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GRAPHICAL MODELLING OF PARTS WITH SIMPLE GEOMETRY IN VARIOUS CAD SYSTEMS

Abstract: *Computer-aided design - CAD is one of the most widespread and efficient methods used in modern engineering. It contributes to increasing precision, optimizing the design process and reducing production costs. Being an essential tool in multiple fields, CAD design has revolutionized the way parts and systems are designed and manufactured. The paper analyses the facilities and features offered by three different CAD programs - Autodesk Inventor, CATIA and NX, providing a practical comparison between them. By modelling the same parts, the differences between the workflows, the production times, as well as the degree of efficiency specific to each program are presented. Thus, the aim is to highlight the technical functionalities, providing users with a solid basis for choosing a CAD solution adapted to the requirements of efficiency and productivity.*

Key words: *NX, graphical modelling, Autodesk Inventor, CATIA.*

1. INTRODUCTION

Graphical modelling refers to the use of visual representations, such as diagrams, schematics and 3D models, to describe the structure, behavior and interactions of systems, components or processes [1], [2].

In graphical modelling, the various elements of a system or product are represented by symbols, pictograms or geometric shapes [3], often following industry- or domain-specific conventions or standards [4], [5].

The most common form of graphical modelling is computer-aided design (CAD), which allows engineers to design, simulate and visualize products [6], [7]. It is an integral part of modern engineering, increasing accuracy, improving the design process and reducing costs. Also, it is an important tool in various fields, revolutionizing the way products and systems are designed and manufactured [8].

There are two types of graphical modelling [9], [10]: 2D modelling (execution and assembly drawings, often used in architecture or engineering to represent the dimensions and specifications of a product) and 3D modelling (representations of objects in three dimensions, essential in various fields such as the automotive industry, mechanical engineering, architecture etc.) [11], [12].

3D modelling refers to the use of programs to create, modify, analyze and optimize 3D models. These models represent physical objects with various geometrical and dimensional attributes.

Unlike 2D modelling, which creates drawings in a two-dimensional environment, 3D modelling provides an overall, three-dimensional view, giving designers the ability to see all sides of an object. One of the advantages of graphical modelling is that complex systems can be visualized in a more intuitive way, allowing engineers and designers to easily understand the structure and behavior of the model [13], [14].

Simulations on graphical models can also help identify potential design flaws before physical prototypes are created, reducing errors and production costs [15].

Also, depending on the program used by the operator, the models can be subsequently modified and optimized to meet the specific requirements of the project [16], [17]. The generic scheme of a CAD system is presented in Figure 1.

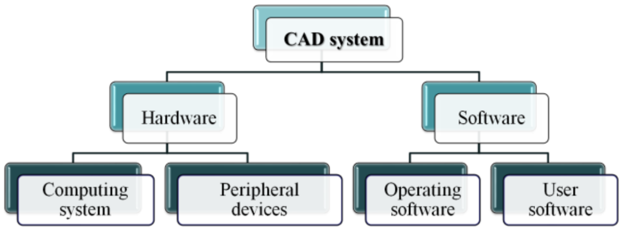


Figure 1 The structure of a CAD system [18].

Every CAD system consists of two main components that must work in harmony to successfully create, edit and manipulate 3D graphical models: the hardware component, which includes the set of physical equipment necessary for data processing), ensuring computational support and efficient interaction with the design environment, and the software component, which consists of specialized applications for computer-aided design in 2D and 3D environments, as well as the simulation of the functional behavior of parts, all of which are managed in an integrated environment that provides advanced tools for creating and optimizing models [18].

In this paper, the functionality of three different software programs - Autodesk Inventor, CATIA and NX - will be presented by parametric modelling of two parts with simple geometry. These are among the most widely used CAD programs in industries such as naval, aerospace, aeronautical, medical, automotive etc. Each of these programs offers a unique set of features, strengths and specialized tools, making them useful for different applications, depending on the requirements.

Each part will be modeled in the three programs, following the way of defining sketches, applying constraints, as well as using the commands specific to each design program. The goal is to highlight the functional differences, as well as the efficiency of each software in the 2D and 3D design process.

2. GRAPHICAL MODELLING OF A CLAMPING FORK HEAD PART

2.1 Graphical modelling of the clamping fork head using the Autodesk Inventor program

The first part, which will be modeled in the Autodesk Inventor program, is represented by a clamping fork head that is usually used in articulation mechanisms or transmission elements, Figure 2. The first step consists of choosing the reference plane for making the arms of the clamping fork head [19].

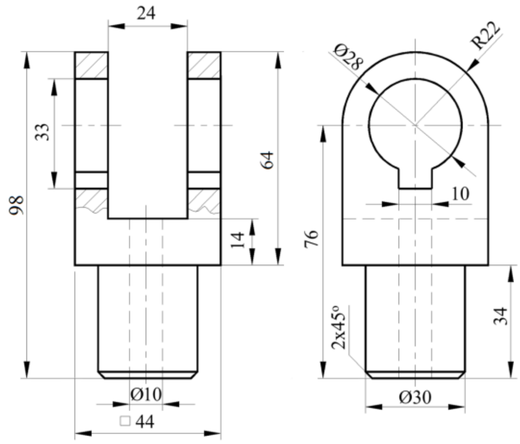
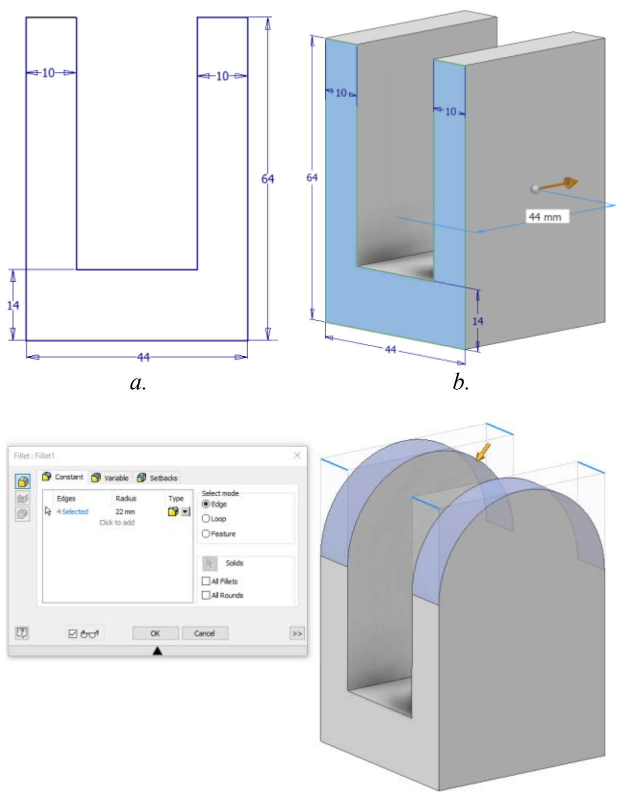


Figure 2 *Clamping fork head - 2D model* [19], [20].

Using *Line* and *Trim* commands, the model in Figure 3a is obtained, the dimensions being taken from Figure 2.



a. b.

c.

Figure 3 *3D model of the arms of the clamping fork head made in Autodesk Inventor: a. 2D model; b. 2D model extrusion; c. obtaining the 3D model.*

In 3D design environment, using the *Extrude* command, at a distance of 44 mm, the model shown in Figure 3b is obtained.

The final three-dimensional model of the arms of the clamping fork head is obtained, using the *Fillet* command, with a connection radius of 22 mm, Figure 3c.

Next, on one of the faces of the arms, the channel sketch is made, according to the dimensions in Figure 2 and using the *Extrude* command, *Cut* module, cutting distance: *Through All*, the model shown in Figure 4 is obtained.

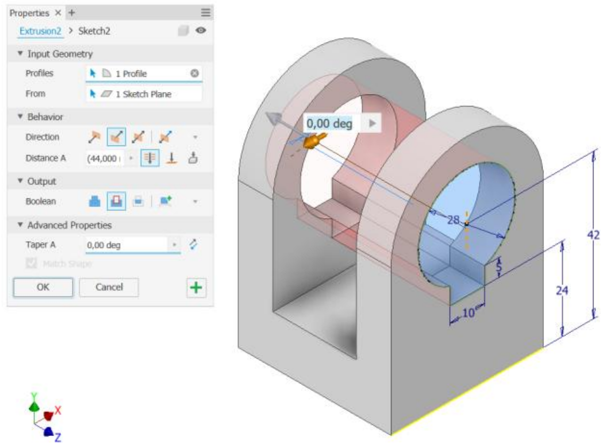
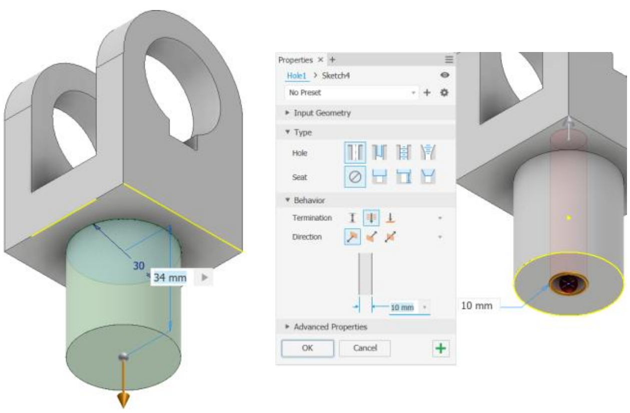


Figure 4 *Graphical modelling of the channel in the Autodesk Inventor program.*

At the base of the fork, a circle is drawn, using the *Circle* command, with a diameter of 30 mm. In the 3D graphical design environment, using the *Extrude* command, at a distance of 34 mm, the cylinder of the part is obtained, Figure 5a.

The Ø10 mm bore is obtained using the *Hole -> Through all* command, obtaining the model shown in Figure 5b.



a. b.

Figure 5 *Modelling of the cylinder (a) and the bore of the part (b) in the Autodesk Inventor program.*

Finally, after cutting the 2 mm cylindrical part using the *Chamfer* command, the 3D model of the clamping fork head part is obtained, created using the Autodesk Inventor program, Figure 6.

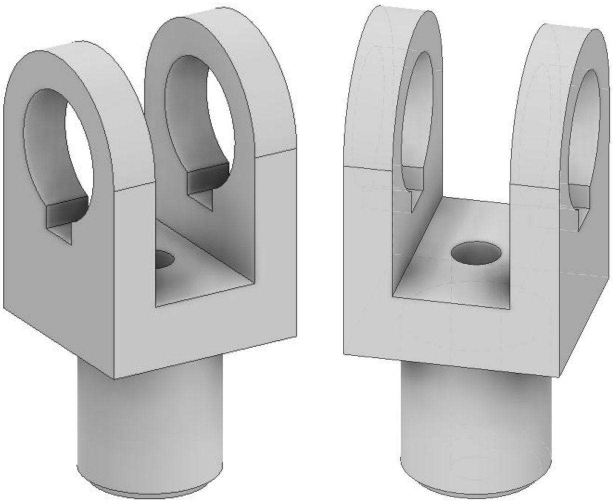


Figure 6 *3D model of the clamping fork head part made in the Autodesk Inventor program.*

2.2 Graphical modelling of the clamping fork head using the CATIA program

The part previously modeled in the Autodesk Inventor program will be recreated using CATIA V5R21 design environment, *Part Design* module, in order to highlight the differences and modelling particularities between the two CAD programs. The modelling process aims to recreate the geometry of the part, following the same logical steps, but adapted to the specifics of the CATIA design environment.

To model the clamping fork head part, the *YZ* plane is chosen and the part arms are constructed using the *Profile* command. By applying the related constraints, the 2D and 3D model, using the *Pad* command, of the part arms is obtained, Figure 7.

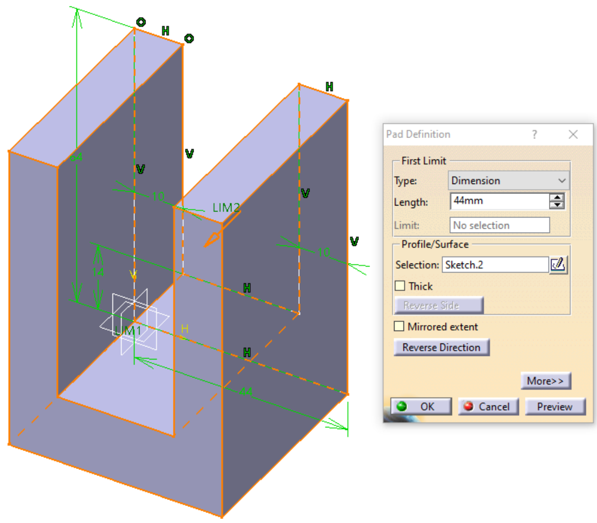


Figure 7 *3D model of the fixing element made in the CATIA program.*

Using the *Edge Fillet* command, the arms are connected at a radius of 22 mm, Figure 8.

The outline of the channel is created on one of the faces of the part arms, using the *Circle* and *Line* commands. The 3D modelling of the channel is created using the *Pocket* command, Figure 9.

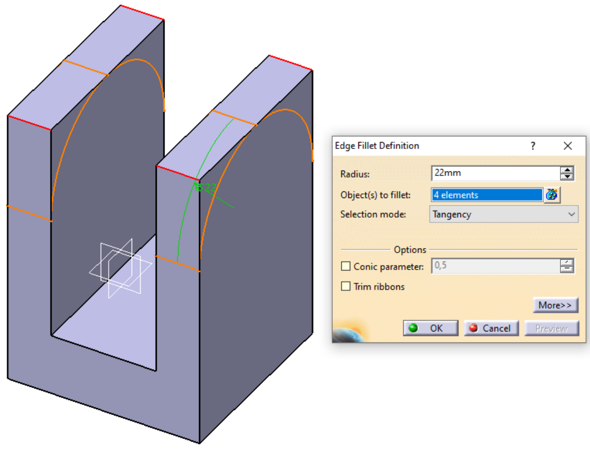


Figure 8 *Connecting the arms of the part using CATIA.*

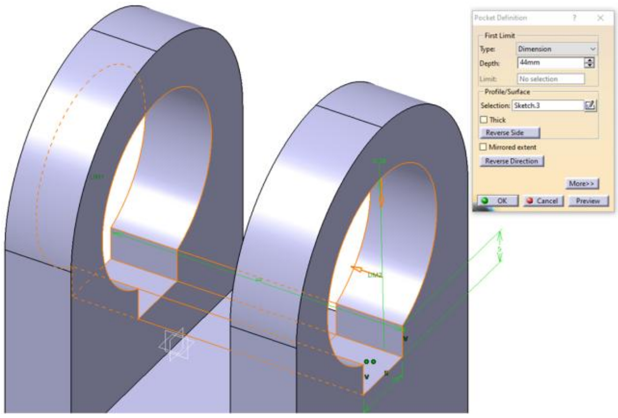


Figure 9 *Graphical modelling of the channel using CATIA.*

Subsequently, the part cylinder and the bore are modeled, using the *Pad* and *Hole* commands, Figure 10.

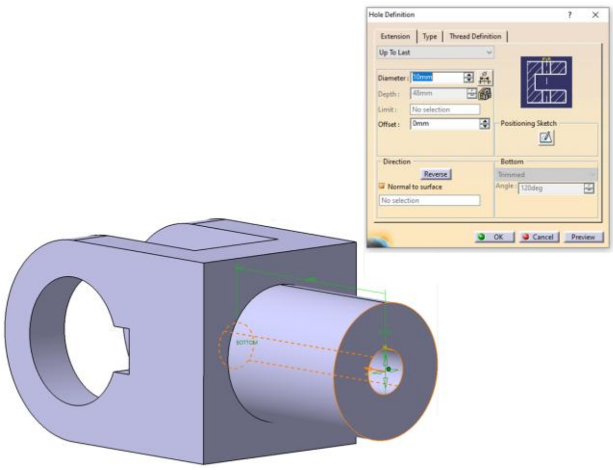


Figure 10 *Modelling of the cylinder and the bore of the part in the CATIA program.*

The 2 mm cut of the cylinder is done in the 3D environment, using the *Chamfer* command.

Thus, in Figure 11 the three-dimensional model of the clamping fork head part, made in the CATIA program, is presented.

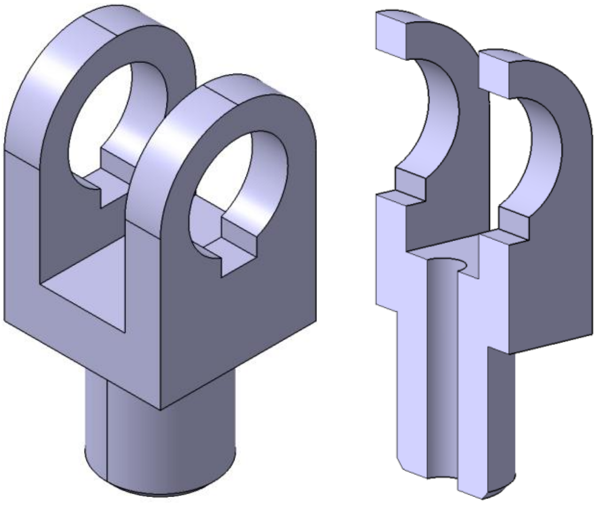


Figure 11 *3D model of the clamping fork head part made in the CATIA program.*

2.3 Graphical modelling of the clamping fork head using the NX program

The clamping fork head part, previously modeled in Autodesk Inventor and CATIA, will be recreated, applying the functions and commands specific to NX, version 2406, student edition, to highlight both the differences and modelling particularities between the three CAD programs, as well as the similarities between them, many of the commands having similar functionality.

Therefore, for modelling the clamping fork head part, the part arms sketch is initially built on the *XZ* plane, using the *Profile* command. In the 3D design environment, using the *Extrude* commands, at a width of 44 mm and *Edge Blend*, with a radius of 22 mm, the part arms are obtained, Figure 12.

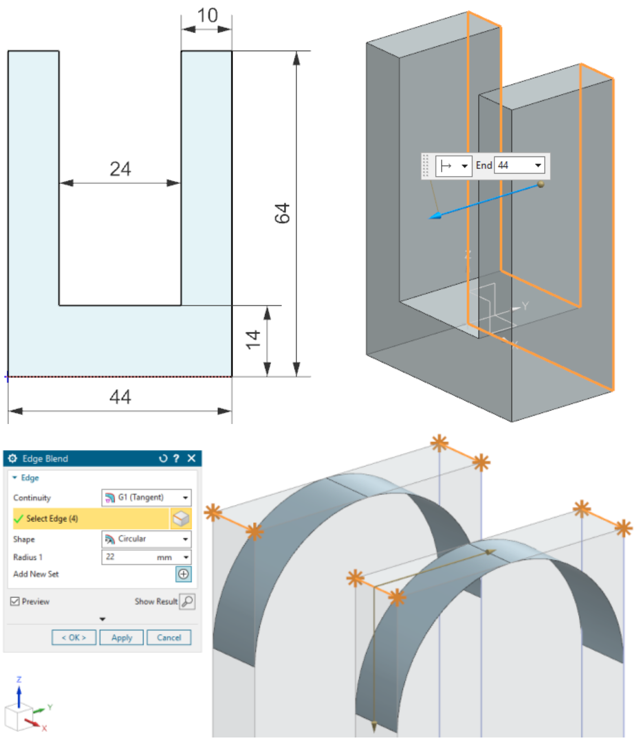


Figure 12 *3D model of the arms of the part using NX.*

The channel is built on one of the faces of the arms of the part, using the *Circle*, *Line* and *Trim* commands, obtaining the 3D model using the *Extrude* command, *Through All* module, shown in Figure 13.

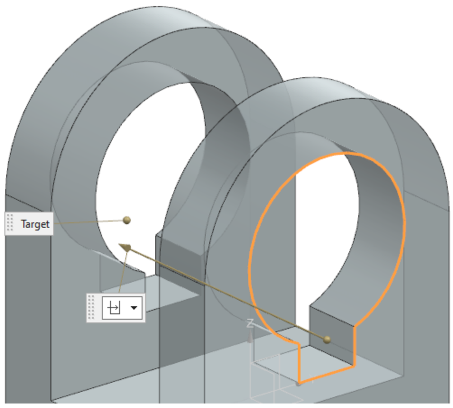


Figure 13 *3D model of the channel made in the NX program.*

The cylinder, with a diameter of Ø30 mm, is obtained using the *Extrude* command, Figure 14. Also, the Ø10 mm bore can be obtained either using the *Hole* command or the *Extrude* command.

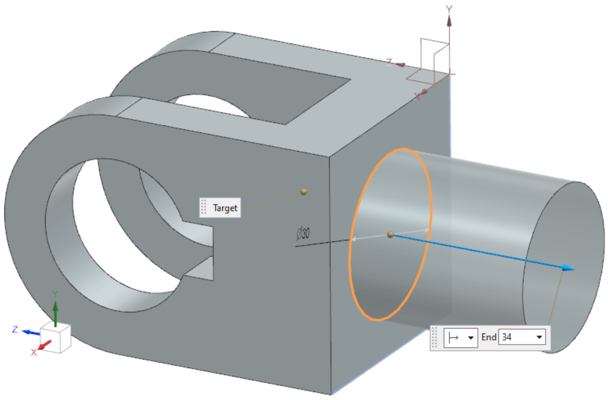


Figure 14 *The 3D model of the cylinder using the NX program.*

Finally, after cutting the cylinder by 2 mm, using the *Chamfer* command, the 3D model of the clamping fork head part is obtained in Figure 15, created using the NX program.

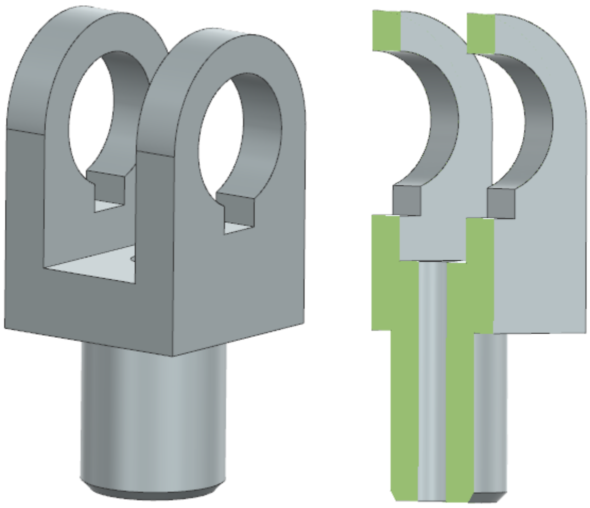


Figure 15 *3D model of the clamping fork head made in NX.*

3. GRAPHICAL MODELLING OF A FIXING PLATE PART

3.1 Graphical modelling of the fixing plate using the Autodesk Inventor program

The second model is represented by the fixing plate, Figure 16.

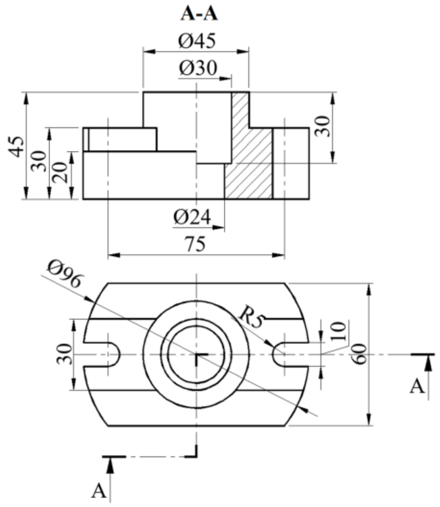


Figure 16 *Fixing plate - 2D model* [19], [20].

Initially, the base plate is created on the *XZ* plane, obtaining the 3D model in Figure 17 using the *Extrude* command, at a height of 20 mm.

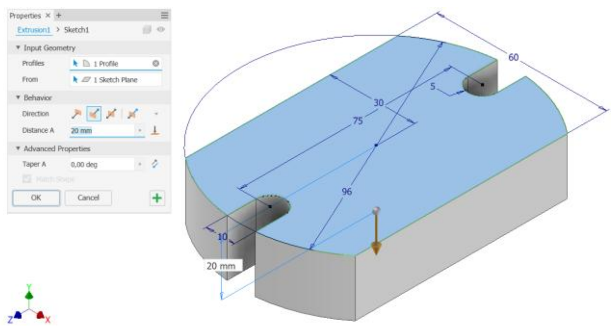
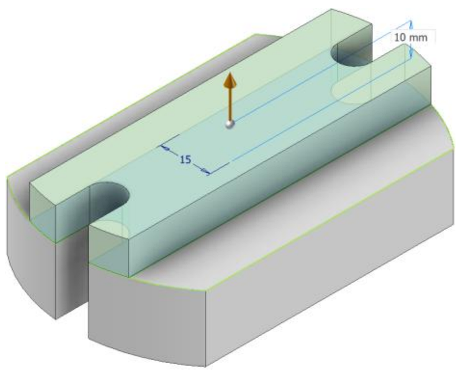


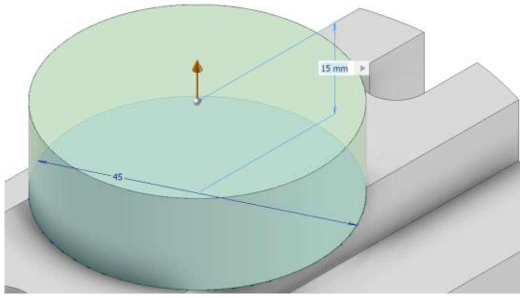
Figure 17 *Modelling the base plate in Autodesk Inventor.*

The base plate sketch was created using the 2D drawing commands, *Line* and *Circle*, as well as the editing command, *Trim*.

Subsequently, the intermediate plate is created on the base plate, using *Project Geometry* and *Line* commands, and extruded to a height of 10 mm, Figure 18a.



a.



b.

Figure 18 *Extrusion of the intermediate plate (a) and cylinder (b) in the Autodesk Inventor program.*

Also, by extruding a circle with a diameter of Ø45 mm, at a height of 15 mm, the cylinder of the part is obtained, Figure 18b.

To create the bore, the *Hole* command is used. The bore type will be *Counterbore*, with the dimensions corresponding to the 2D model of the part, Figure 19.

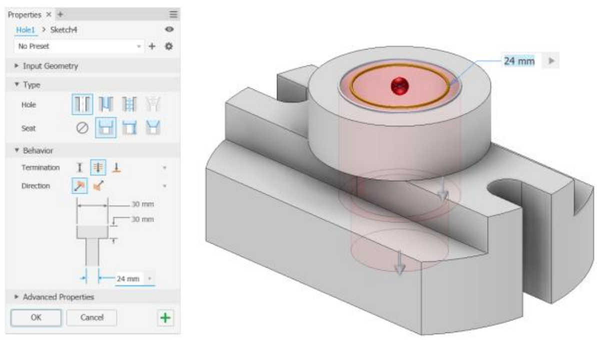


Figure 19 *Creating the bore using Autodesk Inventor.*

Thus, the 3D model of the fixing plate made in the Autodesk Inventor is shown in Figure 20. The *Split* command was used in order to section the model.

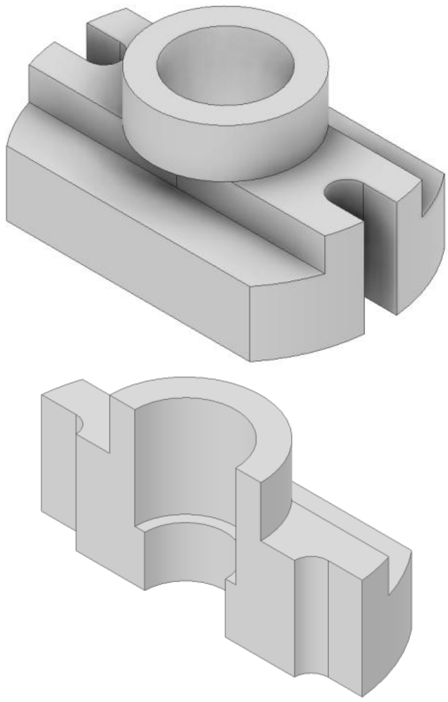


Figure 20 *3D model of the fixing plate using the Autodesk Inventor program.*

3.2 Graphical modelling of the fixing plate using the CATIA program

For the graphical modelling of the fixing plate in the CATIA program, the base plate is initially created. The 3D model is shown in Figure 21, this being created using the *Pad* command.

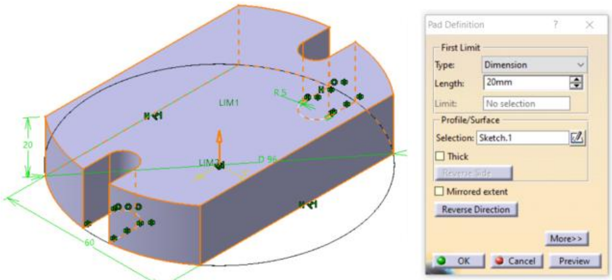


Figure 21 *Modelling the base plate using CATIA.*

The base plate sketch was created using the *Line*, *Circle*, *Mirror* and *Trim* commands. The intermediate plate will be modeled on the base plate, using the same commands as the previous model, Figure 22.

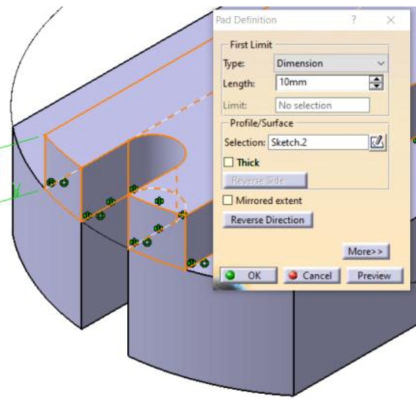


Figure 22 *Modelling the intermediate plate in CATIA.*

The Ø45 mm cylinder on the intermediate plate was modeled using the *Circle* and *Pad* commands, Figure 23.

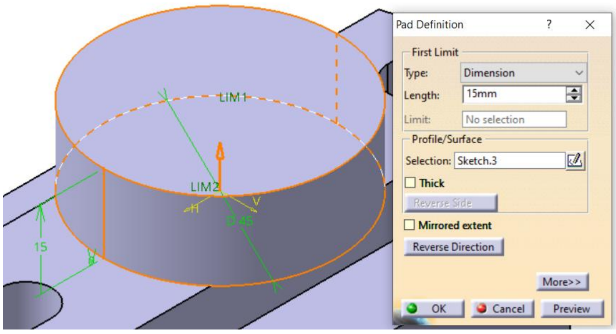


Figure 23 *Modelling the cylinder in the CATIA program.*

The bore is made using the *Hole* command, Figure 24, the bore type being *Counterbore*, with dimensions taken from Figure 16.

Thus, the final three-dimensional model of the fixing plate part, made in the CATIA program, is shown in Figure 25.

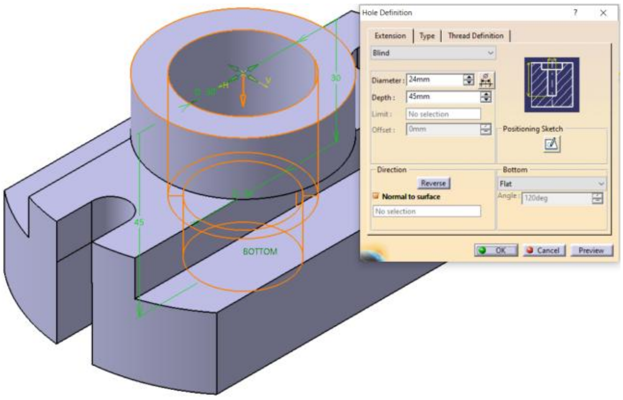


Figure 24 *Modelling of the bore using the CATIA program.*

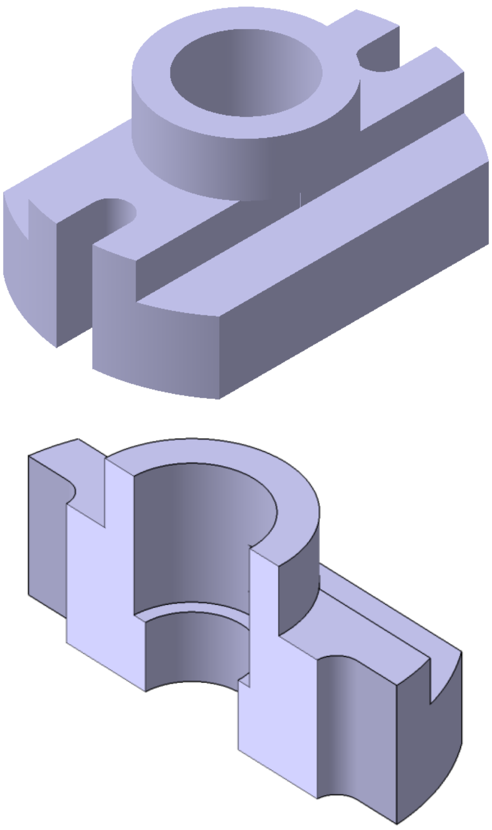


Figure 25 *3D model of the fixing plate made in the CATIA program.*

3.3 Graphical modelling of the fixing plate using the NX program

To obtain the 3D model of the fixing plate part in the NX program, having the dimensions shown in Figure 16, the base plate is initially built, Figure 26.

In the construction of the sketch, the *Circle*, *Line*, *Mirror* and *Trim* commands were used, the base plate being extruded into the three-dimensional environment at a height of 20 mm.

Also, on the surface of the base plate, the intermediate plate will be built which, using the *Extrude* command, at a height of 10 mm, will lead to obtaining its 3D model in the NX program, Figure 27.

Subsequently, on the intermediate plate, the cylinder with a diameter of Ø45 mm will be built, using the *Extrude* command, at a height of 15 mm, Figure 28.

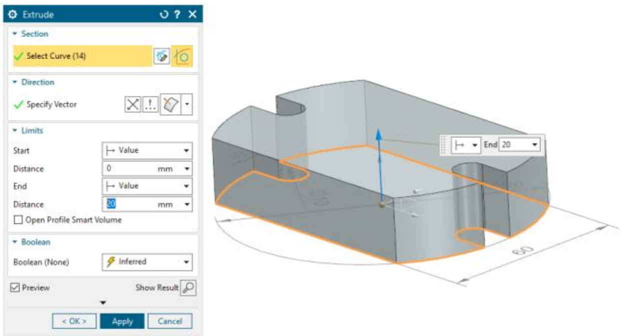


Figure 26 *3D model of the base plate using the NX program.*

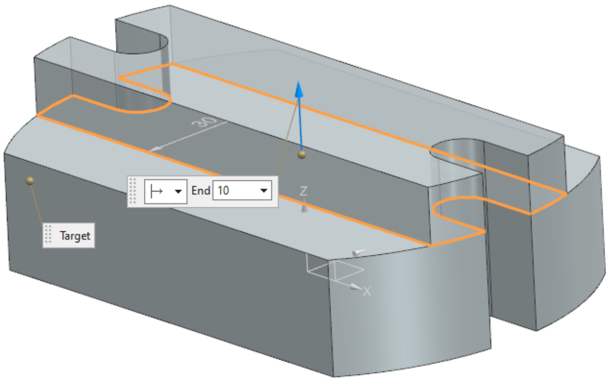


Figure 27 *3D model of the intermediate plate made in NX.*

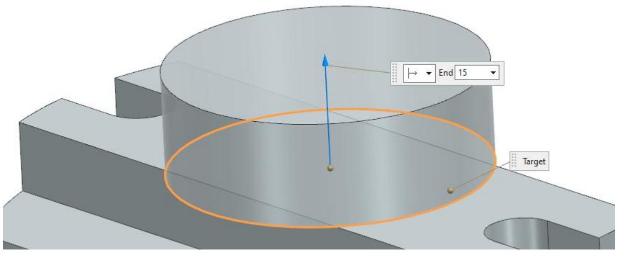


Figure 28 *The 3D model of the cylinder made in NX.*

The final stage involves making the bore of the part, using the *Hole* command, with dimensions Ø30 mm and a length of 30 mm, respectively Ø24 mm and a length of 15 mm, the bore type being *Counterbore*, Figure 29.

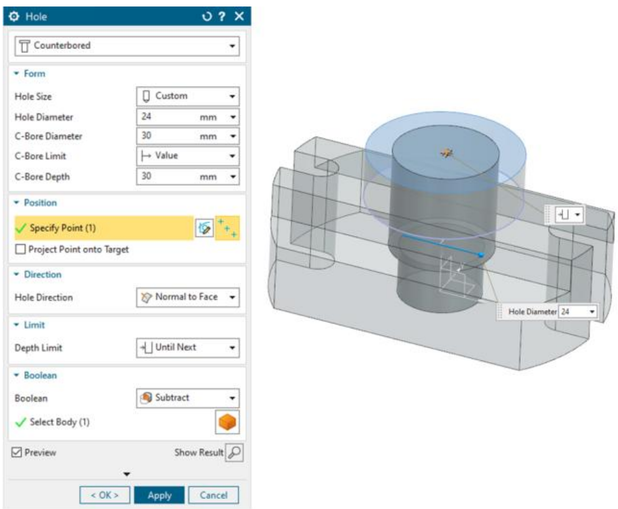


Figure 29 *Constructing the bore of the part in the NX program.*

Thus, Figure 30 shows the final three-dimensional model of the fixing plate part, created using NX.

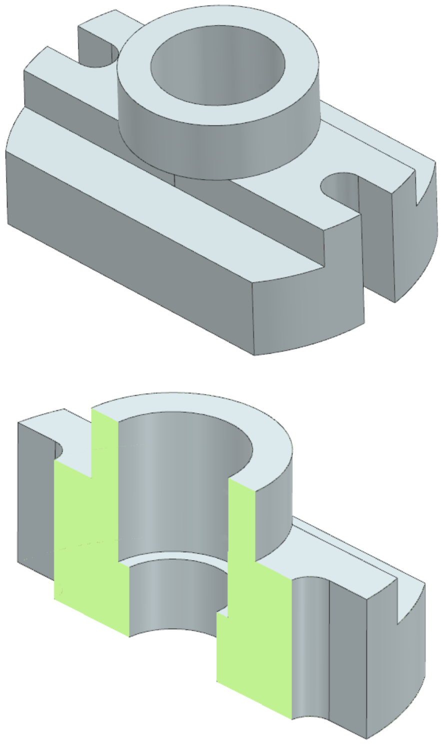


Figure 30 *The 3D model of the fixing plate part made in the NX program.*

4. CONCLUSIONS

The paper proposed the graphical modelling for different types of parts with simple surfaces, using three different design programs, namely: Autodesk Inventor, CATIA and NX.

Initially, the 2D models of the parts are presented, these being made in the AutoCAD program. Subsequently, each part will be explained in detail, to illustrate the design method in three modelling programs.

The main objective is to highlight the facilities and characteristics of each program, giving users the opportunity to understand the differences and advantages of each design solution. In addition to these aspects, the efficiency of each solution in terms of the speed of the design process will also be analysed.

The comparative analysis of the modelling of the two parts in Autodesk Inventor, CATIA and NX shows that the differences in the time required to create the models depend both on the structure and organization of the commands in each program, as well as on the user's level of familiarity with the respective CAD program. Thus, the work speed is not determined exclusively by the software, but is also significantly influenced by the operator's experience, his preferences and the way he uses the tools provided.

The final purpose of the paper was to familiarize users with as many computer-aided design (CAD) tools as possible and to provide them with solid criteria for choosing the optimal program depending on the complexity of the parts and time requirements.

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