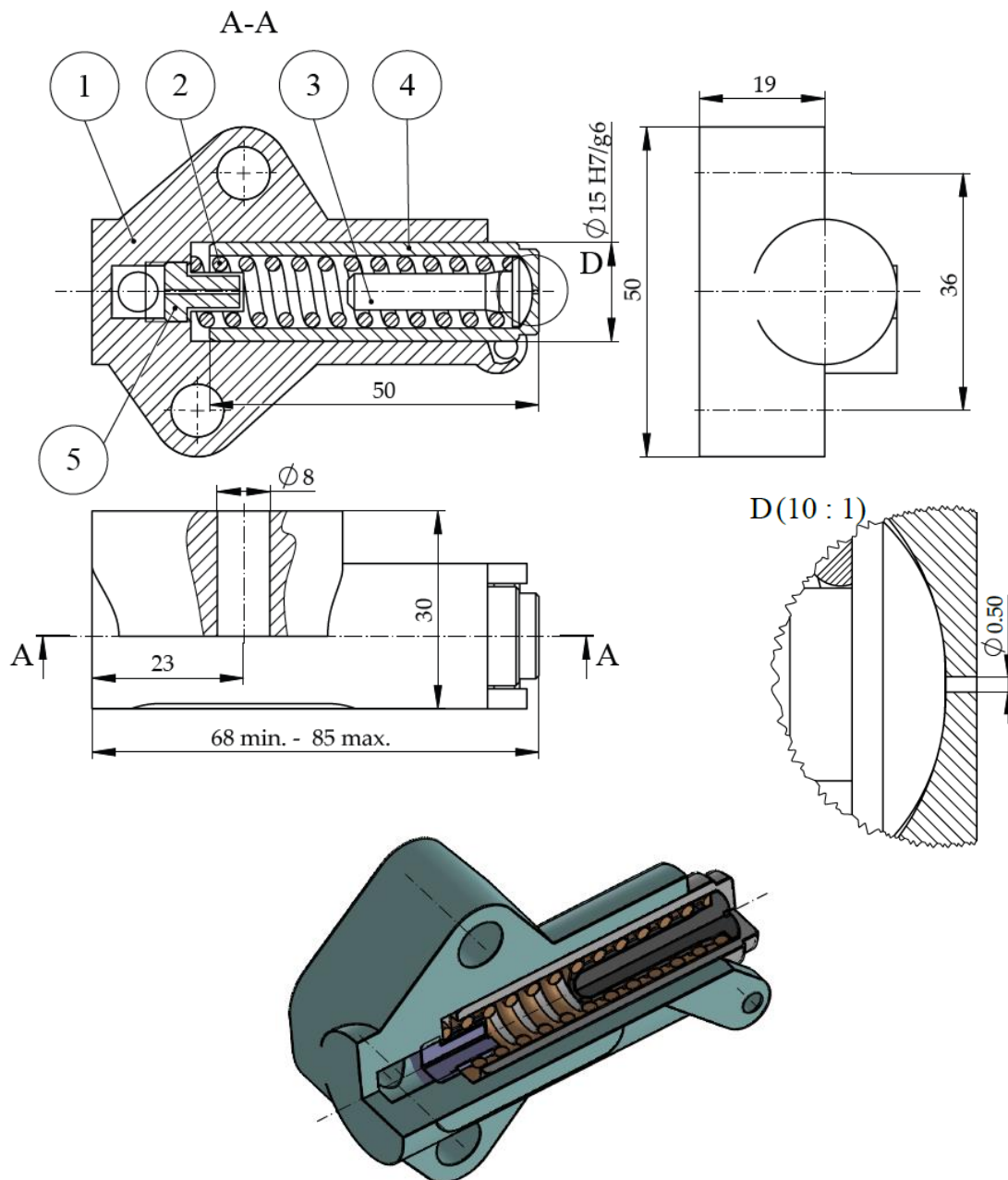
3.3.10.1. Mounting Bracket



3.3.10.2. Modeling the Components of the Hydraulic Chain Tensioner

The hydraulic tensioner serves to maintain constant tension in the timing chain, automatically compensating for the effects of elongation and wear throughout the entire operating life of the engine. Its operating principle is based on the displacement of the piston against the guide shoe under the action of oil pressure supplied by the engine lubrication system. The working chamber is fed through a one-way valve that permits only oil admission, blocking its return and thereby preventing piston retraction. The internal spring provides initial chain pre-tensioning and compensates for rapid tension variations, while oil pressure stabilizes the piston position, limiting vibrations and reducing chain wear throughout the service life of the engine.

The assembly consists of the following components: tensioner body (1) – EN AW-6082, spring (2) – 51CrV4, plastic guide (3) – PA6, piston (4) – 34CrMo4, and valve (5) – C45.



3.3.11. Assessment Questions

1. Which components exhibit axial symmetry and what advantages does this offer in modeling?

2. What are the modeling steps for a part with a complex profile using the Swept Boss/Base command in SolidWorks?

3. What are the main components of the fuel injector?

4. CONCLUSIONS AND PERSPECTIVES

This work aims to introduce students to three-dimensional parametric modeling using SolidWorks, pursuing the progressive development of the competencies required in computer-aided design and in understanding the correlation between the geometric model and the technical functionality of parts.

The first chapter presents the theoretical foundations of computer-aided design, the evolution of CAD/CAM/CAE systems, and the general structure of the SolidWorks platform, providing the conceptual framework necessary for approaching three-dimensional modeling. The second chapter addresses parametric modeling at part level through applications specific to mechanical engineering, highlighting the use of fundamental operations and parametric relations in defining constructive forms. The third chapter is dedicated to modeling the components of a compression-ignition engine fuel injector, covering parts with varying degrees of geometric complexity.

The set of modeled parts serves as the foundation for the study of the Assembly and Drawing modules, in which virtual assembly of components, definition of positioning relations, and creation of technical engineering drawings are addressed, in accordance with the representation rules specific to the field.

The material will subsequently be expanded through the introduction of dedicated applications for the Weldments, Surface Modeling, Sheet Metal, and Simulation modules, as well as through deeper work in Assembly and Drawing, based on integrated examples.

5. SUBJECTS FOR COMPETENCY ASSESSMENT

5.1. Examination Paper No. 1

<p>Final Assessment – CAD Examination Paper 1 Collar Clamp</p>						
<p>Given the engineering drawing and the axonometric representation of the part, produce a full-scale model using the Part module in SolidWorks. All unspecified dimensions are to be determined constructively, in accordance with technical design standards.</p>						
<p>Technical Conditions: All sharp edges are to be chamfered at $0.5 \times 45^\circ$ Unspecified fillet radii are to be R0.5 Medium execution class, in accordance with SR EN ISO 22081:2021</p>						
<p>ASSESSMENT- TOTAL 10 POINTS: 1 P. - GRANTED BY DEFAULT</p>			<p>SolidWorks Modelling - 9 points: 4 p. - geometric accuracy and correctness of constructive relations 3 p. - correct and efficient use of modelling commands 2 p. - organization and structuring of the model in the FeatureManager</p>			
Acest desen aparține UTCN Toate drepturile asupra lui sunt rezervate	Drawn by Reviewed	dr.ing. Szabo Ioan Conf.dr. ing. Scurtu Liviu	2025.05.06	S235JR		
	2:1	TECHNICAL UNIVERSITY OF CLUJ-NAPOCA				
		Clamp				
	A4	C 001				

5.2. Examination Paper No. 2

Final Assessment – CAD_Examination Paper 2
Knob

Given the engineering drawing and the axonometric representation of the part, produce a full-scale model using the Part module in SolidWorks. All unspecified dimensions are to be determined constructively, in accordance with technical design standards.

ASSESSMENT-TOTAL 10 POINTS:
1 P. - GRANTED BY DEFAULT

SolidWorks Modelling - 9 points:
4 p. - geometric accuracy and correctness of constructive relations
3 p. - correct and efficient use of modelling commands
2 p. - organization and structuring of the model in the FeatureManager

Acest desen aparține UTCN Toate drepturile asupra lui sunt rezervate	Drawn by	dr.ing. Szabo Ioan	2025.05.06	HDPE
	Revised	Conf. dr.ing. Scurtu Liviu		
	2:1	TECHNICAL UNIVERSITY OF CLUJ-NAPOCA		
		Knob		
A4	B 001			

5.3. Examination Paper No. 3

Final Assessment – CAD_Examination Paper 1
Connector

Given the engineering drawing and the axonometric representation of the part, produce a full-scale model using the Part module in SolidWorks. All unspecified dimensions are to be determined constructively, in accordance with technical design standards.

ASSESSMENT- TOTAL 10 POINTS: 1 P. - GRANTED BY DEFAULT	SolidWorks Modelling - 9 points: 4 p. - geometric accuracy and correctness of constructive relations 3 p. - correct and efficient use of modelling commands 2 p. - organization and structuring of the model in the FeatureManager																				
Acest desen aparține UTCN Toate drepturile asupra lui sunt rezervate	<table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 15%; padding: 2px;">Drawn by</td> <td style="width: 25%; padding: 2px;">dr.ing. Szabo Ioan</td> <td style="width: 15%; padding: 2px;"></td> <td style="width: 15%; padding: 2px;">2025.05.06</td> <td style="width: 30%; padding: 2px;">HDPE</td> </tr> <tr> <td style="padding: 2px;">Revised</td> <td style="padding: 2px;">Conf. dr. ing. Scurtu Liviu</td> <td style="padding: 2px;"></td> <td colspan="2" style="padding: 2px; text-align: center;">TECHNICAL UNIVERSITY OF CLUJ-NAPOCA</td> </tr> <tr> <td style="padding: 2px; text-align: center;">2:1</td> <td colspan="4" style="padding: 2px; text-align: center;">Connector</td> </tr> <tr> <td style="padding: 2px; text-align: center;"> </td> <td colspan="4" style="padding: 2px; text-align: center;">C 001</td> </tr> </table>	Drawn by	dr.ing. Szabo Ioan		2025.05.06	HDPE	Revised	Conf. dr. ing. Scurtu Liviu		TECHNICAL UNIVERSITY OF CLUJ-NAPOCA		2:1	Connector					C 001			
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Revised	Conf. dr. ing. Scurtu Liviu		TECHNICAL UNIVERSITY OF CLUJ-NAPOCA																		
2:1	Connector																				
	C 001																				

5.4. Examination Paper No. 4

Final Assessment – CAD_Examination Paper 4
Spring - Pressure Plate

Given the engineering drawing and the axonometric representation of the part, produce a full-scale model using the Part module in SolidWorks. All unspecified dimensions are to be determined constructively, in accordance with technical design standards.

Technical Conditions:
 All sharp edges are to be chamfered at $0.5 \times 45^\circ$
 Unspecified fillet radii are to be R0.5
 Medium execution class, in accordance with SR EN ISO 22081:2021

ASSESSMENT- TOTAL 10 POINTS: 1 P. - GRANTED BY DEFAULT	SolidWorks Modelling - 9 points: 4 p. - geometric accuracy and correctness of constructive relations 3 p. - correct and efficient use of modelling commands 2 p. - organization and structuring of the model in the FeatureManager		
Drawn by Revised	dr.ing. Szabo Ioan dr.ing. Szabo Ioan	2025.05.06	51CrV4
2:1		UNIVERSITATEA TEHNICĂ DIN CLUJ-NAPOCA	
		Spring – Pressure Plate	
A4		APC 001	

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 Toate drepturile asupra lui
 sunt rezervate

5.5. Examination Paper No. 5

Final Assessment – CAD_Examination Paper 5
Fitting

Given the engineering drawing and the axonometric representation of the part, produce a full-scale model using the Part module in SolidWorks. All unspecified dimensions are to be determined constructively, in accordance with technical design standards.

Technical Conditions:
 All sharp edges are to be chamfered at $0.5 \times 45^\circ$
 Unspecified fillet radii are to be R0.5
 Medium execution class, in accordance with SR EN ISO 22081:2021

ASSESSMENT- TOTAL 10 POINTS: 1 P. - GRANTED BY DEFAULT	SolidWorks Modelling - 9 points: 4 p. - geometric accuracy and correctness of constructive relations 3 p. - correct and efficient use of modelling commands 2 p. - organization and structuring of the model in the FeatureManager
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Drawn by	dr.ing. Ioan Szabo	2025.05.06	S235
Revised	dr.ing. Ioan Szabo		
2:1	TECHNICAL UNIVERSITY OF CLUJ-NAPOCA		
	Fitting		
A4	F-001		

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