COMPUTER AIDED DESIGN PROGRAM "ÇİZEN" AND DESIGNERS INTERACTION AND EFFICIENCY ANALYSIS; DEVELOPMENT OF INTELLIGENT INTERFACE SUGGESTIONS

A Thesis Submitted to
the Graduate School of Engineering and Sciences of
İzmir Institute of Technology
in Partial Fulfilment of the Requirement for the Degree of

MASTER OF SCIENCE

in Industrial Design

by

Gül Çiçek ZENGİN BİNTAŞ

January 2020

İZMİR

We approve the thesis of **Gül Çiçek ZENGİN BİNTAŞ**

Examining Commitee Members:

Prof. Dr. Önder ERKARSLAN
Department of Industrial Design, İzmir Institute of Technology

Prof. Dr. Ahmet Nuri BAŞOĞLU
Department of Industrial Design, İzmir Institute of Technology

Assist. Prof. Can ÖZCAN
Department of Industrial Design, İzmir Economy University

9 January 2020

Prof. Dr. Önder ERKARSLAN
Supervisor, Department of Industrial Design, İzmir Institute of Technology

Prof. Dr. Ahmet Nuri BAŞOĞLU
Head of Department of Industrial Design

Prof. Dr. Mehtap EANES
Dean of Graduate School of Engineering and Sciences

ACKNOWLEDGMENTS

I would like to express my thanks to my advisor Prof. Dr. Önder Erkarslan for his guidance through this thesis work. I am grateful to my parents İdris and Şerife for their endless love and support. I also want to thank my husband Mustafa for always being my best friend even while criticizing my work and giving support for my survey.

I would also like to thank to Prof. Dr. Bilgin KAFTANOĞLU for his contribution to literature by first CAD industry and research in Turkey.

ABSTRACT

COMPUTER AIDED DESIGN PROGRAM "ÇİZEN" AND DESIGNERS INTERACTION AND EFFICIENCY ANALYSIS; DEVELOPMENT OF INTELLIGENT INTERFACE SUGGESTIONS

Software is not only a rapidly developing and indispensable tool today but also the most prominent driving force of digital transformation. Thanks to the innovation studies carried out in the software of these programs, the productivity growth is achieved in the design processes, and the errors arising from the design could be reduced to a minimum. In Turkey, since 2010, working on CAD development have been initiated; the domestic and national program called “ÇİZEN,” has been developed for sheet metal forming design.

In this study, new techniques and innovative methods were performed in the ÇİZEN program, which was developed and personalized specifically to a particular sector and then, User Experience Survey was conducted. The study aimed to design a simple and intelligent interface by clearly determining the needs and expectations of sheet metal designers and the user's program experience.

Within the scope of this objective, user experience surveys were conducted with the participation of 14 users. In this context, the users were asked pre-configured and open-ended questions, and by giving the users specific tasks, the focus points in the program were revealed with heat maps through the Eye Tracker device. As a result of the finalization of pre-test studies and obtaining eye-tracking data, the final-test phase was initiated; and after the general opinions about the program were received, System Usability and Nonverbal Pictorial Scale research was conducted. The data obtained from the research results were interpreted, evaluated, and suggestions were developed under three main headings: productivity, simplicity/functionality, and interface.

ÖZET

BİLGİSAYAR DESTEKLİ TASARIM PROGRAMI "ÇİZEN" VE TASARIMCI ETKİLEŞİMİ VE VERİMLİLİK ANALİZİ; AKILLI ARAYÜZ ÖNERİLERİNİN GELİŞTİRİLMESİ

Yazılım hızla gelişmekte olan ve günümüzün vazgeçilmez bir aracı olmakla birlikte dijital dönüşümün en temel itici gücüdür. Ürünler istenilen özelliklerde ve ölçülerde Bilgisayar Destekli Tasarım programları ile tasarlanmakta ve üretime hazırlanmaktadır. Bu programların yazılımlarında yapılan inovasyon çalışmaları ile tasarım süreçlerinde büyük oranda verimlilik sağlanmakta ve tasarımdan kaynaklı hatalar en aza indirilebilmektedir. Türkiye'de yerli ve milli bir programın olmayışı ve lisanslama ücretlerinin yüksek olması nedeniyle CAD programlarının kullanımı yaygınlaşamamıştır. 2010 yılından itibaren Türkiye'de de CAD geliştirme çalışmalarına başlanmış sac kalıp tasarım sektörüne özgü yerli ve milli program ÇİZEN geliştirilmiştir.

Endüstri 4.0 ile birlikte ürünün tasarlanması, üretilmesi, geliştirilmesi gibi süreçlerde tamamen insan odaklı, kişiselleştirilmiş teknolojiler ortaya çıkmaya başlamıştır. Bu çalışmada, belirli sektöre özgü geliştirilen ve kişiselleştirilen ÇİZEN programında yeni teknolojiler ve inovasyon yöntemleri kullanılarak kullanıcı deneyimi araştırması yapılmıştır. Araştırma sonucunda sac kalıp tasarımcıların ihtiyaçlarının, beklentilerinin ve program deneyimlerinin net olarak belirlenerek yalın ve akıllı bir arayüz tasarlanması amaçlanmıştır.

Bu amaç kapsamında 14 kullanıcı ile kullanıcı deneyim araştırması yapılmıştır. Araştırmada önceden yapılandırılmış ve açık uçlu sorular sorulmuş, kullanıcılara belirli görevler verilerek Eye Tracker cihazı ile programdaki odak noktaları ısı haritaları ile ortaya çıkarılmıştır. Ön-Test çalışmalarının sonlanması ve göz izleme verilerinin elde edilmesi sonucunda Son-Test aşamasına geçilmiş program hakkında genel görüşler alındıktan sonra Sistem Kullanılabilirlik ve Sözsöz Resim Ölçeği araştırması yapılmıştır. Araştırma sonuçlarından elde edilen veriler yorumlanarak değerlendirilmiş ve productivity, simplicity/functionality and interface olmak üzere üç ana başlık altında öneriler geliştirilmiştir.

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LIST OF ABBREVATIONS

AR	Augmented Realty
APT	Automatically Programmed Tools
CAD	Computer-Aided Design
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing
CNC	Computer Numerical Control
FEA	Finite Element Method
KOSGEB	Small and Medium Enterprises Development Organization of Turkey
NC	Numeric Control
R&D	Research and Development
SME	Small and Medium Enterprises
SUS	System Usability Scale
TTGV	Technology Development Foundation of Turkey
TÜBİTAK	The Scientific and Technological Research Council of Turkey
UKUB	Tool Manufacturers' Association of Turkey
UX	User Experience
VR	Virtual Realty
2D	2Dimensional
3D	3Dimensional
MIT	Massachusetts Institute of Technology

CHAPTER 1

INTRODUCTION

This chapter will explain the basis and the aims of the research. Therefore, background, definition of the problem, aim, route, method and the significance of the study is explained and open questions asked at the beginning of the research, and the structure of the thesis is summarized.

1.1. Background

Until today, Turkey was only among the countries that use the Computer Aided Design (CAD) program. In recent years, however, CAD program and plug-in smart applications have been developed in Turkey, which increase productivity in design. However, while many domestic programs are under way, there are still not enough initiatives in 3D (3-Dimensional) program development. In this context, this study deals with the first domestic and national CAD program "ÇİZEN" which was developed and commercialized in 3D for sheet metal forming design in Turkey.

ÇİZEN was developed by MUBİTEK Design Informatics, a design company like the companies where other programs of foreign origin were developed. MUBİTEK has been engaged in product design and sheet metal forming design activities in the automotive sector since 2005 and has been doing innovation in design and software with KOSGEB R&D supports since 2007. Firstly, in the “Intelligent Design” concept in sheet metal forming design, it developed a plug-in program that works with CATIA V5, which increases the design efficiency by approximately 35%. This program is used by the main automotive industry (Tofaş) and various automotive supplier industries. During the development process, it has the know-how to make a difference to the existing CAD programs in the sheet metal forming design industry. Therefore, as of 2010, TUBITAK-TEYDEB supports program development studies. In this process, Prof. Dr. Bilgin Kaftanoğlu's academic studies and experiences who has done studies on the first CAD and sheet metal forming design sector in Turkey were utilized. ÇİZEN program has been currently developing for small industries (SME's). The goals of the program are:

- To replace Çizen Program with CATIA V5 in sheet metal forming design companies and other SME's.
- To develop a program that makes faster and correct design than other programs
- To sell 100 licences until 2018.

But long-term R&D work is required to achieve this. So Mubitek has been developing the program since 2010. After program's first lunch (in 2014), Mubitek took so much feedback from users for the program. Many of the feedback contained problems about the program.

Developing a software like this is a value added. Despite the problems of the program, thanks to value added, Mubitek in 2014, was awarded with the ÇİZEN and Intelligent Design application programs Microsoft Innovation Special Award, in 2015, Turkey Exporters Assembly Information Technology R&D 1st prize, the same year TUBITAK and TTGV Technology Awards Finalist, the Ministry of Industry in 2016. Productivity 2nd Prize and Automotive Project Market Honorable Mention. In 2019, It became the finalist of the Innovation League.

1.2. Definition of the Problem

Parallel to the developing technology, as in all sectors, the design sector is experiencing a major change. Especially with Industry 4.0, smart applications and advanced CAD programs are introduced in design technologies. New generation CAD programs give the designer speed and provide great advantages in terms of quality and visualization. With the development of smart applications in CAD programs, product-based designs are made even faster, error-free and can be prepared to provide the necessary information flow for production. Therefore, the use of design programs in the sector is increasing.

Design programs used in Turkey are programs of foreign origin who has a history of nearly 60 years. These programs were designed by designer companies who were aeronautics and space companies. Such as, CADAM software by Lockheed Company, McAuto (Unigraphics) by McDonnell Douglas Company, CATIA by Dassault. Since then, programs have been developed to improve their capabilities, efficiency and adapt to current technology (Kaftanoğlu, 2005). The most popular CAD programs on the market today are SolidWorks, Inventor, Catia, AutoCAD, VisiCAD, UG NX and so on. There are approximately 15,000 License usage. Growth and Future Expectations and Competition Analysis forecasts for 2018-2026 are expected to yield 6.8% growth over the global CAD market forecast period.

Quality, efficiency requirements and cost pressure require standardization and automation in design. As a result, companies have started to customize their own design standards, in-house 3D library, sets and various CAD applications with existing applications (Bintaş, Öz, 2016). Particularly due to cost and time pressure, applications that provide sector-specific productivity in sheet metal forming design have become necessity and obligation. The most important point of the ÇİZEN program, which was developed based on this need and necessity; the design not only physically describes the product, but also the material information required for production, purchase order codes, production machines, etc. information. Especially in sheet metal forming production, which is a project based production model, information flow from design to electronic direct production information systems flow planning will provide faster, effective and continuity. In this way, the designer will provide an efficient data flow between the entire eco system, which supplies standard materials and manufactures CNC machining.

In order to develop a CAD program has to use a geometric modelling kernels such as ACIS, Parasolid, C3D, OpenCASCADE. ÇİZEN has been developing C++ programming language and OpenCASCADE CAD kernel. Program interface design made with QT programming. This program has been developing to have the following features:

- Basic drawing and solid modeling functionality
- Models for an easy design
- Wizards for several types of sheet metal forming design in order to save time
- sheet metal forming design templates for frequently used design types
- Specialized libraries for sheet metal forming design
- Dedicated project management flow (Bintaş and et all, 2011).

The program was first launched in 2014, 25 of which are licensed in the automotive sector sheet metal forming design, 15 of them are used in trial version and nearly 650 of them are accessed as .step data viewer.

In spite of all the efforts of the program, which has the vision of replacing the programs belonging to the very big companies that have existed in the market for years, in 2018, it could not reach the targeted 100 units/year licensing. The reason for not achieving the target was evaluated by the company as a result of the feedback received from the users and three main reasons were reached. These are;

- Software problems caused by CAD kernel (OpenCADCADE) used in the infrastructure of the program are experienced. In order to solve this problem, within the scope of TUBITAK TEYDEB priority areas support program, the company initiated a Design 4.0 software project which will last for 36 months. In this context, the CAD kernel of the program is changed.
- The interface of the program is prepared by the employees of the company. No user experience surveys were conducted.
- Designers' needs and design methods have been developed only in-house experience. No comments were received from targeted users.

In order for the program to meet the needs of the sector, to achieve the targeted commercial success and to ensure its competitiveness, it is necessary not only to improve the infrastructure but also to develop user-oriented, smart and simple interface in line with the needs of the sheet metal forming designers by analyzing the command efficiency.

The main problem of the thesis consists of analyzing the efficiency of the program and conducting user experience research and obtaining the necessary suggestions for designing a user-friendly, smart and simple interface for the needs of the sheet metal forming designers. Because simple and intelligent design is focused on eliminating waste and continuously improving auxiliary works (Bintaş, 2011).

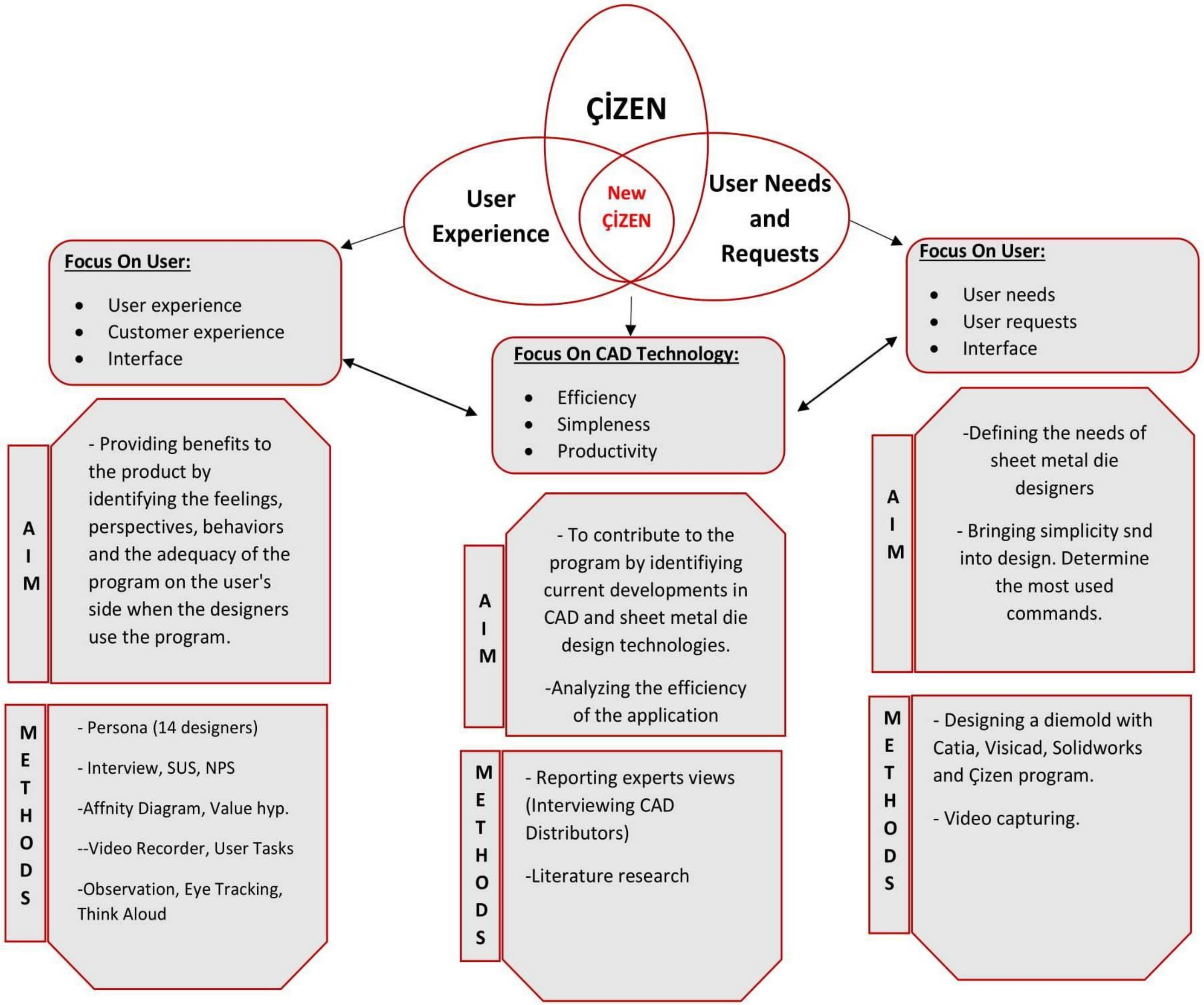

Figure 1. Relevance and Relation of the Thesis

1.3.Aim of the Study

Design and the industry is the base of the economy in all countries of the world. Every day we use mobile phone, automobile, television, air conditioning, etc. Considering the mass production of all plastic and metal parts of the products with the help of separate sheet metal forming, it becomes easier to understand the place of sheet metal forming production in technology and economy. For example, for the production of an automobile 300-350 parts and about 1200 sheet metal formings are required. The process of designing a sheet metal forming takes between 30 and 400 hours depending on the dimensions and properties of the part. It is a high-cost service in every country of the world because it has a direct effect on the final product, sheet metal forming design and manufacturing is highly detailed, precise, technically crafted, computer modeling with advanced technology is required and heavy industrial machines are required to be used.

Product design is the first stage of sheet metal forming production. The finished product is designed with 3D CAD applications in desired specifications and dimensions. Product margins and finished 3D drawings are created. With the 3D Cad data prepared, base sample / prototype is prepared and presented to the manufacturer for inspection before the production of the sheet metal forming. The manufacturer gives the approval when it is ready for mass production by detecting the missing points and conveying the necessary additional requests. Along with Industry 4.0, there are many needs and expectations in the sheet metal forming design and production sector. These are shorter product launch times, lower cost and higher quality production, innovative, flexible, modular, intelligent production that allows the reuse of materials and equipment (Bintaş, Öz, 2017). The importance of new and innovative product design is increasing day by day in international market conditions. In this context, competition is no longer experienced in the final product, but in the process leading to the final product. When the process is designed correctly, ÇİZEN and similar smart design applications are among the programs that will help companies to increase their position in the market. These applications, which gather all the parameters related to sheet metal forming design under a single roof, eliminate all unexpected problems and contribute to the finalization of designs in a shorter time than required (Bintaş, 2011). The aim of the thesis emerges at this point. Our aim is to redesign innovative design methods and program interface in accordance with the targeted sector. Within the scope of this aim, determination of the needs and expectations of sheet metal forming designers and analyzing the efficiency of ÇİZEN is to develop a simple interface suggestions with high competitiveness features. This will contribute to technical infrastructure, CAD literature, commercial and R & D gains both on country and company basis.

1.4.Method of the Study

The research subject of the thesis was conducted in three scopes. The first is efficiency analysis. In order to perform efficiency analysis, firstly a CAD program to be compared must be selected. Therefore ÇİZEN program of similar density and sheet metal forming used in the design of the 2,500 licensed users in Turkey over CATIA V5 program has been selected. Within the scope of efficiency analysis, the results in the CATIA V5 program were analyzed on the basis of application step by using the commands in ÇİZEN and the results were presented in tables.

The second scope of the research is about determining the most common commands in sheet metal forming design and the design logic of sheet metal forming designers. The aim of this research is to provide simplification and intelligent systems for sheet metal forming design in ÇİZEN interface. In this context, the mold ((Please see Appendix B), which contains the most detailed works, was determined and designed in Catia V5, Solidworks, VisiCAD and Çizen program (Please see Appendix I). Each stage of the designs were videotaped and analyzed, and the logic and capability of sheet metal forming design of the programs were demonstrated. As a result of the data obtained here, question headlines were created for the lean and smart interface application and the users were asked open-ended in the pre-test application (Please see Appendix C).

The third scope of the research consists of ÇİZEN-User experience. For this study, pre-structured open-ended questions were asked and 16 tasks ((Please see Appendix E) were determined through a model drawing and the application was made. The application was made by using eye-tracking, stopwatch and, usability test notes (Please See Appendix G) observation methods. With used eye-tracker device, interval duration, visit duration, visit count and click count metrics were measured. After that, post-test (Please see Appendix F) questionire was done to the participants.

With this research, the technical infrastructure and interface of ÇİZEN has been evaluated from the user's point of view, and results have been obtained that will contribute to its commercialization and widespread use in sheet metal forming design sector.

Firstly, persona, professional and CAD program experience were obtained from the participants. Then, it was evaluated as practical in ÇİZEN program. In this context, application was made through a drawing and diameter and thickness values of the model were asked to be connected to the parameter. Here, it was observed how many minutes the drawing was completed, the points it had difficulty in, and which command lost time mainly. Afterwards, interface orientations, use of mause, adequacy of 2-Dimensional (2D) and 3D commands, their opinions about the content and formation of the model tree, the use of parametric modeling method, their opinions about special 3D catalogs in the interface were asked. Finally, suggestions and comments on the overall appearance, design and usability of the interface were requested. In order to understand the sheet metal forming design logic, the process steps they followed while designing, the commands they used most and the points they were forced to do, the revisions in the design changes and the points they needed were asked.

The research questions were designed as qualitative and quantitative research. Questions are focused on user explanations and program experiences. The reports obtained from the interviews constitute the main data of the study. The research was conducted with a total of 14 users. First of all, pilot application was made with 4 designers in MUBİTEK. With the pilot application, the achievement of the purpose of the questions was tested and improved.

In the selection of the users, one of the qualitative sampling methods, homogeneous sample which was suitable for the study purpose was selected. This method involves a more detailed study with a focus. The focus groups generally consist of 4-12 participants, with specially identified similar experiences. Open-ended questions are asked and interviews are conducted using techniques such as brainstorming (Marchall, 1996). In order to achieve the aim of our thesis, face to face interviews were preferred with the participants in the focus groups. In this context, 10 designers working in sheet metal forming design in Bursa province where the automotive sector mainly operates were included in the research. For that, i prepared a persona form (Please See Appendix H) in which writes the qualifications requried for researh.

In 83 case studies conducted by Nilson Norman Group, which is the world leader in Research Based user Experience, it was seen that usability studies gave 5 users the most optimum results. However, in some studies, it was found that they achieved optimum results with 8 participants and with 12 participants. Therefore, 10 participants were included in the research. But i wanted to include one more participant for not getting to risk while analyzing eye traking records. The study lasted between 45 minutes and 1.5 hours. All studies were digitally recorded.

1.5.Research Questions

The aim of this study is to contribute to increase efficiency in infrastructure and interface usage of ÇİZEN program and to ensure its prevalence in sheet metal forming design. In order to achieve this goal, the research questions were grouped under three main headings: (i) productivity, (ii) simplicity and functionality, (iii) Interface.

These investigations were done by answering the research questions below:

(i) Productivity

- Compared to other CAD programs (CATIA V5), how is the efficiency of ÇİZEN on command basis (edit toolbar, file menu, product menu, Part Menu, Sketch Menu)?
- How is ÇİZEN's sheet metal forming design efficiency compared to other CAD programs?

(ii)Simplicity and functionality

- What are the most commonly used and needed commands in sheet metal forming design?
- Are the ÇİZEN'S commands sufficient to design the sample model identified in the research? If not enough, what commands should be added / edited?

(iii) Interface

- Are the instructions in the interface sufficient? If not, what are the recommendations?
- Do the designs of the icons used in the design of the example model reflect the command functions?

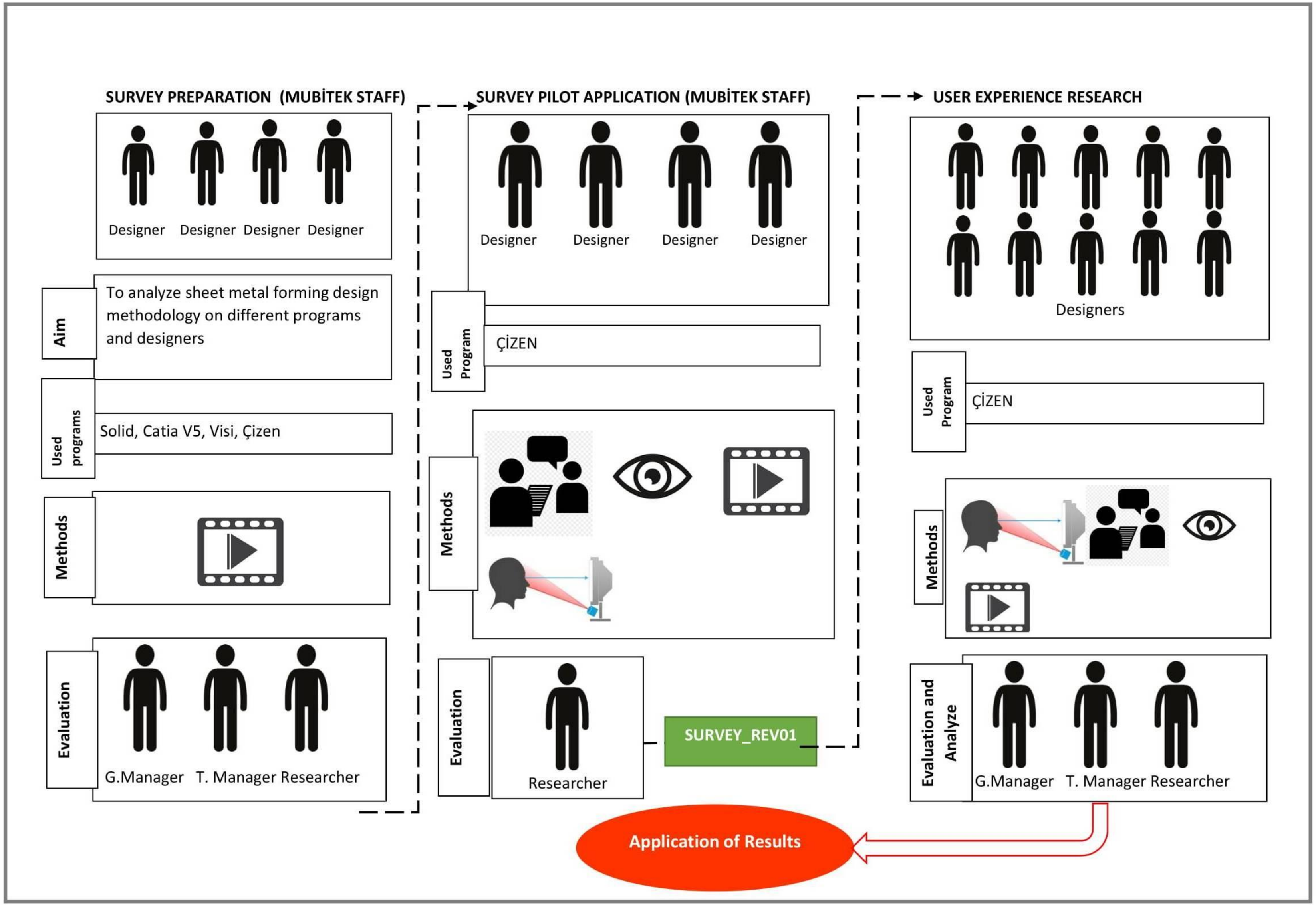

Figure 2. User Research Plan

1.6.Significance of the Study

The software is a great tool that is rapidly developing and has become one of the indispensables of today. Especially for software that is at the key point of production such as design, software brings great cost losses due to security problems and high licensing fees.
The fact that foreign software companies can easily obtain the most critical information if they wish, poses a great threat to companies and public institutions. Without a doubt, the main driving force of digital transformation is the software industry. The basis of achieving value-added production and competitiveness in the economy is hidden in the software and design sector.

The use of Computer Aided Design program in Turkey is about 20 years overdue by member states of the European Union. The lack of a domestic and national program, the lack of attention to technology and the high licensing fees have prevented the use of CAD programs to spread. Because of the high license fees, unlicensed use is still common today. In particular, the state party since 2010 R & D, innovation, with the increasing importance of software and the CAD design program in Turkey has begun nationalizing their work. The government supported the development of these programs, especially due to the risk of information security in overseas programs. Among the developed programs, ÇİZEN program became the first commercialized program. However, the prevalence of the program could not be achieved. In this context, it is necessary to develop the features and capabilities according to user opinions. Our thesis is of great importance in order to ensure that the program, which is a high value-added and strategic product, can compete with international examples, to ensure information security, to reduce foreign exchange expenses and to ensure user satisfaction. Also this study was converted to TÜBİTAK Ardeb Code 1005 project. If this Project is approved, our studies will continue and the interface of the program will be completely changed with the results obtain.

1.7.Limitations of the Study

CATIA V5 was preferred for the efficiency analysis in the research section of the thesis because of its similarity to the ÇİZEN program and the sector in which it is used. As

CATIA V6 is not yet widely used, Version 5 has been analyzed. Within the scope of the analysis, the commands of Edit Toolbar, File Menu, Product Menu, Part Menu and Sketch Menu which are available in both programs were compared. Mold design has been made in Catia V5, VisiCAD, SolidWorks and Çizen programs in order to reveal sheet metal forming design methodologies in different programs. These programs are used in sheet metal forming design in the sector. The user experience research was carried out using the Tobii X2 60 Eye Traker with 16 tasks identified on a model drawing. The participants were selected on the basis of volunteering among the designers working in the companies that design and manufacture sheet metal formings in Bursa.

1.8.Structure of the Thesis

The first part of the thesis; problem definition, purpose of study, method, importance, limits and research questions of research. In the second part, the concept of computer aided design, its history and methods, sheet metal forming design and related literature research are given. In the third part, the efficiency analysis studies in CATIA V5 and ÇİZEN, in the fourth part, the user experience research and result takes place. In the last section, recommendations are given.

Table 1. Structure of Thesis

Exploration	**Chapter Content**	**Question to be Answered**	**Number**
Exploration of Literature	Design Description Computer Aided Design Methods and History Importance of Computer Aided Design Design and Technologies	What is the importance of using CAD program in design? What are the studies in the field of design technologies? What vision is there in the future?	CHAPTER 2
Exploration of efficiency analysis of ÇİZEN	Comparison of ÇİZEN user interface, product menu, part menu, sketch menu with CATIA V5	Compared to other CAD programs (CATIA V5), how is the efficiency of ÇİZEN on command basis (edit toolbar, file menu, product menu, Part Menu, Sketch Menu)? How is ÇİZEN's sheet metal forming design efficiency compared to other CAD programs?	CHAPTER 3 and Appendix I
Exploration of user experience	Understanding user needs and requests	What are the most used and needed commands in design? Are the commands of ÇİZEN sufficient to design the sample model determined within the scope of the research? If not enough, what commands should be added / edited? Are the instructions in the interface sufficient? If not, what are the recommendations? Do the designs of the icons used in the design of the sample model reflect the command functions? How is results of the Research?	CHAPTER 4
Contributions	Discussion of the framework Conclusion	What are recommendations?	CHAPTER 5

CHAPTER 2

THEORETICAL BACKGROUND

In this chapter will explain design, design process, computer aided design and its history. This chapter also includes general information about sheet metal forming design and technologies.

2.1. Design Description and Process

According to the Turkish Language Institution, design is the framework that designs the paths and procedures to be followed in various periods of a research process. Design is an important factor in producing a product. The design stage of the product develops in the human mind and then gains an objective reality. Design is not just a product, a case or an aesthetic. Design is a process. This process can be defined as planning and organizing an action to predict and achieve the intended outcome Altın (Altınok, 1987). All the works from creative thinking to drawing documentation of the product (Rembold, 1985).

Design, as epitome of the goal of engineering, facilitates the creation of new products, processes, software, systems and organizations though which engineering contributes to society by satisfying its needs and aspirations. Every field of engineering involves and depends on the design or synthesis process, which allows us to fulfill needs through the creation of physical and/or informational structures, including machines, software and organizations (Suh, 1990).

At the stage of defining the need, the existence of a problem is recognized by a person and as a result action is taken to correct the problem. The move is to emphasize the need for something to be noticed by any engineer, marketer or other person involved in any defects found in a manufactured part.

Shigley described that the process of designing a product consists of 6 steps.

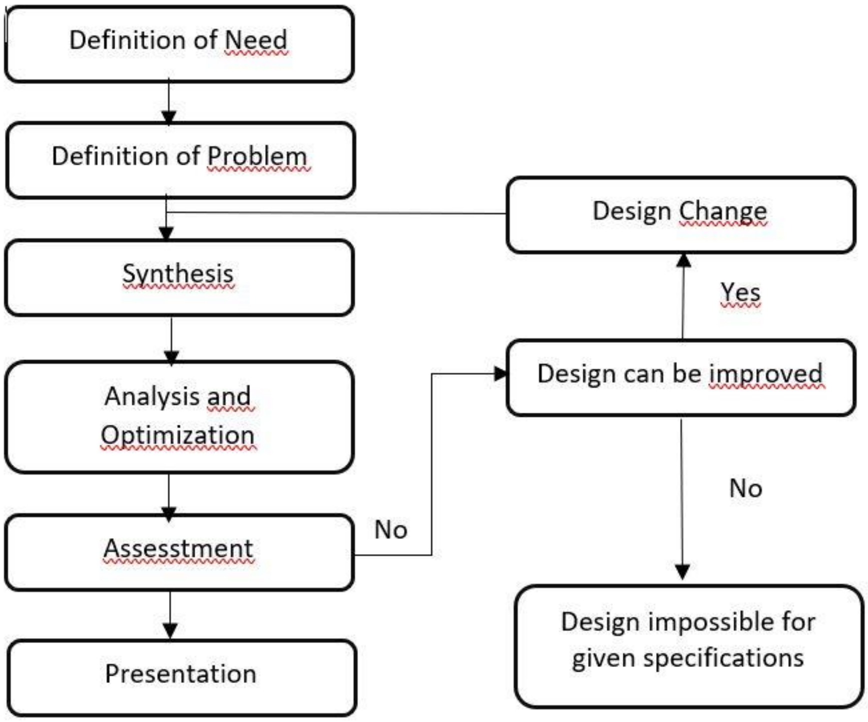

Figure 3. Design Process Stages

The main point in defining the problem is the specification of the object in its design. These specifications in object design include physical and functional properties, cost, quality and performance.

Synthesis and analysis are more related to the design process. A conceptualization of a component or a whole system by the designer is the subject of analysis. Synthesis is a repetitive step until the designer feels he has achieved the optimum design. In the evaluation step, there are issues related to measuring the state of the design according to the specifications in the problem description, ie fabrication features and testing of prototypes. Quality, reliability and other criteria are also considered at this stage. According to Shigley, the final stage of the design process is about the presentation of the design. It contains all the documentation of the design, drawings, material properties and lists. These design documentation basically provides for the creation of a design database.

2.2. Computer Aided Design Concept and Importance of Using CAD Program

As the name implies, Computer Aided Design is the realization of design processes with the help of computers (Singh, 1996). The use of computers in design involves the creation of the model in accordance with the defined desires of the design and the process of achieving the best. Computer Aided Design systems provide a more efficient and efficient working environment in design due to the fact that computers benefit from fast computing power, information storage and new information generation facilities compared to the classical design methods.

In the design process, the parts to be drawn are designed and modeled in two or three dimensions, and the technical drawings are created with CAD software. CAD is the blending of human and machine, working together to achieve the optimal design and manufacture of product (Amirouche, 2014).

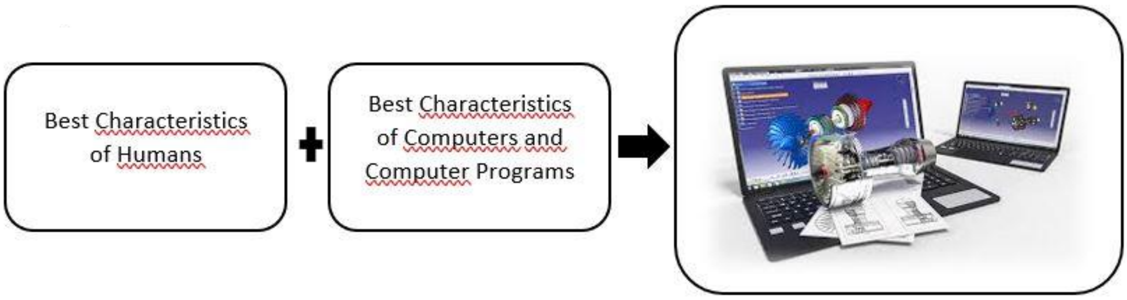

Figure 4. Characteristic of CAD

CAD is the act of utilizing computer technologies in order to achieve goals such as shortening time, increasing quality, increasing efficiency and reducing costs during the design process (Narayan, 2008). In addition to the traditional method, the designer uses three The CAD system consists of a hardware, software and user trio-dimensional programs such as Inventor, Solidworks, Catia, AutoCAD, NX, Fusion 360 and so on and can realize the design of virtual environments. It allows the design results to be used directly in the CAM environment, integration of design and manufacturing. CAD design results are transmitted to CNC (computerized numerical control) part programming stage and the part is manufactured and CAD / CAM integration required for automation is provided. A CAD can have many different drawing applications. Software is avaliable that will produce many types of drawing. Some of common drawing applications of CAD include: mechanical, architectural, electrical, electronic, piping, civil and 3D (Betoline, 1985).

The CAD system consists of a hardware, software and user trio:

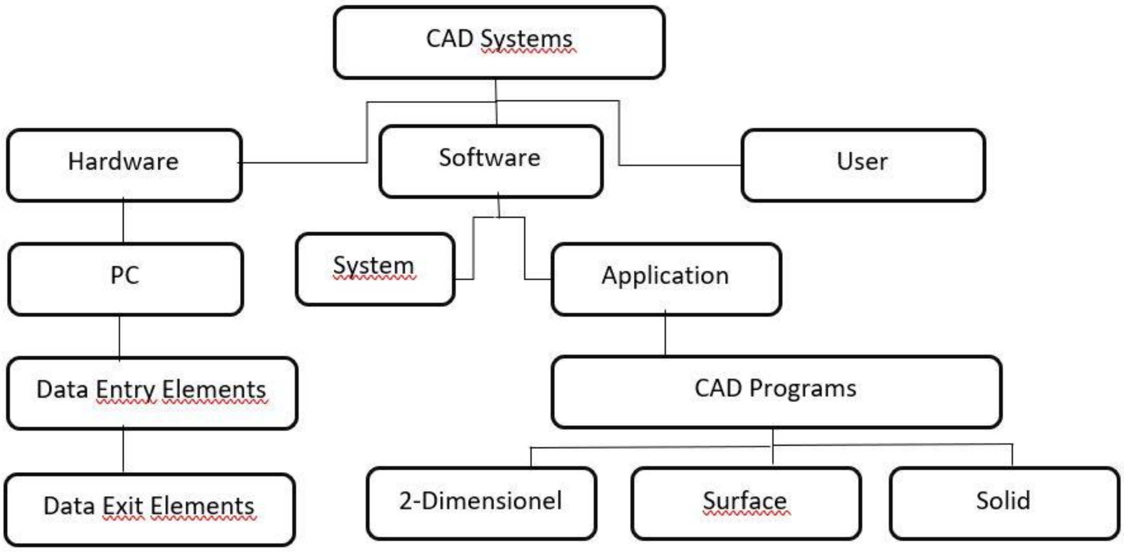

Figure 5. Elements of CAD Systems

There are many advantages of using CAD over the traditional tools. Although some advantages are quite apparent, others are more difficult to recognise. Following is a summary of advantages of CAD:

- Faster production of graphics
- Modifications are easier to make
- Shorter time for design of similar parts
- Improved accuracy of the design
- Better line weight and consistency
- Faster placement of text on drawing
- Automatic to create symbol library
- Creation of isometrc and axonometric views of a 3D model (Filiz, Dereli, 1997).

2.3. History of Computer Aided Design Program

The histroy of CAD can be viewed as a dialogue between the engineer and the computer. Over the last several years, that dialogue has changed (Filiz, Dereli, 1997).
CAD grew from te needs of automotive and aerospace industries in the fifties. The drive behind the computer systems revolution was production led and that the CAD systems that also evolved were also based on that premise (Pugh, 1991). The general structure of CAD systems is based on the interactive computer graphics (ICG) system. In these systems for the user, the user is the designer himself; it provides data communication and allows the creation of various images and drafts on the screen by giving commands to the computer through various input formats (Keskinel, 1985). CAD is generally used for the creation, renewal, analysis and optimization of a design (Narayan, Mallikarjuna, Sarcar, 2008). In the last 40 years, software and hardware technologies have developed rapidly in the fields of computer aided design and modeling. The programs used in the fields of engineering, aviation and automotive have been specialized over time; Cinema 4D, Form-Z, Maya, 3DS Max, Blender, Lightwave, Modo, SolidThinking, Rhino has developed software for art and design in recent years, creating effective production opportunities for artists and designers.

The history of computer use in design and manufacturing engineering dates back to 1946, when the first computers that followed Electronic Numerical Integrator and Computer emerged. At that time, finite element analysis and NC programs worked in batch processing without graphical features. The first NC milling machine was developed with the research project of the US Air Force to the Massachusetts Institute of Technology (MIT) between 1949 and 1952. In the same years, the first graphical display at MIT was connected to the Whirlwind computer. As NC machines and NC drawing machines were introduced to the industrial market in the following years, work began to facilitate the programming of these machines used in the production of complex parts in the aircraft industry. The term Computer Aided Design was introduced for the first time during the studies conducted at MIT between 1955 and 1959, which resulted in the development of the first high-level NC programming language, Automatically Programmed Tools (APT) (Ross, 1960). The feature of the APT language was that the part geometry was described instead of the tool path. In the same years, G.C. The first industrial robot concept came into being in 1954 with a patent granted by Devol.

The first CAD system emerged at the academic level in 1963 with a doctoral dissertation by Ivan E. Sutherland at MIT. In the system called Sketchpad drawing, Sutherland demonstrated the possibilities of interactive operation on a refresh screen with appropriate programming techniques and data structures. This system is the beginning of computer aided design and also it has laid the foundation of computer graphics and flight simulation.

For the first time, the designer could press the monitor and draw directly on the screen with a lighted pen. Object-oriented programming and icon system were used in Scetchpad software. In the same year T.Johnson made the idea of a drawing board three-dimensional. In parallel with this study of Sutherland, a system called Design Augmented by computer which will be used in automobile design in IBM has been developed.

This system was introduced at the fall joint computer conference in 1964 and led to the expansion of many interactive CAD systems in the late 1960s. With this development, CAD became widespread in civilian use and had the opportunity to work in electrical, chemical and industrial engineering.

When 3D CAD programs were first introduced in the 1980s, most of the designs were made on 2D CAD tools or on a hand-drawing board. 3D modeling was generally limited to finite element analysis (FEA). On the production side, CAM programmers created a CNC machining profile by converting the 2D drawing from the design department into 3D.

In the 1990s, engineers began accessing desktop computers that were powerful enough to create 3D designs using a desktop 3D CAD package. However, 2D drawings were still the main means of communication between design and production. Mostly, 2D drawings were converted into 3D models by CAM programmers so that they could be processed in 3D CAM programs.

It was only in the late 1990s that design and manufacturing engineers shared 3D models, eliminating the need for remodeling for CAM. This was especially true in the sheet metal forming design and manufacturing industries. In these sectors, the tight integration between the designer's 3D model and the sheet metal forming maker's 3D sheet metal forming design has led to a fundamental change in sheet metal forming production and has significantly reduced sheet metal forming production times.

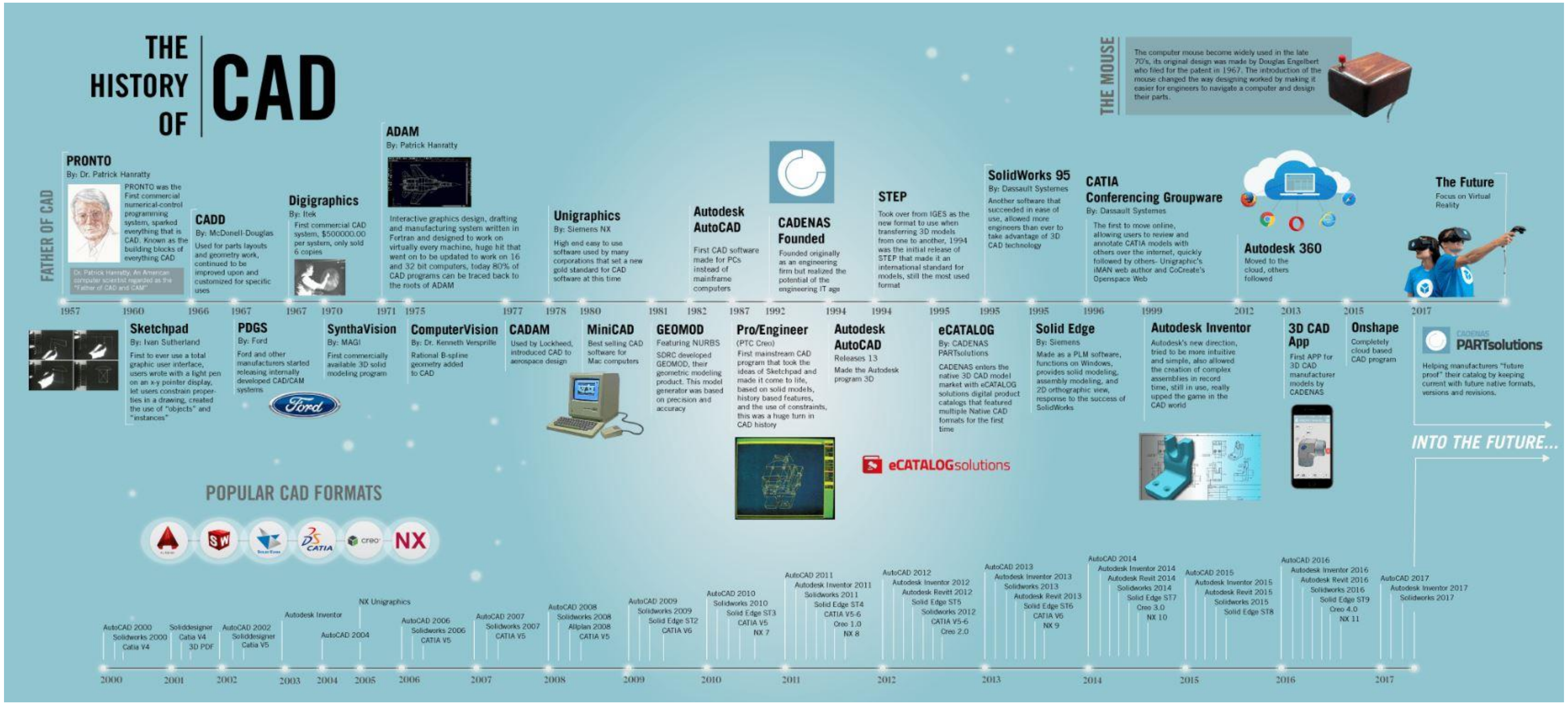

Figure 6. The History of CAD

2.4. Computer Aided Design Methods

CAD has evolved at a tremendous pace, from simple 2D draughting, to more complex 3D modelling, to the use of animation, parametric modelling and the linking with artificial intelligence (Allen and Kouppas, 2012).

When CAD was introduced in the late 1960s, it essentially provided an electronic drafting board for drawing in two dimensions. Through the 1970s CAD systems were improved to provide three dimensional wireframe and surface model. By the mid-1980s nearly all CAD products had true solid modelling capabilities. More recently, tools for kinematic, stress and thermal analysis have been seamlessly linked to the sloid model so that can be done along with modeling (Dieter, 1987). During the 1980s, CAD developers gave engineers increasingly sophisticated geometry to help them represent the 3D World. Wireframe modelling was the most rudimentary method for capturing this new World. Using the same geometric elements that made up a blueprint: points, lines, arcs and splines, engineers developed “stick-figure” representations of their designs. Surface modelling then allowed the engineer to add “skin” to his wireframe model. Finally solid modelling took CAD to the next logical step bye giving volüme to the surface design. With each of these advances, the engineer gave the computer more information of design (Filiz and Dereli, 1997).

2.4.1. Two-Dimensional Design

The evolution of CAD has followed an interesting path, migrating from aircraft to automotive to architecture to animation to artificial intelligence, gaining richness en route. Computer Aided Design also used to be known as Computed Aided Draughting, and this is an indicator of CAD’s two-dimensional origins. The first CAD packages were largely replacements of the drawing board, producing predominantly orthographic and isometric drawings. The huge advantage of CAD was its ability to duplicate elements allowing for quick modifications of part drawings, easy sharing of files, and dramatic reduction in the time taken to produce and edit drawings. There was also a key advantage in electronic storage of files, meaning that files could readily be duplicated and shared (particularly important when the design team are working in different locations and manufacture is occurring overseas). The need for plan chests full of drawings in each location quickly became a thing of the past. Thankfully, but also sadly for the loss of this artistry, those days are gone and technical drawings are now almost exclusively computer generated. This has dramatically accelerated the design detailing and development process. Many CAD packages now include standardised parts files, so designers can simply 'drag and drop' items into their design drawings.

For instance, standardised items such as doors, windows and various building panels can be retrieved from a directory and dropped into architectural plans; or in engineering, standardised pipes, tubes and mechanical fixings can be quickly inserted into drawing files. This not only speeds up the process of producing detailed drawings and plans, but also ensures dimensional accuracy.

Importantly, producing drawings in CAD means that the same computer file that produced the drawing can also be used to generate cutting paths for the manufacture of items, thus resulting in a far more efficient and accurate process. This translation from CAD draughting to digital making is often referred to as CAD/CAM, where CAM stands for Computer Aided Manufacture. Many 2D CAD applications now incorporate 3D capabilities, but the development of 3D CAD systems have, by and large, superseded their 2D cousins (Allen and Kouppas, 2012). Figure 7 shows orthographic 2D design example.

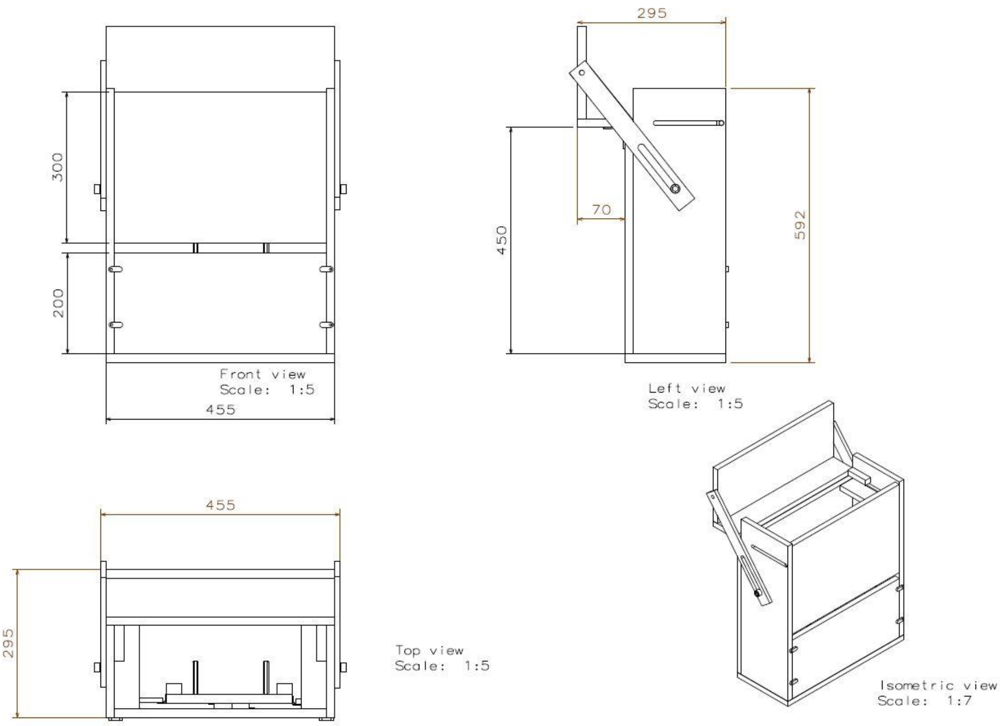

Figure 7. Two-Dimensional Design Example (Designed with Catia V5)

2.4.2. Three-Dimensional Design

The three-dimensional shape is conveyed with orthographic projections. This is little better in delivering information than a conventional hand-drawn portrayal. The advantage is in the ability to make easy drawing modifications and the ability to use predawn modules (Dieter, 1987). We can say that 3D Design includes three modelling types.

2.4.2.1. Wireframe Modelling

Wireframe models also called edge-vertex models are the simplest methods of modelling anda re most commonly used to define computer models of parts especially in computer aided drafting systems (Filiz, Dereli, 1997). Wireframe models are the easiest method of modeling and are used in the definition of models in computer aided presentation systems. The most important reason for this is the simple and easy creation of wire mesh patterns. At the same time, little time and memory is sufficient in relation to each other. Thus, this type of modeling needs can be met in low-configuration computers. At the same time, the wire mesh models are used as a projection that allows the NC machine to follow the cutting path. Wireframe models often are confusing and difficult to interpret because of extraneous lines. Wireframe models require less computer time and memory space and provide little information about surface discontinuities. Specifying point and lines in space creates them (Amirouche, 2004).

The wireframe method, which is used to represent a three-dimensional geometry, is based on the expression of the object in the three-dimensional space with border lines delimiting the object. There are no surfaces to identify the object, except for lines and curves indicating edges. Wireframe models are early examples of geometric models and their origins date back to the 1960s. Figure 8 shows Wireframe examples.

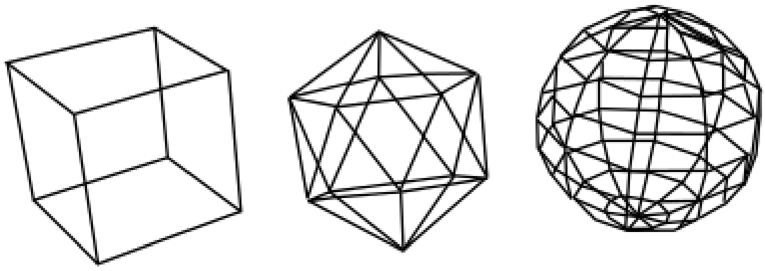

Figure 8. WireFrame Modellig Exapmles

2.4.2.2. Geometrical Modelling

A geometric model representing a component in computer memory may be a 2-dimensional or 3-dimensional type depending on the capabilities of the CAD system and the requirements of the users. It is not difficult to understand that a 2D model represents a flat part and a 3D model provides respresentation of generalised part shape. In general there are three types of modelling in common use to represent a physical object in CAD systems (Filiz, Dereli, 1997). Geometric modelling is associated with surface and solid modeling (Amirouche, 2004).

2.4.2.2.1. Surface Modelling

Surface models overcome many ambiguities of wireframe models. They define part geometries precisely and help produce numerically controlled machining instructions, in which the definition of structural boundaries is diffucult. Connecting various types of surface elements to user-specified lines creates surface models (Amirouche, 2004).

A higher level of sophistication in geometric modelling is the surface model which can overcome of ambiguities of wireframe models. A surface model can be built by defining the surface on the wireframe model. The surface model may be constructed using a large variety of surface features often provided by many CAD systems (Filiz, Dereli, 1997). With such a surface model the defination of the object is more complete and the model can be used with finite-element analysis and NC machining programs. Color-shaded images (rendering) may be created (Dieter, 1987).

The plane is the most basic feature to represent a surface element. More complex shapes can be defined by tabulated cylinders, ruled surface of revolution, sweep surface and fillet surface. Surface models do not actually represent the solid nature of parts. Because they contain no information describing what lies within the part interior. They can not be used as a basis for engineering analysis programs such as finite element model analysis for stress abd strain predictions (Filiz, Dereli, 1987). Figure 9 shows surface modellin example. This model was designed with ÇİZEN CAD Program.

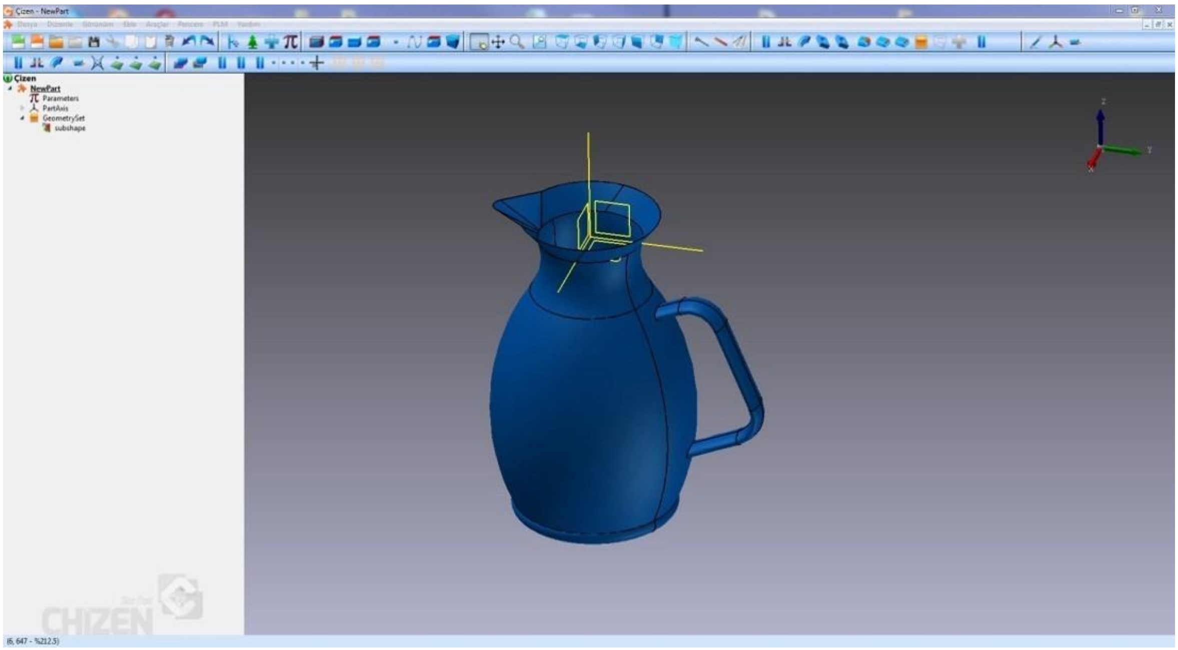

Figure 9. Surface Modelling Example

2.4.2.2.2. Solid Modelling

Solid models are recorded mathematically in the computer as volumes bounded by surfaces rather then as stick-figures. As a result, it is possible to calculate mass properties of the parts which are often required for engineering analysis such as finite element methods for engineering for interference checking (Filiz, Dereli, 1997). Solid models provide information about what is inside the 3D model as well as information about the surface of the object (Dieter, 1987).

A solid modelling system must unambiguously describe the three-dimensional shape of the object. It must be able to distinguish between the inside and the outside of the object and be able to support the calculation of mass properties such as wieght and moment of inertia (Dieter, 1987). Solid modeling is used define geometry and volume unambiguosly; it provides the ultimate way to describe mechanical parts in the computer. Unlike wireframe and surface modelling, solid modelling provides the accuracy needed for precise mechanical design. It has the potential to create a database that provides a complete description of the part to downstream applications. Solid model are constructed in two ways, with primitives or with boundary defination. Both of these methods develop complex geometries from successive combinations of simple geometric operations (Amirouche, 2004).

Solid modelling mathematical algorithms require a certain manipulation of geometry to obtain the desired shape. There are several techiques used fort he construction and editing of solid object. Some are promising for future developments, and others are currently in use and quite popular. These techniques include; boolean operation, sweeping, automated filleting and chamfering, tweaking, fleshing out of wireframes and projections. None of these techniques seems to be adequate by themselves, so the ideal solid model system should support several them. A common feature of the techniques considered in this section is the ability to do a number of modifications with minimum user input (Amirouche, 2004). Figure 10 shows solid model example. This model was design with ÇİZEN CAD program.

2.4.2.3. Parametric Modelling

The meaning of the parametric modelling is that parameters of the model can be changed, to modify geometry of the model. With developing technology, parametric design method has been developing in order to add flexibility to design. It is the most powerful technique to help you make topological changes in design (Khan, 2017).

Parametric design is about setting dimensions and tolerances so as to maximize quality and performance and minimize cost (Dieter, 1987). In a parametric design, the engineer selects a set of geometric constraints that can be applied for creating the geometry of the component. The geometric elements include lines, arcs, circles, and splines. A set of engineering equations can also be used to define the dimesiones of the component (Amirouche, 2004). Parametric design systems help users manipulate the lenght, angle, and pitch of a particular component. Moreover, the parametric system can produce designs that are more meaningful than those generated by traditional CAD systems. Designers, analysts, and production engineers can work with the same solid model, extracting an adding information according to need (Amirouche, 2004).

In these techniques, design process starts by parameterizing the important features of design models. Relationships between design parameters are defined as design constraints, which help to retain the model feasible during the modification of its design parameters (Khan, 2017). Figure 11 shows how parametric design can be do. This design made with ÇİZEN CAD Program.

Commonly, 3D surfaces in CAD are referred to as parametric surfaces, and they can be mathematically represented by Nurbs, or Non-uniform rational B-Splines, which are like rubber sheets stretched over the surface. Imagine then, a grid of lines projected over this sheet with control points to allow the surface to be stretched and manipulated by the designer to create complex free-form surfaces. This is how dynamic models can be created, by patching together a series of these NURBS surfaces (Allen, Koupass, 2012). The goal of the CAD parametric modelling is to create a 3D representation, flexible and complex enough to encourage the engineer to easily consider a variety of designs. (Khan, 2017)

Parametric design systems help users manipulate the lenght, angle, and pitch of a particular component. Moreover, the parametric system can produce designs that are more meaningful than those generated by traditional CAD systems. Designers, analysts, and production engineers can work with the same solid model, extracting an adding information according to need (Amirouche, 2004).

In recent years, special software has been developed for parametric modeling (Generative Components, Paracloud). The characteristic of these programs is that each component of the three-dimensional model is created in relation to the other component and its numerical values can be changed at any time. In such programs, rather than standard commands, mathematical formulas of these commands, for example; The mathematical series of copying or duplication operations can be achieved by sequence operations. Parametric design and modeling programs are widely used in engineering, industrial design and transportation industry (Akipek and İnceoğlu, 2007). In recent years, it is seen that parametric modeling studies have been done in our country.

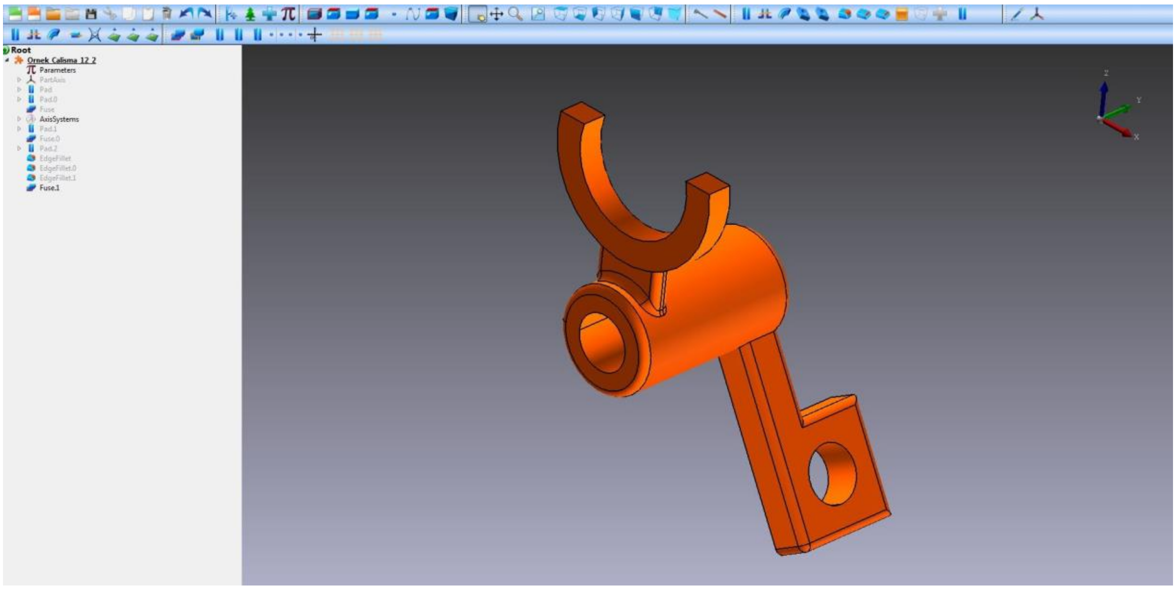

Figure 10. Solid Modelling Example

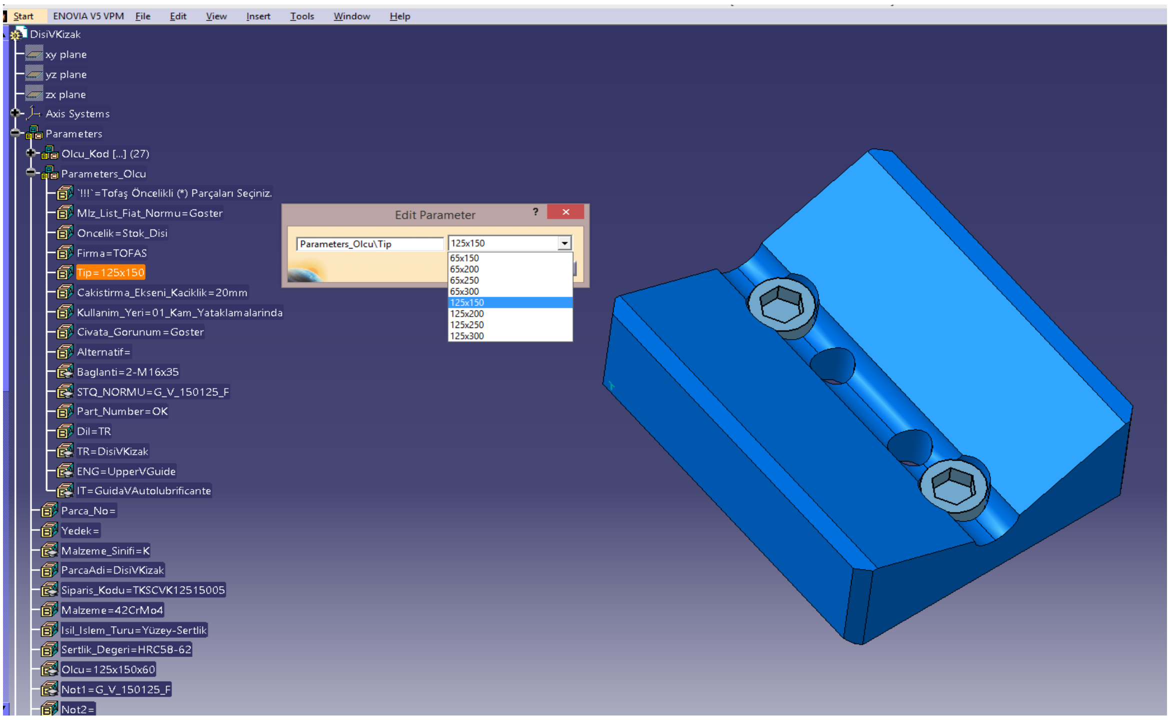

Figure 11. Parametric Design Example

2.5. Use of Computer Aided Design in Sheet Metal Forming Design and Technologies

The device, which produces identical parts within desired size limits and without chip removal in the shortest time, helps to keep the material consumption and manpower to a minimum and which works with machine tools, is called sheet metal sheet metal formings. Parts produced with sheet metal formings have entered every part of our daily life. Parts produced with sheet metal formings are used in kitchen materials, electrical-electronics industry, especially in the automotive industry and other branches of industry that cannot be counted (Ministry of Education, 2011).

Sheet metal forming design and manufacturing is one of the most complex and expensive production stages in the industry. For example, an average of 1200 sheet metal formings is required for the production of an automobile. Between 100 and 600 standard parts are required for each sheet metal forming. A design with such a complex process is quite prone to error, cost and loss of time. The error or prolongation of the sheet metal forming design steps adversely affects the cost and delivery times of a new product.
Improvements in sheet metal forming design processes greatly affect sheet metal forming lead times. Sheet metal forming design times and design quality are the most important factors that affect the cost of sheet metal forming sets to be made. Design times vary depending on the type of sheet metal forming to be made, the size of the part, the symmetry of the part and the sheet material used. The fact that the sheet metal forming design process is as accurate and efficient as possible reduces the cost of parts.

Figure 12 is an example sheet metal forming design. This design was designed for a car body side by Mubitek Company. Design lasted 350 hours and had 580 standart parts.

Straight from the National Union of Turkey (UKUB) in our country with an official letter to the data received 5,000 sheet metal forming manufacturers and has around 100,000 employees. 35% of this number (1,750 companies, 35,000 employees) is the sheet metal sheet metal forminging sector. It is stated by UKUB that this number does not include design offices. In addition to these companies, dozens of designers work in the sheet metal forming design departments in the main and automotive sub-industries. The biggest challenge faced by these designers is to use the information available to predict the future. The accuracy of the estimation is achieved in the degree of accuracy (Bayazit, 1994).

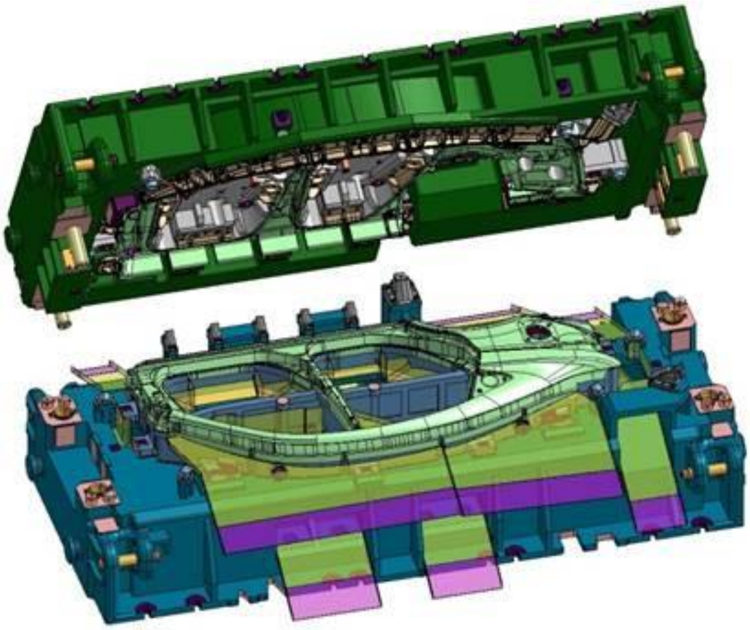

Figure 12. Bodyside Sheet Metal Forming Design

The design of sheet metal sheet metal formings by classical methods consists of long-term calculations and drawings. Selection and drawing of many elements in sheet metal forming design is time consuming because it is the repetition of similar processes (Çetinkaya, 2000).

The introduction of automation to modern technology is increasing day by day and the necessity of this is heard in our country. It is seen that computers are adopted as an auxiliary tool for designers (Bayazit, 1994). Designers have to evaluate everything and place it in the most appropriate place. The structure, the relations with the environment, the connections with the infrastructure, the materials, the technology have to think about them all. Function, the producer and the consumer should address together (Bayazit, 1994). Sheet metal forming industry is a sector that includes advanced technology.

Sheet metal is widely used for industrial and consumer parts because of its capacity for being bent and formed into intricate shapes. Sheet-metal parts comprise a large fraction of automotive, agricultural machinery and aircraft components and of consumer appliances. Successful shee-metal forming depends on the selection of material with adequate formability, the proper design of the part and tooling, the surface condition of the sheet, selection and application of lubricants, and speed of the forming press. The cold stamping of a strip or sheet of metal with dies (sheet metal forming) can be classified as either a cutting or forming operation (Dieter, 1987).

During new vehicle model development, the stamping process and sheet metal forming development for sheet metal parts play an important role in assuring the assembly of body in white within the limited lead time (Chen and et all, 2013).

There are various studies in literature to improve sheet metal forming design and manufacturing process and to reduce costs.

Lee and his friends (1997) gave various assembly relationships to a base sheet metal forming set, and assigned these relationships to parameters. In this way, they were able to update the pattern and adapt it to new situations. They also entered the information of the standard elements in the prepared parameters so that they could quickly create a material list. With this technique they have created sheet metal forming design modules. This technique is a computer aided parametric design.

Myung and Han (2001) assembled the machine elements with the database they prepared. To do this, they used the expert system approach technique.

Duffey and Sun (2003) used computer programming to design progressive patterns. With a user interface prepared by Kumar and Singh (2004), a skeleton was created to help draw programmatic patterns in the AUTOCAD environment.

Chu and his friends (2004) were able to make the necessary element picture ready for design with a user interface before the sheet metal forming design of the automobile tire was prepared.

By Lin and his friends (2006), the rules were assigned to the drawing sheet metal forming set prepared in this study, so that the sheet metal forming could be adapted to new situations. This set, which is prepared in the “Pro / E” solid model design program, takes the information about the press and other sheet metal forming design information from the parameters which are a feature of cad program. This technique is a computer aided parametric design approach.

Kim and his friends (2006) with an interface have been able to prepare to control a small and simple cutting pattern. In this study, Basic Visual Basic 6.0 programming language is used. With this study, the sheet metal forming set was standardized. Pattern information is entered into the Access Ms Access database. The simplicity of the sheet metal forming is the most important factor in the effectiveness of the study. They used parametric design approach in their study.

Skarka (2006) used the “Knowledge” module, a special extension of the CATIA program. This module allows writing rules for the desired transactions. The study is an example of the development of solid model programs. The technical expert is the system approach.

Bintaş and et all (2011) wrote macros and obtained parametric intelligent design libraries by using “Knowledge CA module, a special extension of CATIA program.

As a result of the rapid progress of today's technology, technical infrastructure and features of CAD programs are improving day by day. In order to determine future predictions and to reveal the CAD market in sheet metal design sector, various interviews were made with CAD distributors and a report was prepared. You can see the report in detail in Table 2.

Table 2. CAD Distributors Report

Company	About Company	Licensing Fees How much (per year)	How will CAD applications be in the future?	Usage of Programs in Sheet Metal Forming Design?	Other information
CADBİM	Cadbim Computer Industry Tic. Ltd. Sti. Was established in 1993 in Izmir. The company aims to establish infrastructure for CAD companies and provide necessary training and technical consultancy.It is Autodesk authorized dealer.	Rental system is applied. The licensing fee is between $ 500 and $ 20,000.	All software will be Cloud Based. Virtual Reality and Augmented Reality applications will be used. For example, instant meetings can be held and designs can be controlled regardless of space and time. Artificial Intelligence will be used. For example; working conditions of the part to be designed, production information, material preference and so on. subjects will be selected and the design will be made up to a certain point automatically.	Autodesk has modules for plastic sheet metal forming design. Also it is used in Turkey. But for sheet formwork There are people who use our Inventor program. I don't know the number.	-

(Cont. on next page)

Table 2. (Cont.)

Company	**About Company**	**Licensing Fees How much (per year)**	**How will CAD applications be in the future?**	**Usage of Programs in Sheet Metal Forming Design?**	**Other information**
Erdal Fırat Engineering	Design, Manufacturing and Engineering software sales, training, support, consulting and service provides services. It is the authorized dealer of Surf Designer program of American origin.	Prices for our Surf Designer program range from $ 7,000 to $ 20,000	Drawings drawn by hand in 2 dimensions can be transferred easily in 3 lengths.	We have 100 pieces of sheet metal sheet metal forming design license in Turkey. We have 40 licenses only in Bursa. Plastic sheet metal forming design is not as difficult as sheet.	There is a special program used in Germany for the design of sheet formwork. But I can't remember his name. The design is difficult because sheet sheet metal formings have a complex structure. Most programs do not have the capabilities.
SES 3000	It was established in 1994 and is engaged in the sale and technical support of CNC machine tools and CAD / CAM. Solidworks is an authorized reseller.	Solidworks prices range from $ 3,000 to $ 30,000.	Cloud computing, artificial intelligence will be used.	The Solidworks program is used in sheet metal design. However, it is insufficient on multi-form surfaces. That's why Catia is preferred.	-

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Table 2. (Cont.)

Company	About Company	Licensing Fees How much (per year)	How will CAD applications be in the future?	Usage of Programs in Sheet Metal Forming Design?	Other information
Armada Software	Provides engineering services and is an authorized Solidworks dealer.	Solidworks prices range from $ 3,000 to $ 12,000.	The Solid program is also moving to the cloud. Everything will be on the virtual server. Therefore, there is no possibility of making mistakes. With artificial intelligence, they work for a similar product search feature in design. When the program is loaded, similar designs will be drawn. This feature will help to determine the price of designs and to facilitate the design	In progressive patterns, the capabilities of the solid program are insufficient. That's why it's not used much.	The Solidwork program will soon switch to the annual rental system. However, this situation will decrease the sales of the program. Because our country does not lean towards the rental system.
Tekyaz	Turkey distributor SolidWorks is a company specializing in the sector as a SolidCAM Turkey Distributor and Cimatron CAD Turkey Distributor / CAM / CAE.	Solidworks prices range from $ 3,000 to $ 15,000.	Cloud computing, AR and VR technologies will be used.	There's a Cimatron program in America. A special program for sheet sheet metal forminging. Planning to set the price between $ 4,000-5,000 for next year.	programs made for sheet sheet metal forminging in Turkey will be a very accurate work. Domestic software is very important for the domestic manufacturer.

CHAPTER 3

COMMAND BASED PRODUCTIVITY ANALYSIS BETWEEN ÇİZEN AND CATIA V5

In this chapter, the commands were analyzed in the Çizen program and the Catia V5 program. Catia developed by Dassault Systems Company especially is used automotive sector, aircraft production and other simulation sector. Catia has CAD/CAM/CAE solution packet. But Çizen has just CAD solution. So we just compared with CAD solutions. We analyzed command based productivity between Çizen and Catia V5. The commands in Çizen program were compared with Catia V5. These commands are Edit Toolbar, File Menu, Product Menu, Part Menu, Sketch Menu. We also we examined general interface.

3.1. Comparison of Çizen User General Interface with Catıa V5

In this part Çizen working place and general interface including Start Screen, Worksheet, Save, Sketch, Part and Product was compared with Catia V5. Fistly Çizen and Catia V5 general and start screen interface was compared. As a result of comparison, it was seen that the programs have similar interface. But in the Catia program the caliper element was found to be more useful visually and functionally. There is not a different interface in Çizen for moduls. Solid and surface command are in same interface. In Catia V5 there are diffent interfaces each modul. Another interface comparison is that in Çizen the feedback link is in login screen, in Catia there is in all working place.

In Table 3 also, opening worksheet and save document were compared. Opening worksheet is similar in Çizen and Catia V5. But it is seen that, the variety components created in Catia program is high and interface design is more advenced. Redirects were found useful in both programs to save documents. A comparison was made between the programs for sketch and part. Both programs generally is similar. There is a different in product tree. Catia program has more flexible product tree. So that this is found more useful. In sketh in Çizen, it was seen that commands are limited. Product menu also was compared between. In this, interface design are similar. Just in product tree there are structural differences. In Çizen, the assigned parameters and relationship between the parts are at the top of product tree, but in Catia at the end. Also in Çizen, assembly commands are limited.

As can be seen Table 3, Catia V5 has more commands and capabilities than Çizen. Although the design style and commands are similar, the interface usage and placement and capabilities are different.

3.2. Comparison of Çizen Open and Edit Toolbar Menu With Catia V5

In this part it was given information about the open and edit toolbar in which there are the commands such as assembly design, part design, open, import and export.

As can be seen Table 4, opening new file and/or reopening existing file, editing and exporting can be 50% faster and efficient than the Catia program. Because Catia is a general purpose design program, it contains many different moduls. This results in waste of time and ineffiency. Also, when the comparisons were examined, it was concluded that, assembly, import, export, part and returning part ease of use are similar. But, it was seen that, Catia has much more file types.

Table 3. Summary Table Comparison of Çizen General Interface With Catia

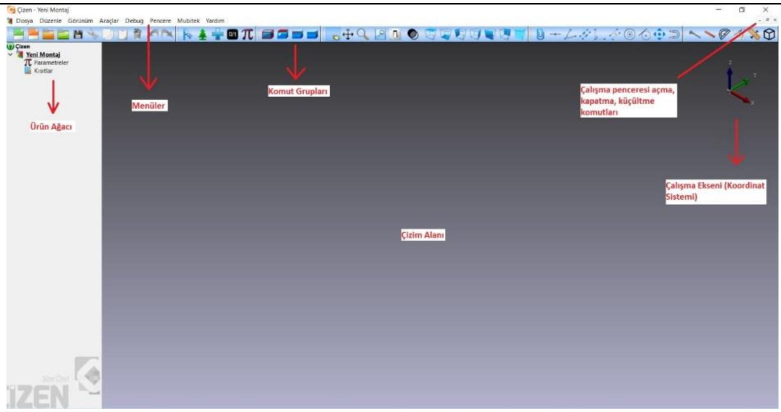
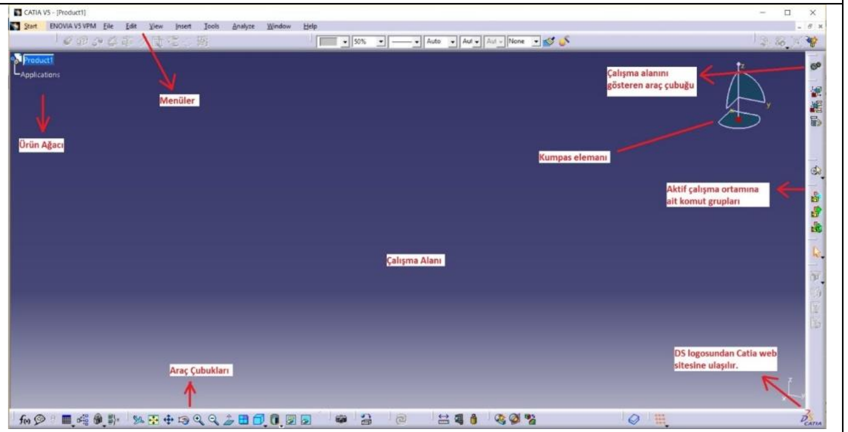
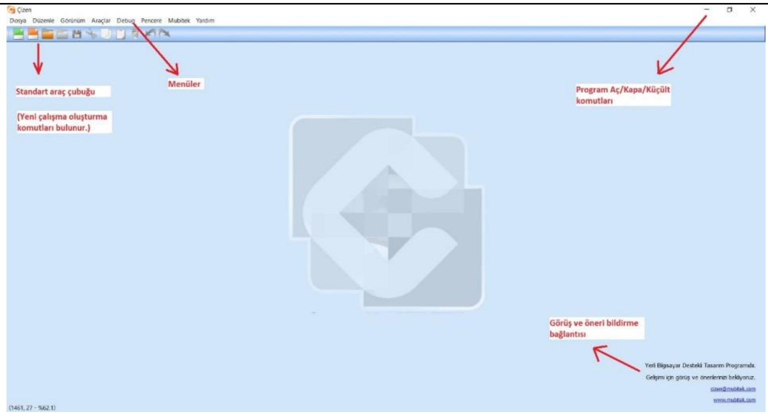
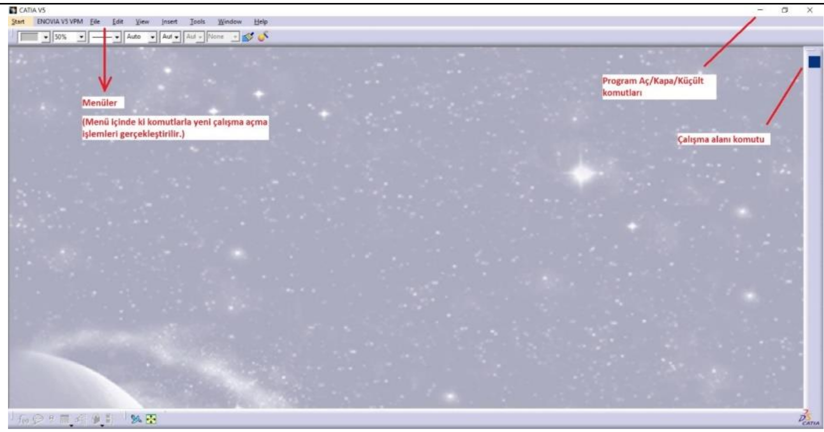
<table>
<tr><th></th><th>ÇİZEN INTERFACE</th><th>CATIA V5 INTERFACE</th></tr>
<tr><td>Working Place and General Interface</td><td>
</td><td>
</td></tr>
<tr><td>Start Screen</td><td>
</td><td>
</td></tr>
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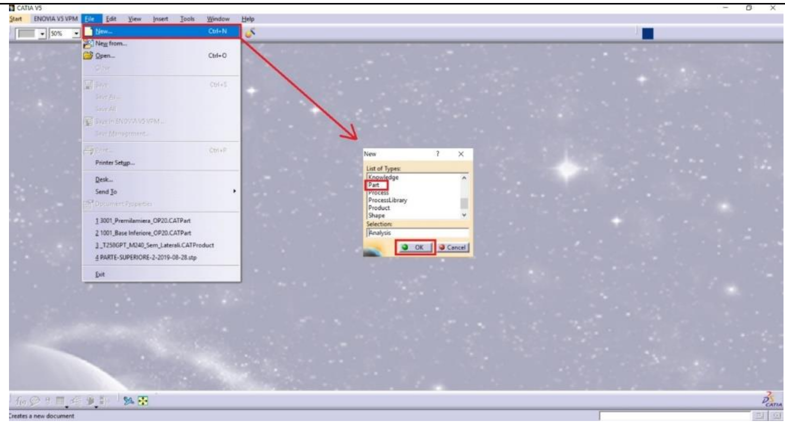
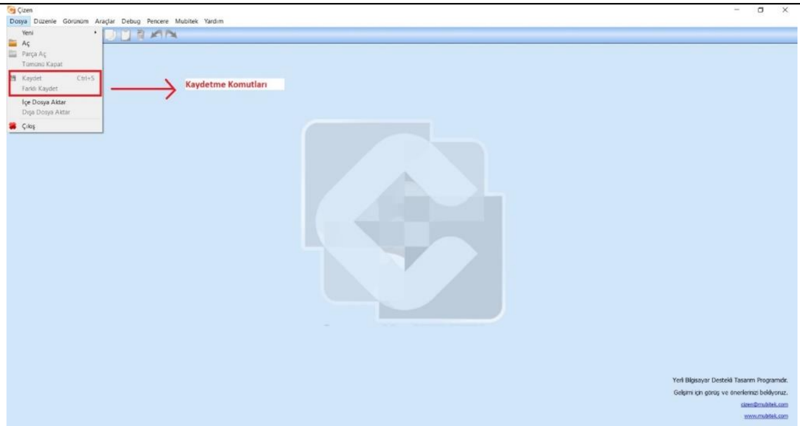
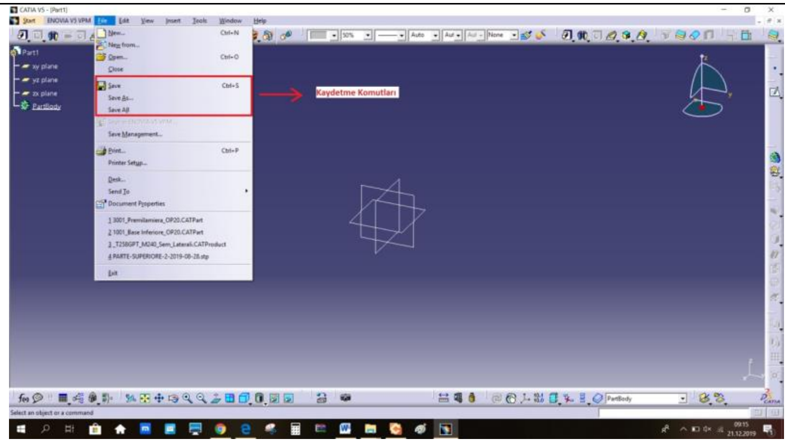
Table 3. (Cont)

	ÇİZEN INTERFACE	CATIA V5 INTERFACE
Worksheet		
Save		

(Cont. on next page)

Table 3. (Cont.)

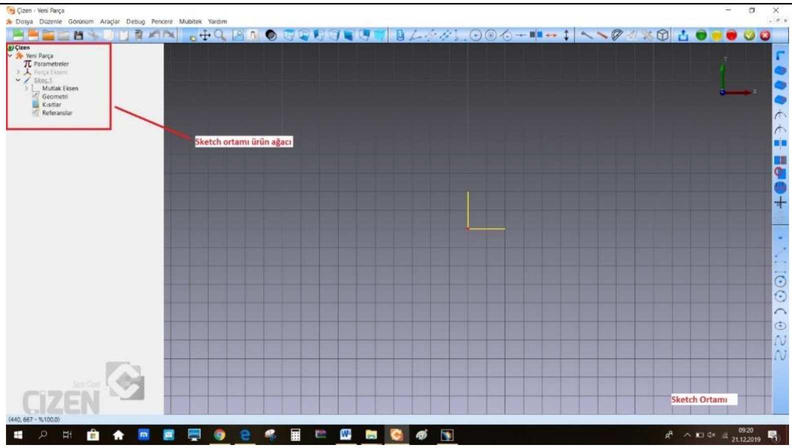
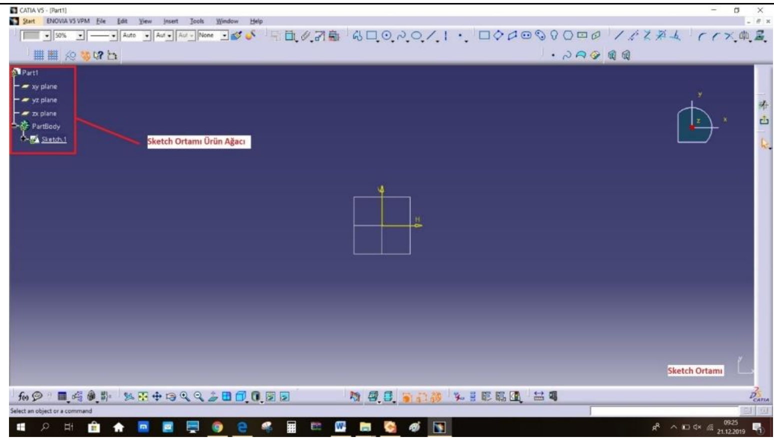
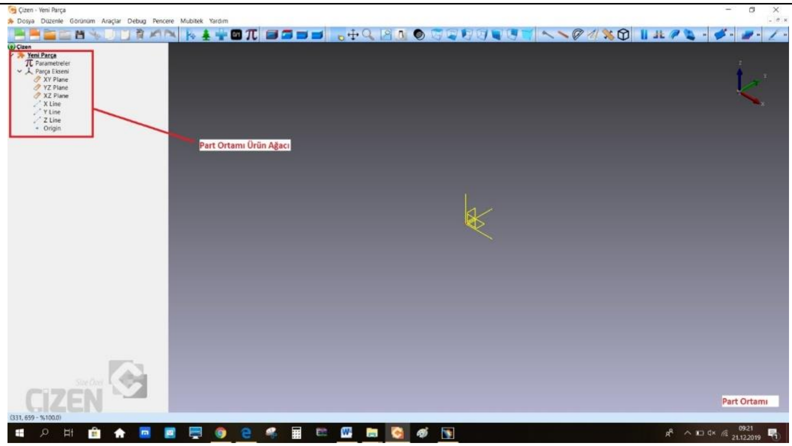
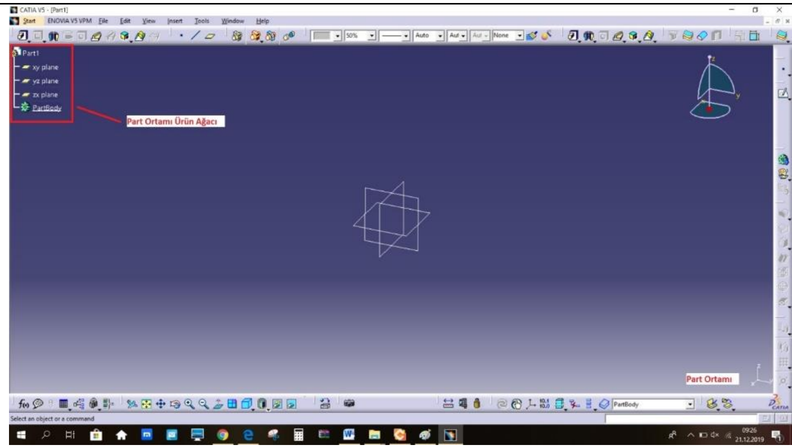
<table>
<tr><th></th><th>ÇİZEN INTERFACE</th><th>CATIA V5 INTERFACE</th></tr>
<tr><td>Sketch</td><td>
</td><td>
</td></tr>
<tr><td>Part</td><td>
</td><td>
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Table 3. (Cont.)

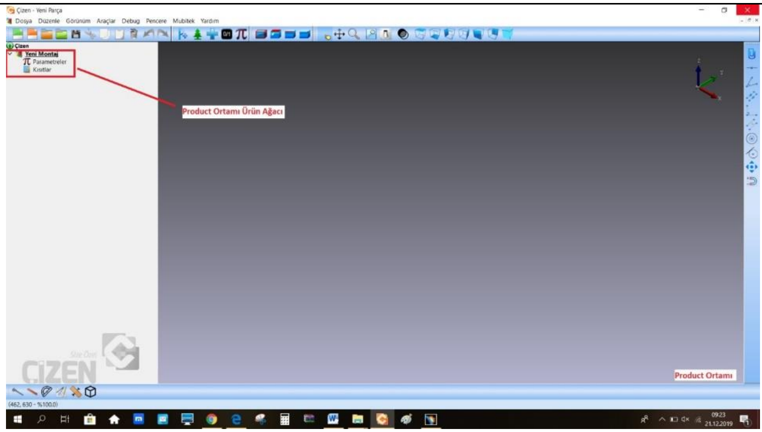
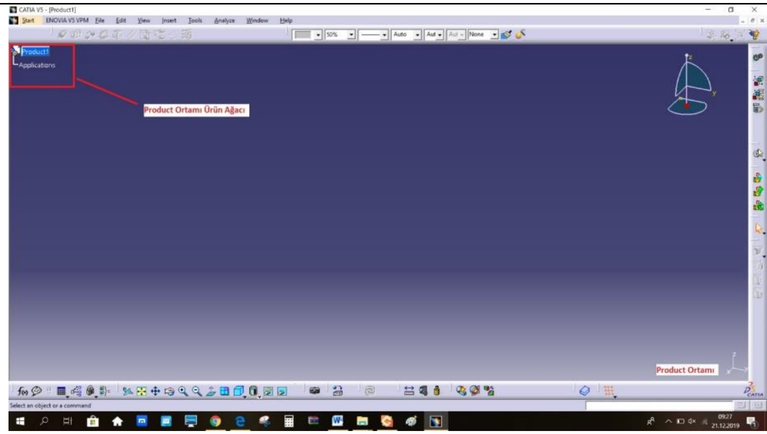
	ÇİZEN INTERFACE	CATIA V5 INTERFACE
Product		

Table 4. Summary Table Comparison of Çizen Open and Edit Toolbar With Catia V5

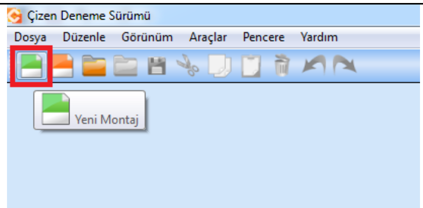
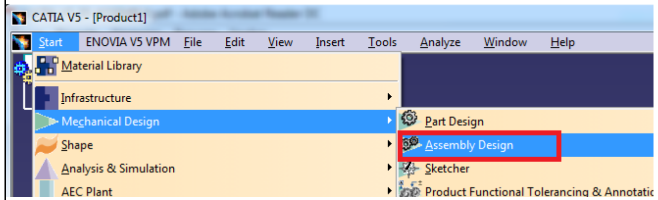
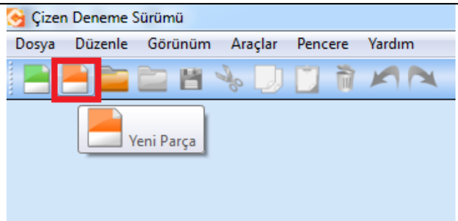
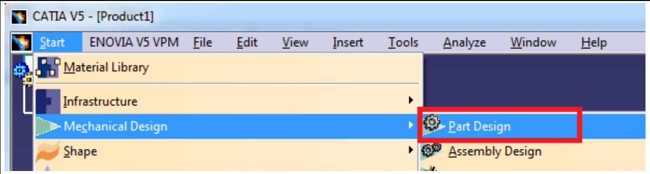
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Assembly Design	 **Yeni Montaj:** It allows us to open a new assembly file. Located at the top left of the Home page.	 **Assembly Design:** It allows us to open a new assembly file. From the Start tab at the top left of the screen, scroll to the Mechanical Design tab, and then click the Assembly Design tab.	-Click Yeni Montaj Icon	-Click Start Icon - Click Assembly Design from Mechanical Design Tab
Part Design	 **Yeni Parça:** It allows us to open a new part file. Located at the top left of the Home page.	 **Part Design:** It allows us to open a new part file. From the Start tab at the top left of the screen, scroll to the Mechanical Design tab, and then click the Part Design tab.	-Click Yeni parça Icon	-Click Start Icon -Click Part Design from Mechanical Design

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Table 4. (Cont.)

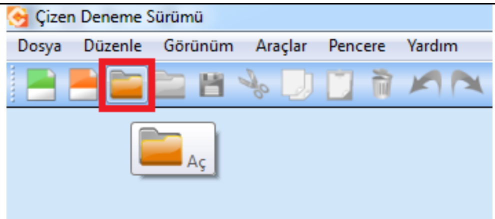
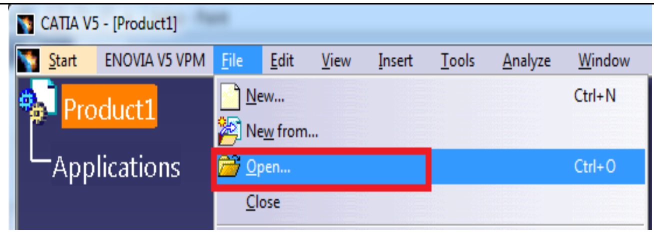
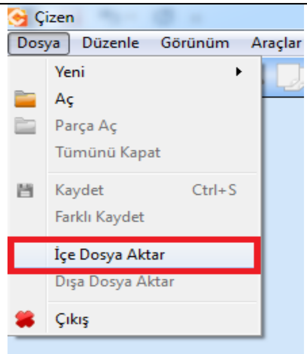
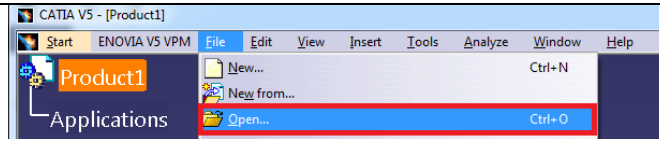
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Open	 **Aç:** It allows us to open an assembly or part file previously created in Çizen.	 **Open:** It allows us to open an assembly or part file previously created in Catia. From the File tab at the top left of the screen, click the Open tab. (Shortcut Ctrl + O).	-Click Aç Icon	-Click File Icon -Click Open
Import	 **İçe Dosya Aktar:** It allows us to open the file created in other CAD programs. From Dosya tab, click İçe Dosya Aktar to select the file you want to open in the Çizen and click Aç.	 **Import:** It allows us to open the file created in other CAD programs. It is not directly in the form of an operation command. In the Product environment, select the file you want to open from the Open tab and open it.	-Click Dosya Icon -Click İçe Dosya Aktar Icon	-Click Start -From the Infrastructure tab, click on the Product Structure tab and switch to the Product environment. -Click File -Click Open

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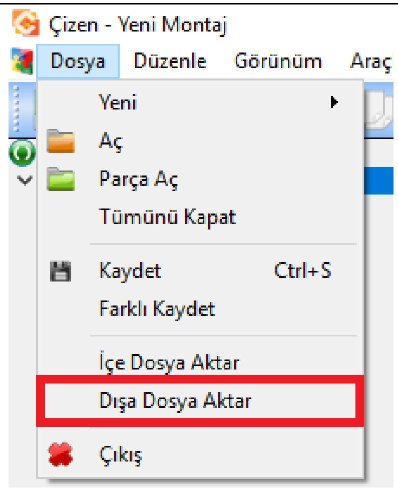
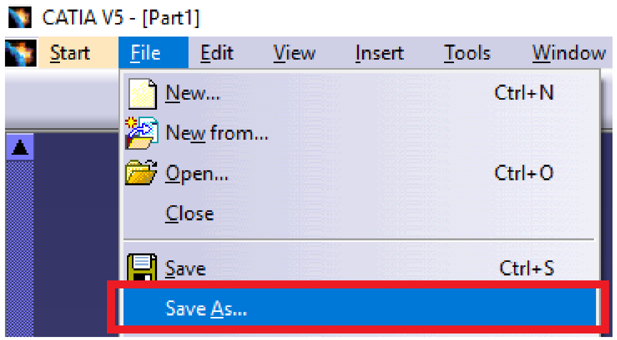
Table 4. (Cont.)

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Export	 **Dışa Dosya Aktar:** Allows us to export files created in Çizen. From the Dosya tab, click Dışa Dosya Aktar to select the file you want to export from the Çizen and click Kaydet.	 **Export:** Allows us to export files created in Catia. It is not directly in the form of an operation command. The data to be transferred from File Save As tab is exported in the desired extension format.	-Click Dosya Icon -Click Dışa Dosya Aktar Icon	-Click File Icon -Click Save As -Select the desired extension. - The export is performed.

3.3. Comparison of Çizen Product Menu With Catia V5

Assembly design can be made with intuitive and flexible user interface thanks to product menu. Assembly designs consist of multiple part or pruduct. An assembly file is created by defining the relationship between these files. The most commonly used modules in Catia is product menu. This menu consist of existing component, save, delete, capture, volume filter, surface filter, curve filter, point filter, pan, zoom in-zoom out, fit all in, hide all axis system, opacity-graphic properties transparan, top view, bottom viex, left view, right view, front view, back view, isometric view distance, fix, coindicence, angle, paralelism, perpendicular, formula, snap, formula string, formula operasyonel, formula boolen, concentricity, tangency, manipulation, measure between, measure item, meausre the thickness, knowledge adviser, measure inertia and bounding box.

In this comparision it was seen that, the design can be made in a few short steps in some operations. But it was also seen that, Catia has more advanced features for some commands such as measure commands.

3.4. Comparison of Çizen Part Menu with Catia V5

Part menu includes command groups in which part design can be as solid model. It can be done 3D model using 2D model which is previosly designed. The designs made in this menu are transferred to other moduls. After that design process is started. This menu consist of pad, shaft, rib, basic hole, gradual hole, edge fillet, angle-edge fillet, edge-edge chamfer, draft angle, spiral, add, remove, translation, rotation, scaling, symmetry, mirror, axis to axis, rectangular pattern, user pattern, extrude, revolve, sweep, split, join and offset.

In this menu, some operations can be made in short steps. But it was seen that, Catia has some user-friendly functions such as hole, edge and champher command. But Çizen does not have these functions.

Table 5. Summary Table Comparison of Çizen Product Menu With Catia

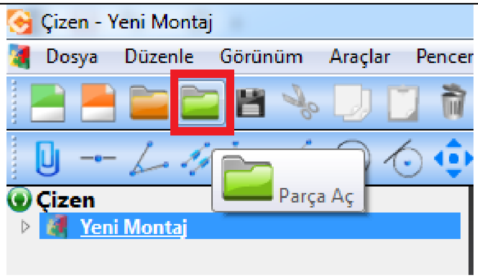
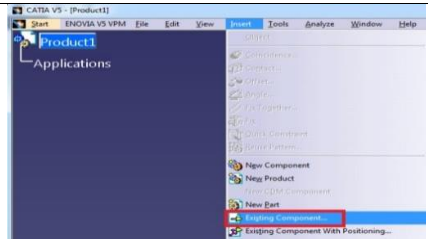
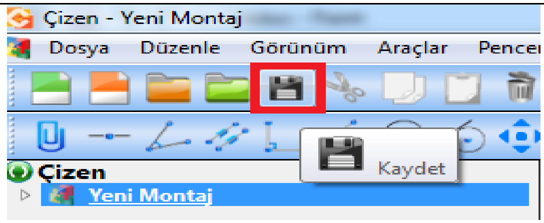
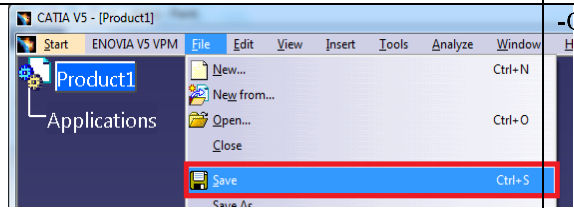
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Existing Component	 **Parça Aç:** It enables us to open your part or assembly which was created in Çizen before. In the assembly environment, click the Parça Aç icon, select the file to be opened, and click Aç.	 **Existing Component:** It enables us to open your part or assembly which was created in Catia before. In the Assembly Design environment, click the Insert and Existing Component tabs, respectively. Select the file you want to open from the incoming screen and click Open.	-Click Parça aç Icon	-Click Insert -Click Existing Component -Choose File and click Open
Save	 **Kaydet:** Saves files. Activates in assembly or part environment.	 **Save:** Saves files. In Part Design and Assembly Design environments, file is saved by clicking File and Save tabs respectively.	-Click Kaydet Icon	-Click File -Click Save

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Table 5. (Cont.) 1

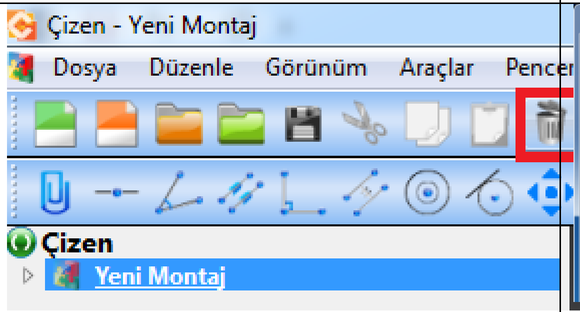
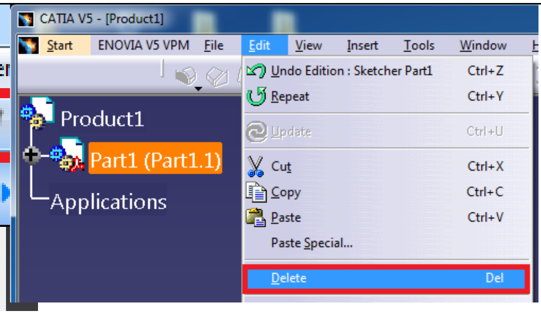
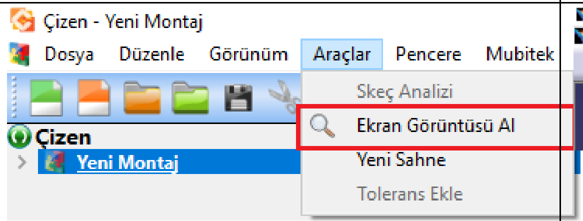
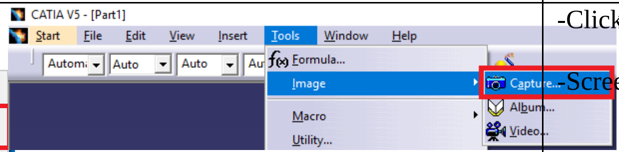
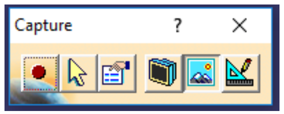
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Delete	 **Sil:** Allows us to delete selected items from the design tree or drawing screen.	 **Delete:** Allows us to delete selected items from the design tree or drawing screen. After selecting the item to be deleted, click Edit and Delete respectively to delete.	-Click Sil	-Click Edit -Click Delete
Capture	 **Ekran Görüntüsü Al:** Allows us to take a screenshot.	 **Capture:** Allows us to take a screenshot.	-Click Araçlar -Screenshot	-Click Tool -Click Image Command -Click the Capture command and take a screenshot from the window that appears.

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Table 5. (Cont.)

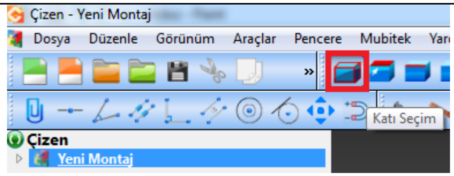
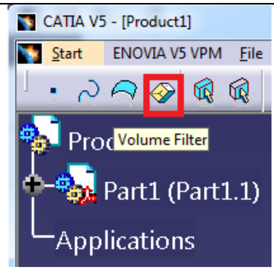
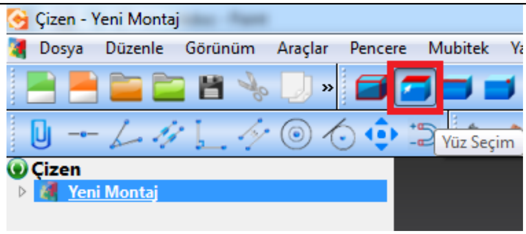
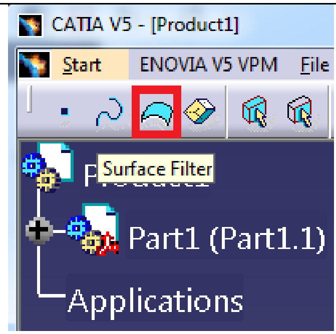
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Volume Filter	 **Katı Seçim:** It is a kind of filtering that facilitates solid selection.	 **Volume Filter:** It is a kind of filtering that facilitates solid selection. Used in Assembly Design or Part Design environments. Click the View, Toolbars, and User Selection Filter tabs.	-Click Katı Seçim	-Click View -Click Toolbars -Click User Selection Filter -Click Volume Filter
Surface Filter	 **Yüzey Seçim:** It is a kind of filtering that facilitates surface selection.	 **Surface Filter:** It is a kind of filtering that facilitates surface selection. Sırasıyla Click the View, Toolbars, and User Selection Filter tabs.	-Click Yüz Seçim	-Click View -Click Toolbars -Click User Selection Filter -Click Surface Filter

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Table 5. (Cont.)

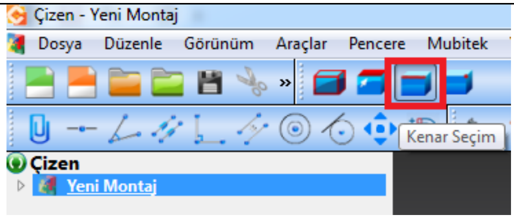
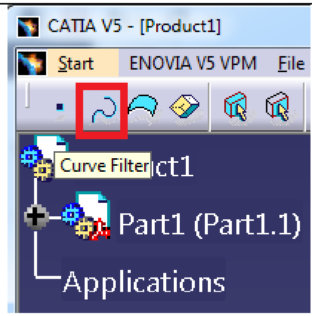
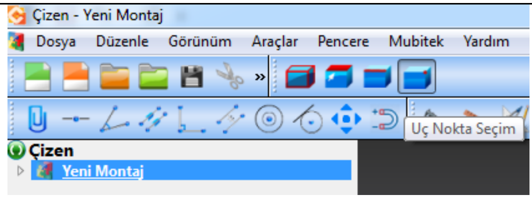
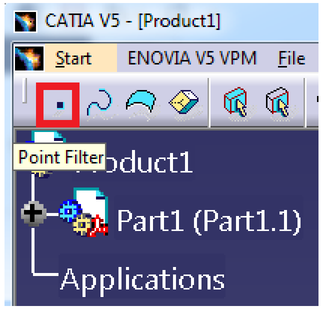
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Curve Filter	 **Kenar Seçim:** It is a kind of filtering that makes curve selection easier.	 **Curve Filter:** It is a kind of filtering that makes curve selection easier. Click the View, Toolbars, and User Selection Filter tabs.	-Click Kenar Seçim	-Click View -Click Toolbars -Click User Selection Filter -Click Curve Filter
Point Filter	 **Uç Nokta Seçim:** It is a kind of filtering that makes point selection easier.	 **Point Filter:** It is a kind of filtering that makes point selection easier. Click the View, Toolbars, and User Selection Filter tabs.	-Click Uç Nokta Seçim	-Click View -Click Toolbars -Click User Selection Filter -Click Point Filter

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Table 5. (Cont.)

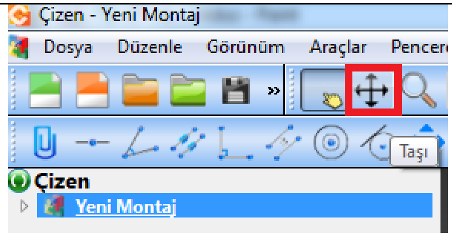
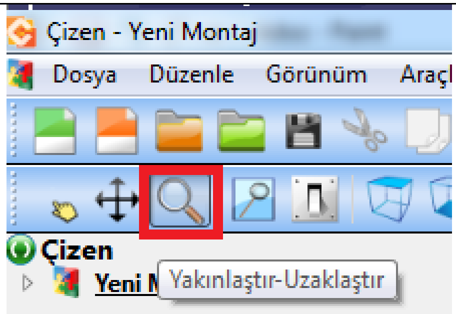
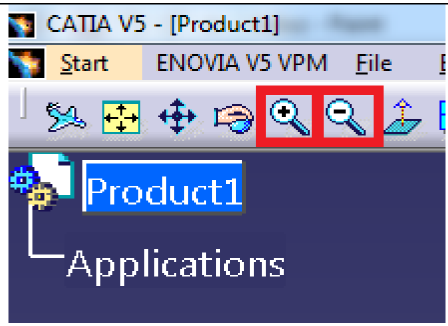
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Pan	 **Taşı:** It allows us to move the drawing. After click the Taşı icon, you can move to the desired location on the screen by holding down the left mouse button.	 **Pan:** It allows us to move the drawing. After click the Pan icon, you can move to the desired location on the screen by holding down the left mouse button.	-Click Taşı -The operation is performed by holding down the left mouse button.	-Click Pan -The operation is performed by holding down the left mouse button.
Zoom In **Zoom Out**	 **Yakınlaştır-Uzaklaştır:** Allows us to zoom in and out. Zooms in and out of the drawing by moving the mouse back and forth while holding down the left button.	 **Zoom In - Zoom Out:** Allows us to zoom in and out. The shortcut for zooming in CATIA is as follows; Ctrl and mouse wheel are held down and the mouse is moved back and forth.	-Click Yakınlaştır – Uzaklaştır - The operation is performed by moving the mouse back and forth.	- Zoom in-out click according to the desired operation and move the image a little. - The same operation is done by holding down Ctrl + mouse wheel and moving the mouse forward.

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Table 5. (Cont.)

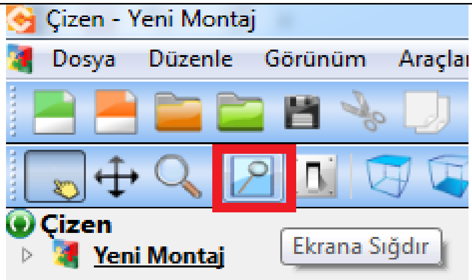
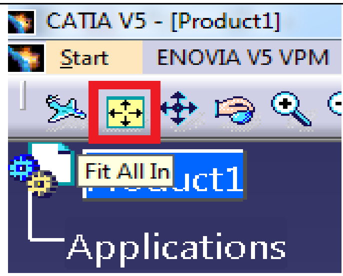
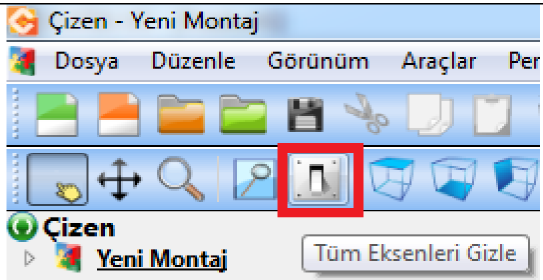
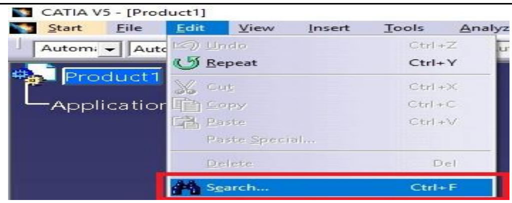
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Fit All In	 **Ekrana Sığdır:** Allows us to fit the drawing to the screen.	 **Fit All In:** Allows us to fit the drawing to the screen.	-Click Ekrana Sığdır	-Click Fit All In
Hide All Axis Systems	 **Tüm Eksenleri Gizle:** It allows us to hide and open all axes.	 **Hide All Axis Systems:** It allows us to hide and open all axes. While doing this, we select Edit-Search-General-Type-From Element and make an axis selection. When we press Search and Select button, all axes are selected. We can hide / hide all axes by right-clicking on any axis on the product tree or click icon.	-Click Tüm eksenler gizle	-Click Edit, Search and General -Select from element from Type tab. -Choose axis -Click Search and Select - All axes are selected. -Hide/Show operation is performed.

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Table 5. (Cont.)

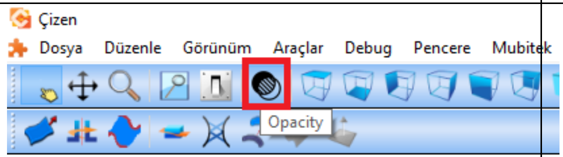
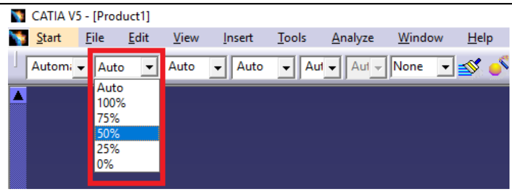
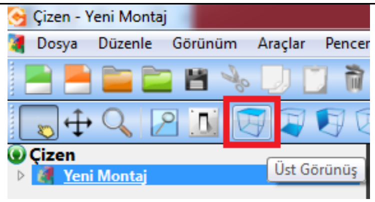
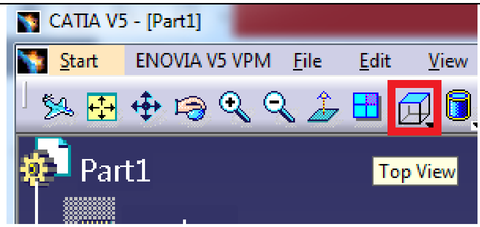
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Opacity-Graphic Properties Transparan	 **Opacity:** It enables us to make solid parts 50% transparent.	 **Graphic Properties Transparan:** It enables us to make solid parts transparent.	-Click Opacity -50% transparent.	- Right-click on the area where the commands on the Catia screen are located and select the grapic properties window - From the Graphic properties tab, click on the 2nd combo Auto combo box - Select the desired transparent value from the Auto combo box field.
Top View	 **Üst Görünüş:** It allows us to see the top view of the drawing.	 **Top View:** It allows us to see the top view of the drawing.	-Click Üst Görünüş	- Click Top View

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Table 5. (Cont.)

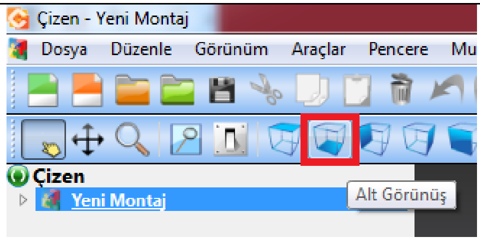
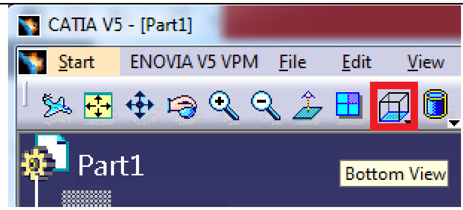
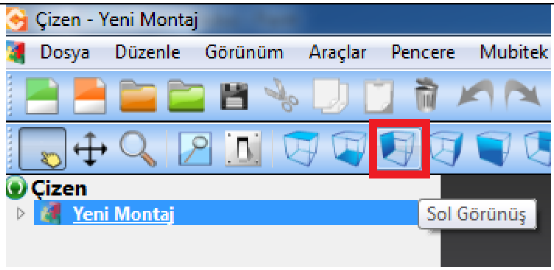
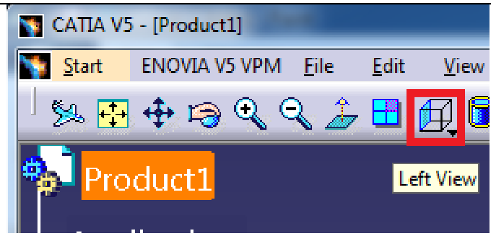
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Bottom View	 **Alt Görünüş:** It allows us to see the bottom view of the drawing.	 **Bottom View:** It allows us to see the bottom view of the drawing.	- Click Alt Görünüş	- Click Bottom View
Left View	 **Sol Görünüş:** Lets you see the left view of the drawing.	 **Left View:** Lets you see the left view of the drawing.	- Click Sol Görünüş	- Click Left View

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Table 5. (Cont.)

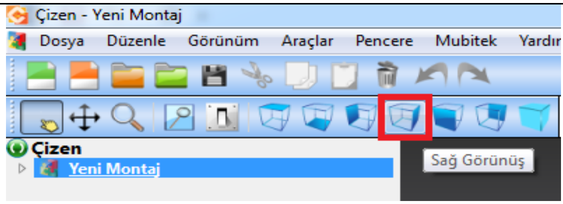
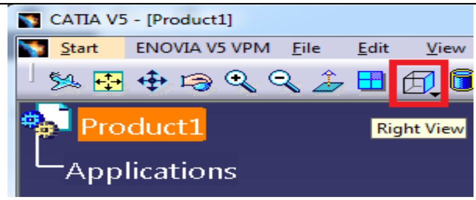
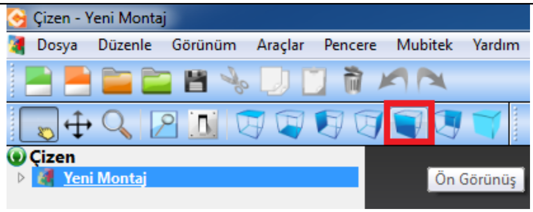
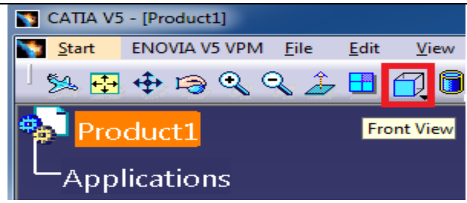
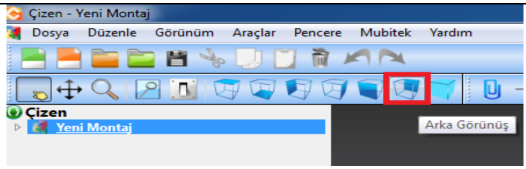
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Right View	 **Sağ Görünüş:** Lets you see the rigth view of the drawing.	 **Right View:** Lets you see the right view of the drawing.	- Click Sağ Görünüş	- Click Right View
Front View	 **Ön Görünüş:** Lets you see the front view of the drawing.	 **Front View:** Lets you see the front view of the drawing.	- Click Ön Görünüş	- Click Front View
Back View	 **Arka Görünüş:** Lets you see the back view of the drawing.	 **Back View:** Lets you see the back view of the drawing.	- Click Arka Görünüş	- Click Back View

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Table 5. (Cont.)

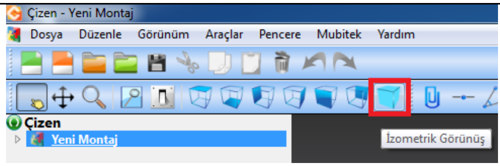
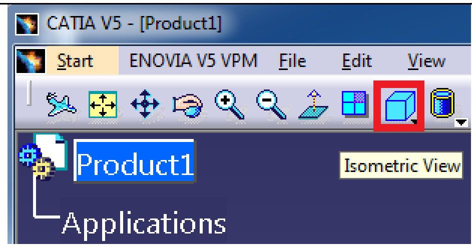
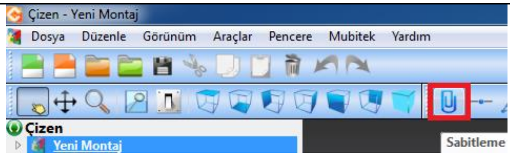
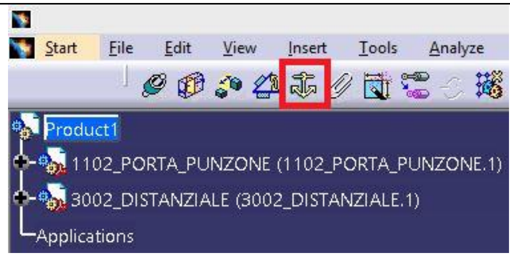
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Isometric View	 **İzometrik Görünüş:** Lets you see the isometric view of the drawing	 **Isometric View:** Lets you see the isometric view of the drawing	- Click İzometrik Görünüş	- Click Isometric View
Fix	 **Sabitleme:** this command fixed the size and position of the selected drawing elements.	 **Fix:** this command is a component of a parent component geometric for fixing to the center or to the geometric center of assembly used.	- Select the component to fix. -Click on the operation icon.	-Select the component or components that you want to fix. -Click on the operation icon. or -Select the component or components that you want to fix. -Click on the operation icon.

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Table 5. (Cont.)

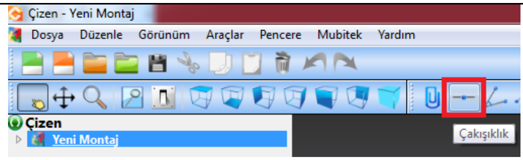
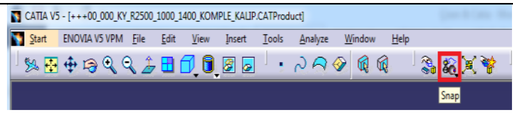
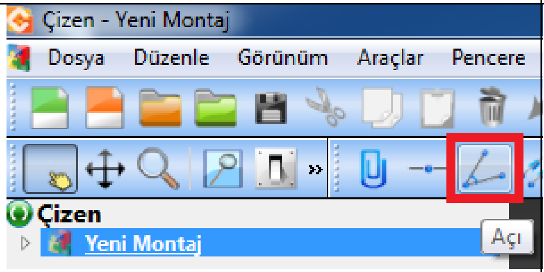
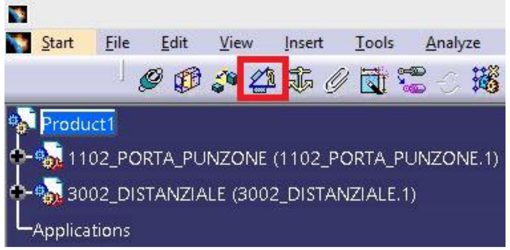
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Coincidence	 **Çakışıklık:** This command makes the selected two elements overlap.	 -This command is not available in Catia. The Snap command in Catia performs the function of the Coincidence command in Çizen. But the Snap command is not assigned a relationship.	-Click on the operation icon. -Select the first component. -Select the second component.	-This command is not available in Catia. -Click on the operation icon. -Select surface of first somponent. -Select surface of second component.
Angle	 **Açı:** It enables us to form an angle between two selected elements.	 **Angle:** It enables us to form an angle between two selected elements.	-Click on the operation icon. -Select the first component. -Select the second component.	- Select the first component you want to angle. - Select the second component you want to angle. -Click on the operation icon. (You can change the order of these commands.) -In the window that opens, enter the angle value.-Click Update All

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Table 5. (Cont.)

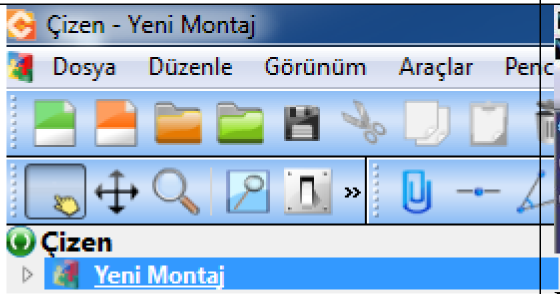
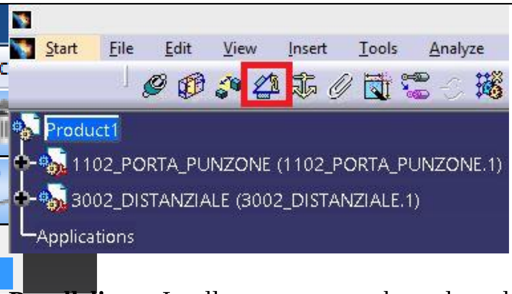
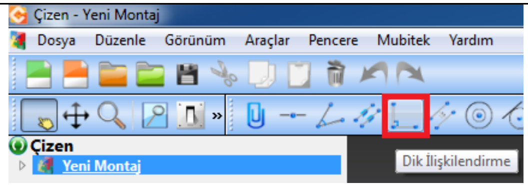
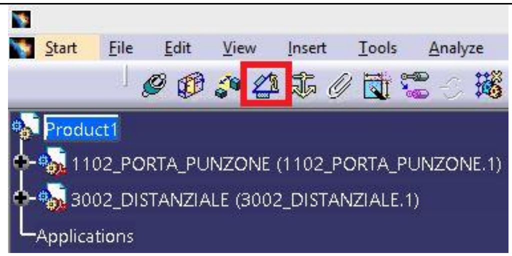
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Parallelism	 **Paralel İlişkilendirme:** It allows us to make selected surfaces parallel to each other.	 **Parallelism:** It allows us to make selected surfaces parallel to each other.	-Click on the operation icon. -Select the first component. -Select the second component.	This command has no equivalent in Catia. - When the angle is written as zero degree in the angle command, the components are parallel. -Click Update All
Perpendicular	 **Dik İlişkilendirme:** It allows us to make selected surfaces perpendicular to each other.	 **Perpendicular:** It allows us to make selected surfaces perpendicular to each other. Click Constraints Defined in Dialog Box,check Perpendicular and click OK.	-Click on the operation icon. -Select the first component. -Select the second component.	This command is not available in Catia. - When the angle is written as ninety degrees in the angle command, the components are parpendicular. -Click Update All

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Table 5. (Cont.)

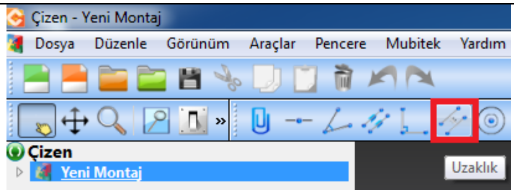
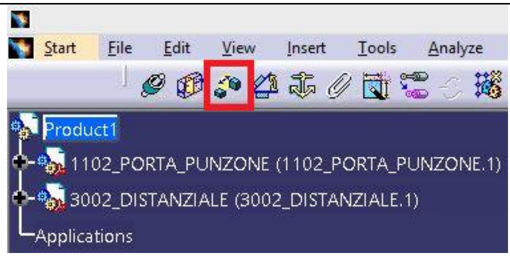
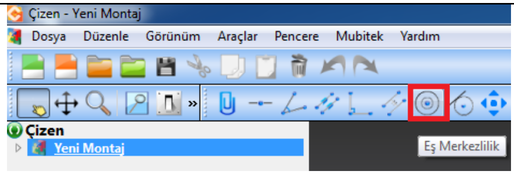
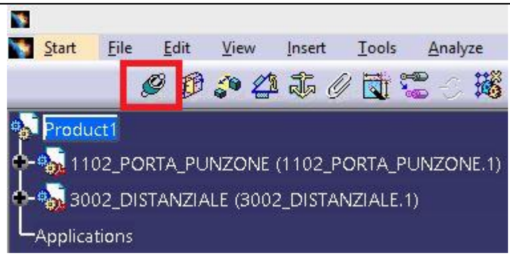
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Distance	 **Uzaklık:** This command creates a distance measure between two selected elements.	 **Distance:**. This command creates a distance measure between two selected elements.	-Click on the operation icon. -Select the first component. -Select the second component.	- Select the first component. - Select the second component. -Click on the operation icon. (You can change the order of these commands.) -In the window that opens, enter the distance value. -Click Update All
Concentricity	 **Eş Merkezlilik:** It allows us to concentrate two objects with circular cross-section.	 **Concentricity:** It allows us to concentrate two objects with circular cross-section.	-Click on the operation icon. -Select the first component. -Select the second component.	- Select the first component. - Select the second component. -Click on the operation icon. (You can change the order of these commands.) -Click Update All

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Table 5. (Cont.)

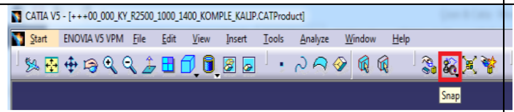
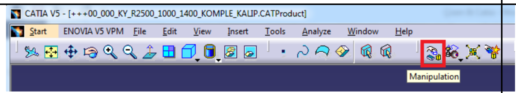
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Tangency	 **Teğetlik:** It allows us to tangent two objects, one of which is circular and the other is straight or both circular.	 This command is not available in Catia. The Snap command in Catia performs the function of the Tangency command in Çizen. But the Snap command is not assigned a relationship.	-Click on the operation icon. -Select the first component. -Select the second component.	This command is not available in Catia.
Manipulation	 **Hareket Ettir:** It allows us to move and rotate part files freely.	 **Manipulation:** It allows us to move and rotate part files freely.	-Click on the operation icon. -The move method is selected from the Dialog Box. -If a special direction is defined, the direction is selected first. - The piece is selected and moved freely with the mouse.	-Click on the operation icon. -The move method is selected from the Dialog Box. -If a special direction is defined, the direction is selected first. - The piece is selected and moved freely with the mouse.

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Table 5. (Cont.)

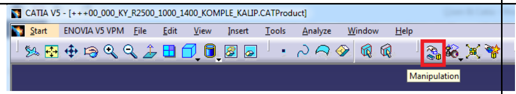
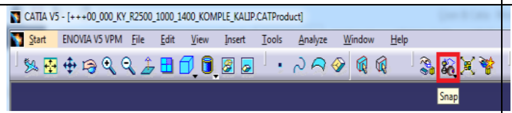
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Manipulation	 **Hareket Ettir:** It allows us to move and rotate part files freely.	 **Manipulation:** It allows us to move and rotate part files freely.	-Click on the operation icon. -The move method is selected from the Dialog Box. -If a special direction is defined, the direction is selected first. - The piece is selected and moved freely with the mouse.	-Click on the operation icon. -The move method is selected from the Dialog Box. -If a special direction is defined, the direction is selected first. - The piece is selected and moved freely with the mouse.
Snap	 **Yakala:** It allows us to freely overlap the selected surfaces on the drawing.	 **Snap**: It allows us to freely overlap the selected surfaces on the drawing.	-Click on the operation icon. -Select surface of first somponent. -Select surface of second component.	-Click on the operation icon. -Select surface of first somponent. -Select surface of second component. (You can change the order of these commands.)

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Table 5. (Cont.)

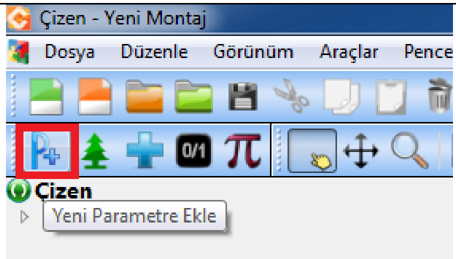
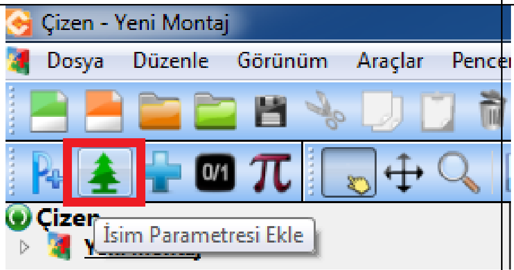
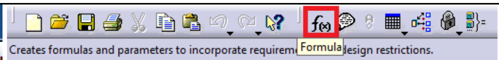
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Formula	 **Yeni Parametre Ekle:** Adds a new parameter.	 **Formula:** Adds a new parameter.	-Click Komuta - The parameters create in the product tree as NewStrParam1.	-Komuta tıklanır. - Select the optional parameter from the new parameter type of field from the parameter screen.
Formula String	 **İsim Parametresi Ekle:** Adds a new name parameter.	 **Formula String:** Adds a new name parameter.	-Click Komuta - The parameters create in the product tree as NewStrParam1.	-Komuta tıklanır. - Select the optional parameter from the new parameter type of field from the parameter screen.

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Table 5. (Cont.)

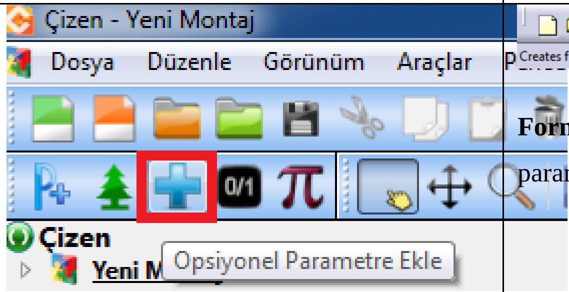
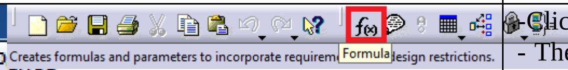
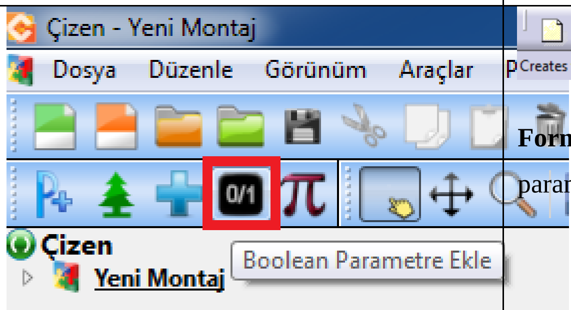
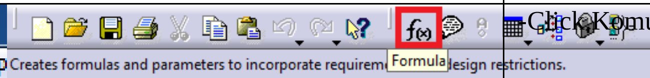
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Formula Operasyonel	 **Opsiyonel Parametre Ekle:** Adds a new optional parameter.	 **Formula Opsiyonel:** Adds a new optional parameter.	-Click Komuta - The parameters create in the product tree as NewStrParam1.	-Komuta tıklanır. - Select the optional parameter from the new parameter type of field from the parameter screen.
Formula Boolean	 **Boolean Parametre Ekle:** Adds a new boolean parameter.	 **Formula Boolean:** Adds a new boolean parameter.	-Click Komuta - The parameters create in the product tree as NewBoolParam1.	-Komuta tıklanır. - Select the boolean parameter from the new parameter type of field from the parameter screen.

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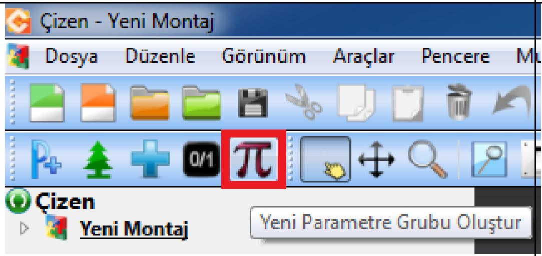
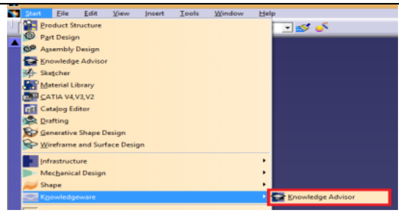
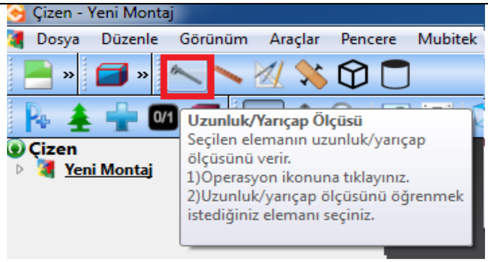
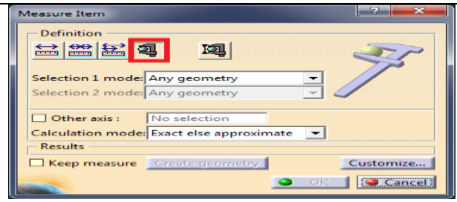
Table 5. (Cont.)

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Knowledge Advisor	 **Yeni Parametre Grubu Oluştur:** Allows us to create a new parameter group.	 **Knowledge Advisor:** Allows us to create a new parameter group. Click Knowledge advisor and then click add set of parameters icon.	-Click Komuta - The parameters create in the product tree as ParamContainer1.	-Click Start -Click Knowledgeware - Click Knowledge advisor. Then click the add set of parameters icon. - A new parameter group is created in the product tree.
Measure Item	 **Uzunluk/Yarıçap Ölçüsü:** Allows us to measure the length of a selected edge. If the selected edge is a circular edge, it also allows us to measure the radius value.	 **Measure Item** Allows us to measure the length of a selected edge. If the selected edge is a circular edge, it also allows us to measure the radius value. After clicking the Measure Item icon, select the element whose length measurement is to be learned.	-Click Uzunluk/Yarıçap Ölçüsü Icon - Select the element whose length / radius is to be measured.	-click Measure Item Icon - Select the element whose length / radius is to be measured.

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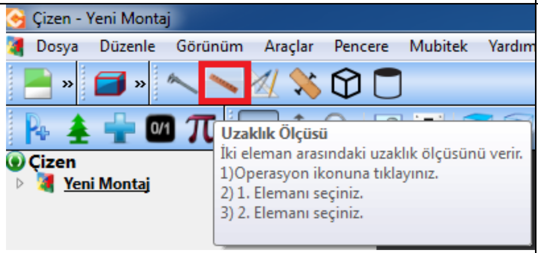
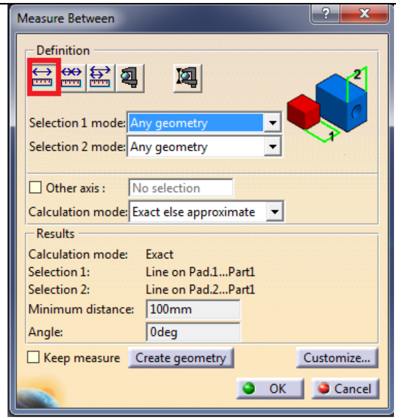
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Measure Between	 **Uzaklık Ölçüsü:** It allows us to measure the distance between two elements.	 **Measure Between:** It allows us to measure the distance between two elements. After clicking the Measure Between icon, the first and second elements are selected and the distance measurement is obtained.	-Click Uzaklık Ölçüsü Icon -Choose 1.Element - Choose 2.Element	-Click Measure Between -Choose 1.Element - Choose 2.Element

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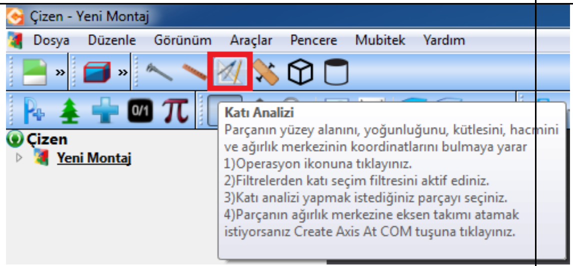
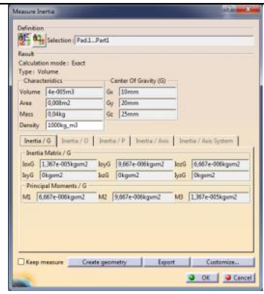
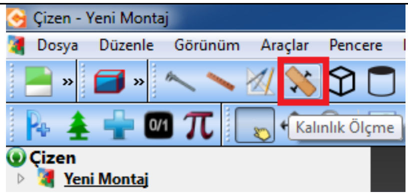
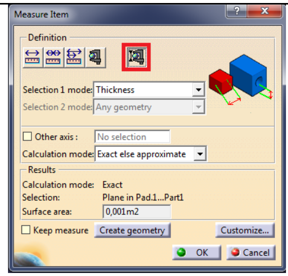
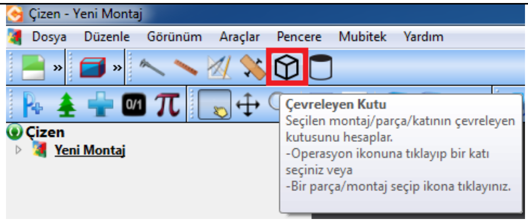
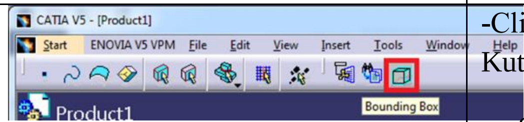
Table 5. (Cont.)

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Measure Inertia	 **Katı Analizi:** It allows us to see the center of gravity, density, mass and volume of solids and to place an axis on the center of gravity.	 **Measure Inertia::** It allows us to see the center of gravity, density, mass and volume of solids and etc.	-Click Katı Analizi - Select the part you want to perform solid analysis. - If you want to assign an axis tool to the center of gravity of the part, click Create Axis At COM.	-Click Measure Inertia Icon - Select the part you want to perform solid analysis. If you want to assign an axis tool to the center of gravity of the part, click Create geometry. -Click Axis system
Measure The Thickness	 **Kalınlık Ölçme:** It allows us to measure the thickness of solid objects.	 **Measures the thickness:** It allows us to measure the thickness of solid objects.	-Click Kalınlık Ölçme -Choose the drawing	-Click the Measure the thickness icon from the Measure Item menu - Choose the drawing

(Cont. on next page)

Table 5. (Cont.)

<table>
<tr><th>COMMAND</th><th>ÇİZEN ICON AND EXPLANATION</th><th>CATIA ICON AND EXPLANATION</th><th>ÇİZEN COMMAND USAGE</th><th>CATIA COMMAND USAGE</th></tr>
<tr><td>Bounding Box</td><td>

Çevreleyen Kutu: It allows us to see the dimensions of the box (billet) required to produce a solid object.</td><td>

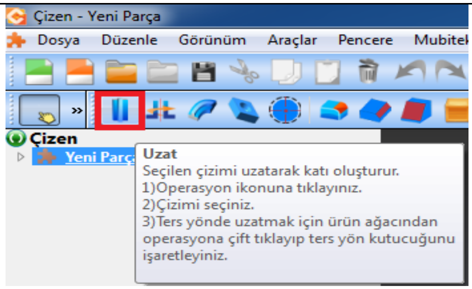
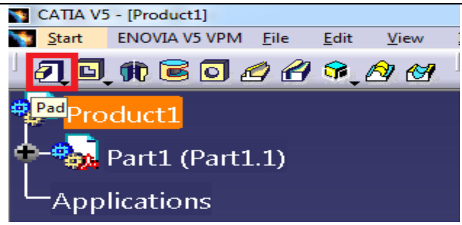
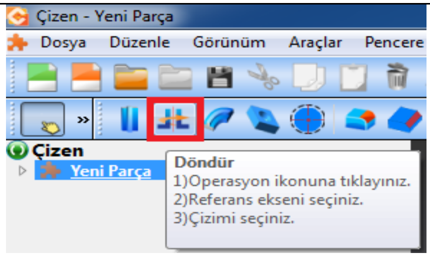
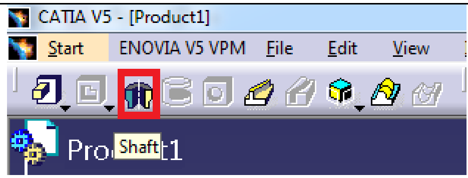
Bounding Box: It allows us to see the dimensions of the box (billet) required to produce a solid object. Click Start and then select Core & Cavity Design from Mechanical Design Tab.</td><td>-Click Çevreleyen Kutu

- Select a part or assembly.</td><td>- Click on Start, Mechanical Design and Core & Cavity Design.

-Click Bounding Box

- Select a part or assembly.</td></tr>
</table>

Table 6. Summary Table Comparison of Çizen Part Menu with Catia

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Pad	 **Uzat:** It allows us to create solid by extending.	 **Pad:** It allows us to create solid by extending. Select the drawing for which you want to create a solid and then click the Pad command. Enter the length of the desired solid to be created from the Pad Definition screen that appears on the screen and click OK. Click Reverse Direction to create a solid in the opposite direction, and then click OK.	- Select the drawing -Click Uzat Icon - Enter the thickness from the menu and click the figure again.	- Select the drawing -Click Pad Icon -Enter the thickness from the menu and click Ok.
Shaft	 **Döndür:** Allows us to form solids by rotating.	 **Shaft:** Allows us to form solids by rotating. After selecting the object to rotate, click on the Shaft icon. Select the reference axis, write the amount of rotation in degrees and click OK	- Select the drawing and Click Döndür Icon - Enter the amount of rotation from the menu and select the reference axis.	- Select the drawing -Click Shaft - Enter the amount of rotation from the menu and click Ok

(Cont. on next page)

Table 6. (Cont.) 1

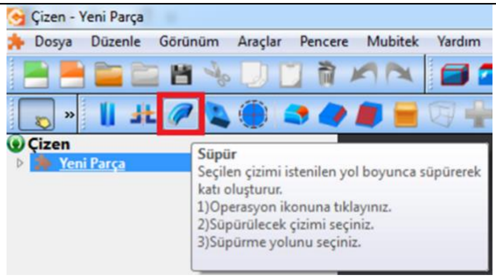
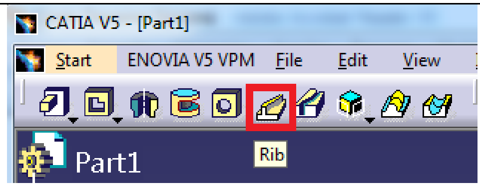
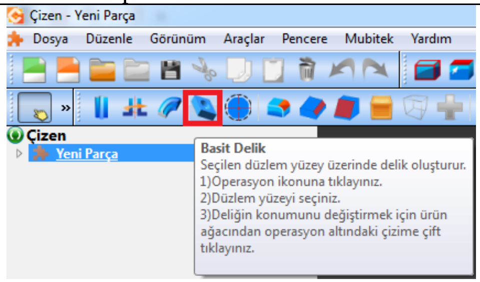
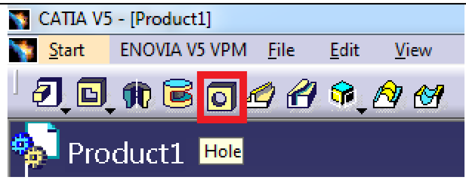
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Rib	 **Süpür:** Sweep the selected drawing along the desired path to create solids.	 **Rib:** Sweep the selected drawing along the desired path to create solids.	-Click Süpür Icon -Select the drawing -Select path.	-Click Rib Icon - Select the drawing - Select path
Basic Hole	 **Basit Delik:** It allows us to create holes on the surface.	 **Hole:** The selected plane allows us to form a stepped hole on the surface. Select the surface to be drilled and click the Hole icon. Ekrana gelen After determining the position of the hole from the Positioning Sketch in the Hole Definition menu, select the type of the hole as Simple from the Type tab in the menu. Enter the depth and diameter values from the Extension tab and click OK.	-Click Basit Delik Icon -Select the plane - Enter depth and diameter values from the corresponding menü - To change the position of the hole, double-click the drawing under the process from the product tree.	-Click Hole Icon - Select the plane - The position of the hole is determined from Positioning Sketch in the menu that appears - Select the hole type Simple from the Type tab in the menu. - Enter the depth and diameter from the Extension tab and click OK.

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Table 6. (Cont.)

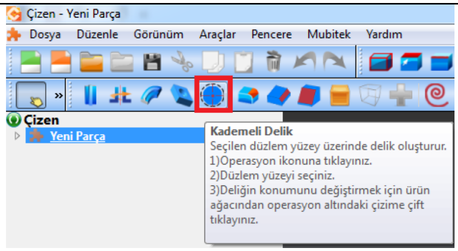
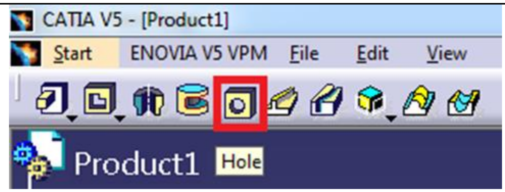
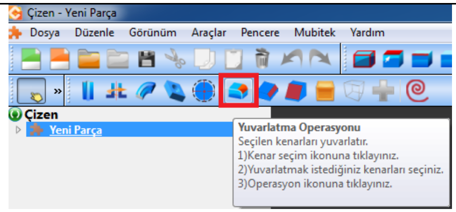
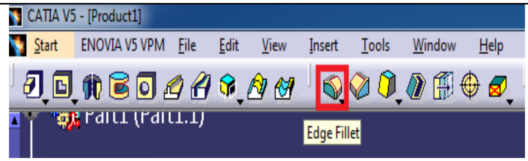
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Gradual Hole	 **Kademeli Delik:** The selected plane allows us to form a stepped hole on the surface.	 **Hole:** The selected plane allows us to form a stepped hole on the surface. Select the surface to be drilled and click the Hole icon. Ekrana gelen After determining the position of the hole from the Positioning Sketch in the Hole Definition menu, select the type of hole as Tapered from the Type tab in the menu. Gerekli After entering the other parameters, enter the depth and diameter values from the Extension tab and click OK.	-Click Kademeli Delik Icon - Select the plane - Enter the depth and diameter values from the corresponding menu - To change the position of the hole, double-click the drawing under the process from the product tree.	-Click Hole icon and Select the plane - The position of the hole is determined from Positioning Sketch in the menu that appears. - The type of hole is selected as Tapered from the Type tab in the menu. - Enter the depth and diameter from the Extension tab and click OK.
Edge Fillet	 **Yuvarlatma:** It allows us to round the edges. After selecting the edges to be rounded, click the Yuvarlama icon. Click OK after entering the required Radius value.	 **Edge Fillet:** It allows us to round the edges. After selecting the edges to be rounded, click the Edge Fillet icon. Ekrana gelen Enter Radius value in Edge Fillet Definition menu and click OK.	- Select the edges to round -Click Yuvarlatma Icon and enter the Radius value.	- Select the edges to round - Enter the Radius value from the Edge Fillet Definition menu that appears.

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Table 6. (Cont.)

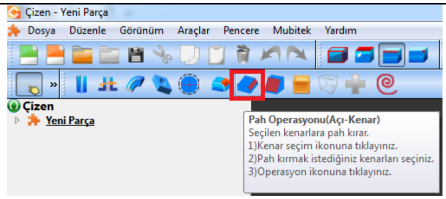
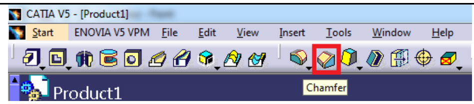
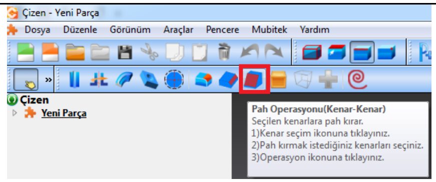
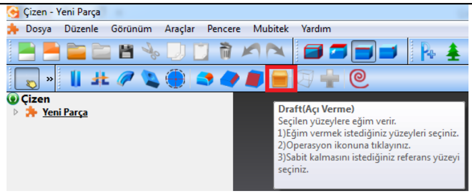
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Angle-Edge Fillet	 **Pah (Açı-Kenar):** Allows us chamfer to the selected edges.	 **Chamfer:** Allows us chamfer to the selected edges. Select the edges to be chamfered and select Length1 / Angle from the Mode section of the Chamfer Definition menu that appears. Enter the required values (Length1-Edge, Angle-Angle) and click OK.	- Select the edges to round - Click Yuvarlatma Icon and enter the Radius value.	- Select the edges to round - Enter the Radius value from the Edge Fillet Definition menu that appears.
Edge-Edge Chamfer	 **Pah (Kenar -Kenar):** Allows us chamfer to the selected edges.	 **Chamfer:** Allows us chamfer to the selected edges. Select the edges to be chamfered and select Length1/Length2 from the Mode section of the Chamfer Definition menu that appears. Enter the required values (Length1-Edge, Angle-Angle) and click OK.	- Select the edges to chamfer -Click Pah Operasyonu (Kenar-Kenar) Icon - Enter length values.	- Select the edges to chamfer -Click Chamfer Icon - Select Length1 / Length2 from Mode of the incoming menü - Enter length values.
Draft Angle	 **Draft (Açı Verme):** Gives angle to selected edges.	 **Draft Angle:** Gives angle to selected edges. Select the surface you want to give angle and click the Draft Angle icon.	-Select the planes -Click Draft(Açı Verme) Icon - Select the reference surface to remain constant.	-Click Draft Angle - Select the planes - Select the reference surface to remain constant.

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Table 6. (Cont.)

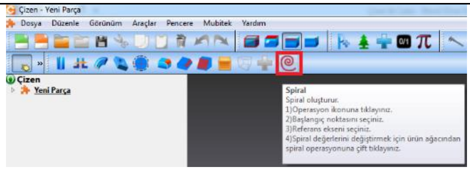
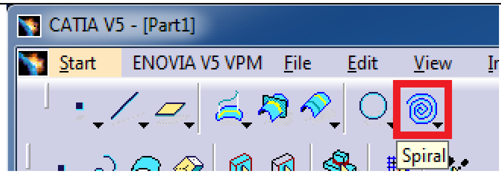
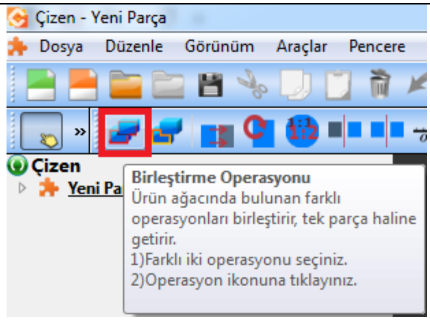
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Spiral	 **Spiral:** It allows us to form spirals.	 **Spiral:** It allows us to form spirals. Available in Generative Shape Design mode. Click the Start tab and the Generative Shape Design can be opened from the Shape tab.	-Click Spiral Icon - Select the starting point - Select the reference axis - If you want to change the spiral values, double-click the spiral process from the product tree.	- Click the Start, Shape, and Generative Shape Design tabs, respectively -Click Spiral Icon - Specify Support, Center Point, and Reference Direction, respectively - Enter the required angle, radius and other values and click OK.
Add	 **Birleştirme Operasyonu:** It allows us to combine different operations in the product tree and make it into one piece.	 **Add:** It allows us to combine different operations in the product tree and make it into one piece.	-Click Yer Değiştirme Icon - Select direction - Enter the amount of displacement - Select the drawing to change location.	- Activate the part to be joined (define in work object). - Select the part to be joined from the product tree with the right mouse button -Click Add Icon

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Table 6. (Cont.)

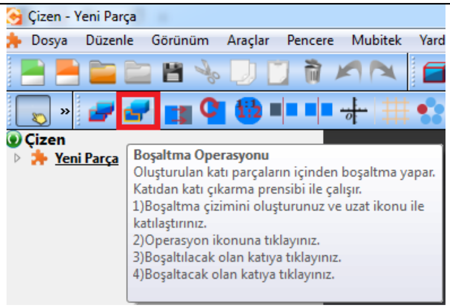
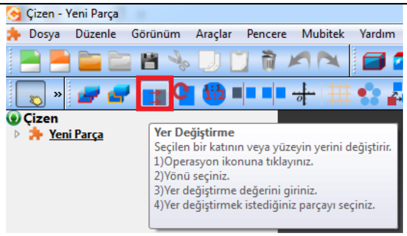
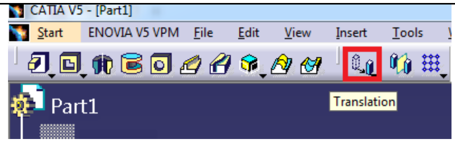
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Remove	 **Boşaltma Operasonu:** It allows us to remove the different operations in the product tree and make them into one piece.	 **Remove:** It allows us to remove the different operations in the product tree and make them into one piece.	-Click Yer Değiştirme Icon - Select direction - Enter the amount of displacement - Select the drawing to change location.	- Activate the part to be emptied (define in work object) - Select the product to be emptied from the product tree with the right mouse button -Click Remove Icon
Translation	 **Yer Değiştirme:** It allows us to relocate a selected solid / surface.	 **Translation:** It allows us to relocate a selected solid / surface. Select the drawing to be moved and click the Translation icon. Enter the distance from the Translation Definition menu or press and hold the direction arrows that appear on the figure and move to the desired location and click OK.	-Click Yer Değiştirme Icon - Select direction. - Enter the amount of displacement. - Select the drawing to change location.	-Click Translation Icon -Select direction - Enter the amount of displacement - Select the drawing to change location.

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Table 6. (Cont.)

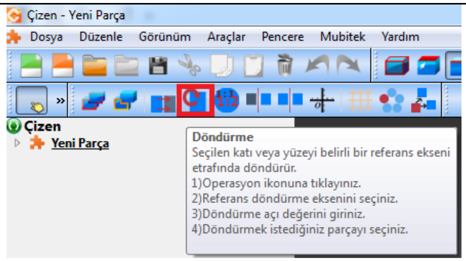
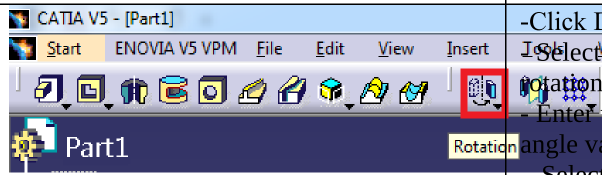
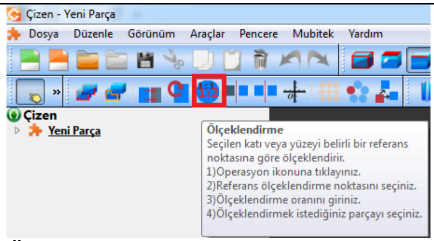
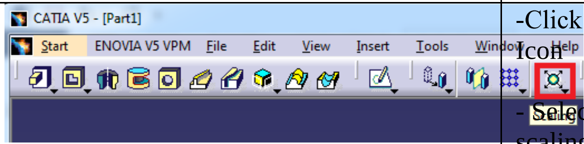
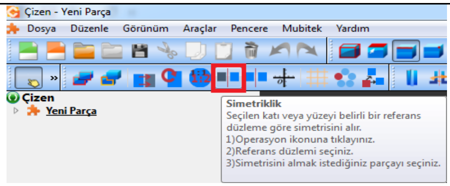
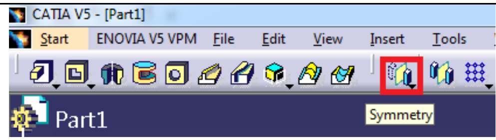
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Rotation	 **Döndürme:** Allows us to rotate a selected solid / surface around a specific reference axis.	 **Rotation:** Allows us to rotate a selected solid / surface around a specific reference axis. Select the drawing to rotate and click the Rotation icon. After entering data from Rotation Definition menu, select reference axis and click OK.	-Click Döndürme Icon - Select the reference rotation axis. - Enter the rotation angle value. - Select the part to be rotated.	- Click Rotation Icon -Ekrana gelen Enter data from the Rotation Definition menü - Select the reference rotation axis and click OK.
Scaling	 **Ölçeklendirme:** Allows us to scale a selected solid / surface to a specific reference point.	 **Scaling:** Allows us to scale a selected solid / surface to a specific reference point.	-Click Ölçeklendirme Icon - Select a reference scaling point - Enter the scaling ratio -Select the part	-Click Scaling Icon - Select a reference scaling point and the part - Enter the scaling ratio.
Symmetry	 **Simetriklik:** It allows us to take a symmetry of a selected solid / surface with respect to a specific reference plane.	 **Symmetry:** It allows us to take a symmetry of a selected solid / surface with respect to a specific reference plane.	-Click Simetriklik Icon - Select the reference plane. - Select the piece to be symmetrical.	- Click Symmetry Icon. -Select the reference plane. - Select the piece to be symmetrical an click Ok.

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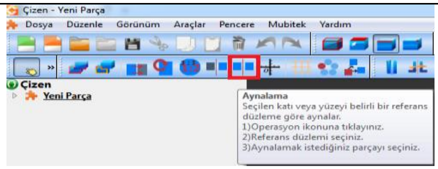
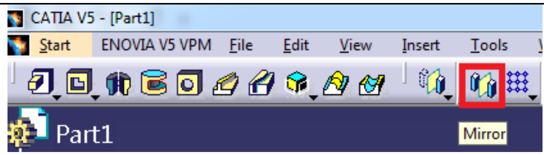
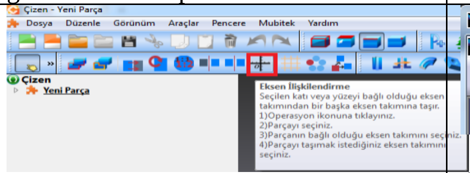
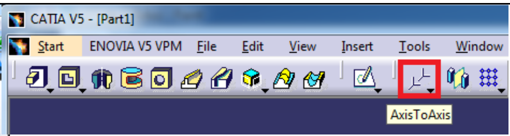
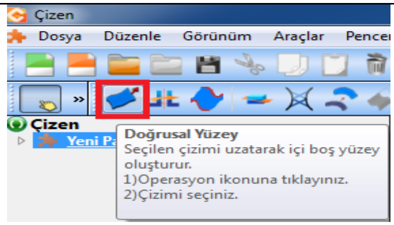
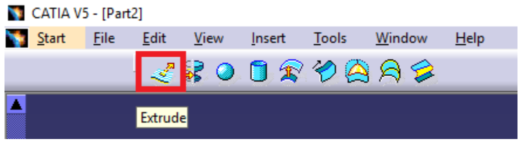
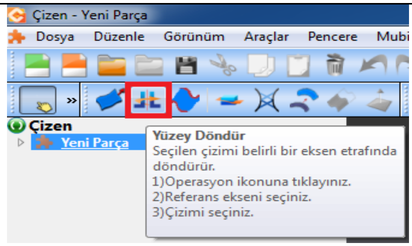
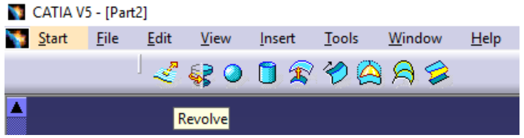
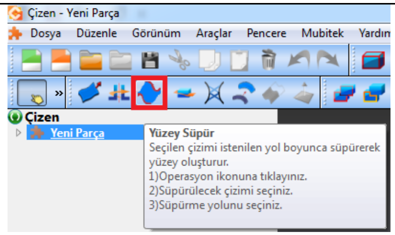
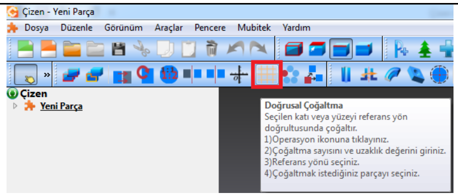
Table 6. (Cont.)

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Mirror	 **Aynalama:** Enables mirroring of a selected solid / surface relative to a given reference plane.	 **Mirror:** Enables mirroring of a selected solid / surface relative to a given reference plane.	- Click Aynalama Icon - Select the reference plane. - Select the part	- Click Mirror Icon - Select the reference plane. - Select the part
Axis to Axis	 **Eksen İlişkilendirme:** It enables us to move a selected solid / surface from the axis set to another axis set.	 **AxisToAxis:** It enables us to move a selected solid / surface from the axis set to another axis set.	- Click Eksen İlişkilendirme Icon and Select the part. - Select the axis tool to which the part is connected - Select the axis set from which you want to move the part.	- Click AxisToAxis Icon and Select the part - Select the axis tool to which the part is connected - Select the axis set from which you want to move the part.
Extrude	 **Doğrusal Yüzey:** It allows us to create a surface by extending the selected drawing.	 **Extrude:** It allows us to create a surface by extending the selected drawing.	- Click Doğrusal yüzey Icon - Select the drawing -Surface is formed.	- Click Extrude Icon - Select the drawing - Surface is formed.

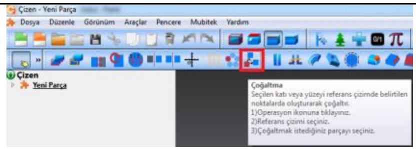
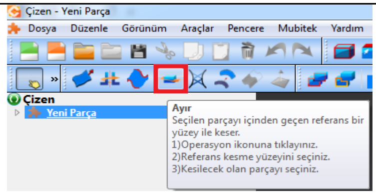
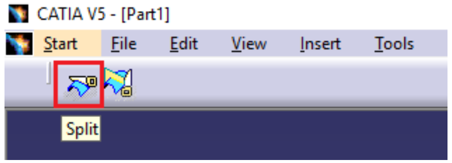
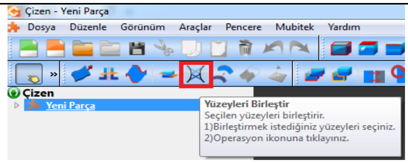
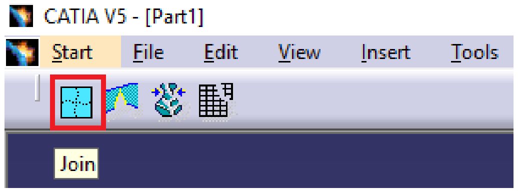
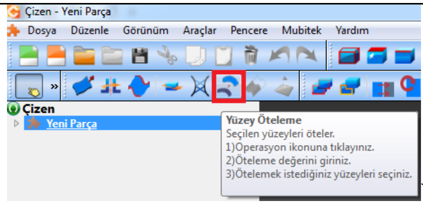
(Cont. on next page)

Table 6. (Cont.)

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Revolve	 **Yüzey Döndür:** Rotates the selected drawing to create a surface.	 **Revolve:** Rotates the selected drawing to create a surface.	- Click Yüzey döndür Icon - Select the reference axis - Select the drawing - Surface is formed.	- Click Revolve Icon - Select the drawing - Select the reference axis - Surface is formed.
Sweep	 **Yüzey Süpür:** Sweep the selected drawing along the desired path to create a surface.	 Sweep: A profile is offset along a selected curve.	- Click Yüzey süpür Icon - Select the drawing - Select the path.	- Click Sweep Icon - Select the drawing - Select the path.
Rectangular Pattern	 **Doğrusal Çoğaltma:** It allows us to reproduce a selected solid / surface in the direction of reference.	 **Rectangular Pattern:** It allows us to reproduce a selected solid / surface in the direction of reference.	- Click Doğrusal Çoğaltma Icon - Enter the number of replicates and the offset value - Select the reference direction and Select the part.	- Click Rectangular Pattern Icon - Enter the number of replicates - Select the reference direction - Select the part.

(Cont. on next page)

Table 6. (Cont.)

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
User Pattern	 **Çoğaltma:** It allows us to reproduce a selected solid / surface according to the points specified in the reference drawing.	 **User Pattern:** It allows us to reproduce a selected solid / surface according to the points specified in the reference drawing.	- Click Dairesel Çoğaltma Icon - Select the reference drawing - Select the part	- Click User Pattern Icon - Select the reference drawing - Select the part
Split	 **Ayır:** It allows us to cut the selected solid with the surface.	 **Split:** It allows us to cut the selected solid with the surface.	- Click Ayır Icon - Select the reference cutting surface - Select the part.	- Activate the part to be split -Select Split Icon - Specify the direction to be split and click Ok.
Join	 **Yüzeyleri Birleştir:** Allows us to combine selected surfaces.	 **Join:** Allows us to combine selected surfaces.	- Select the surfaces to be joined - Click Yüzey birleştir Icon.	- Click Join Icon - Select the surfaces to be joined - Select the desired states (eg tangency) - Click Ok.
Offset	 **Yüzey Öteleme:** Allows us to shift selected surfaces.	 **Offset:** Allows us to shift selected surfaces.	- Click Yüzey öteleme Icon - Select the surface to be offset - Enter the value to be offset.	- Click Offset - Select the surface to be offset - Enter the value to be offset.

3.5. Comparison of Çizen Sketch Menu with Catia V5

Sketch menu is most basic modul in Çizen and Catia program. It makes it possible to create a 3D design. It allows 2D design. There are multiple methods in Sketcher Workbench. This menu consist of sketch, sketch analysis, Point by clicking, line, rectangle, circle, three point circle, three point arc, ellipse, draw spline, angle and first length chamfer, first and second length chamfer, quick trim, trim, mirror, translate, rotate, scale, offset, project 3d elements, exit workbench, usual, low light, no 3d background, pickable visible background, unpickable low intensity background, fix, angle, distance, paralellism, perpendicular, radius/diameter, concentricity, tangency, coincidence, symmetry, horizontal measure direction and vertical measure direction.

There is not much difference between Çizen and Catia program in the sketch menu. The biggest difference in this menu is that, there are too many sketching alternatives in the Catia program. Also in sketch, the relationship identification can be practiced in Catia thanks to right-click window. But in Çizen this is more difficult due to lack of right-click window. In addition to this in Çizen there are different commands all each relationship types, but in catia there is a commands to do this. This is adventageous for the user.

Table 7. Summary Table Comparison of Çizen Sketch Menu with Catia

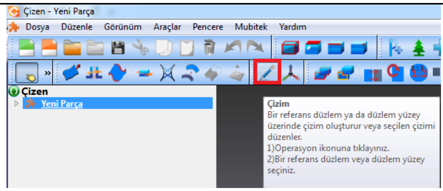
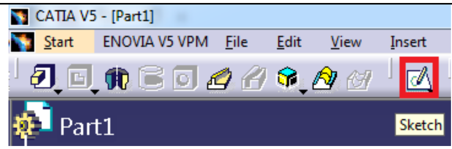
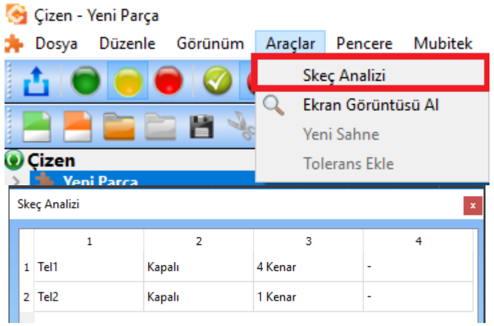
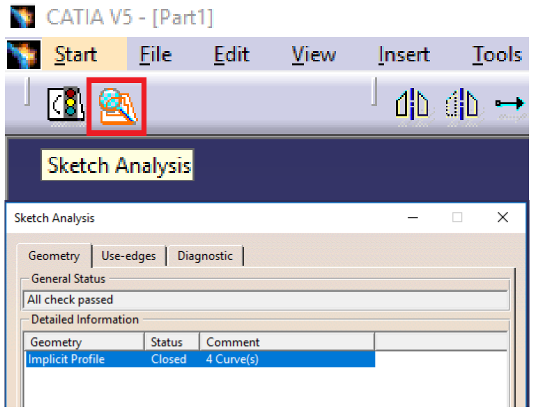
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Sketch	 **Çizim:** Switches to drawing / sketch environment. After selecting the drawing icon, select a reference plane or plane surface.	 **Sketch:** : Switches to drawing / sketch environment. After selecting the drawing icon, select a reference plane or plane surface.	- Click Çizim Icon -Select a reference plane or plane surface.	- Click Sketch -Select a reference plane or plane surface.
Sketch Analysis	 **Skeç Analiz:** Shows the closedness and openness of the lines with each other in the drawing / sketch environment.	 **Sketch Analysis:** Shows the closedness and openness of the lines with each other in the drawing / sketch environment.	- Click Araçlar Icon - Click on the sketch analysis.	- Click Sketch Analysis Icon

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Table 7. (Cont.)

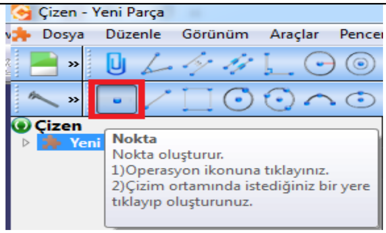
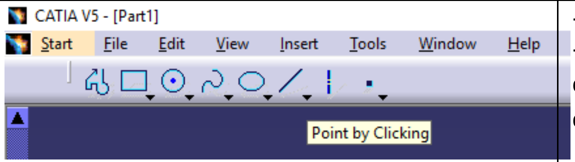
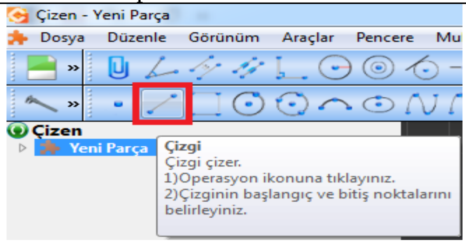
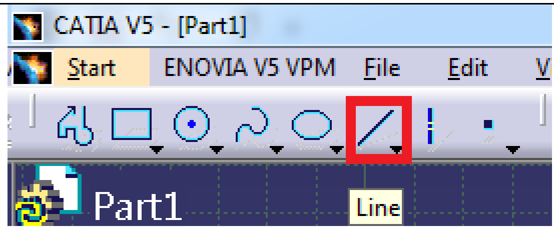
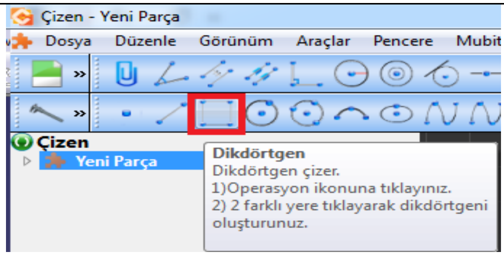
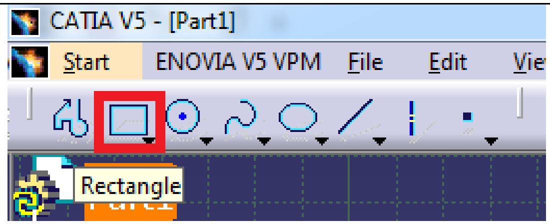
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Point by Clicking	 **Nokta:** Create point.	 **Point by Clicking:** Create point.	- Click Nokta Icon - Click to create a desired place in the drawing environment.	- Click Point by Clicking Icon - Click to create a desired place in the drawing environment.
Line	 **Çizgi:** Create line.	 **Line:** Create line.	- Click Çizgi Icon - Specify the start and end points of the line.	- Click Line Icon - Specify the start and end points of the line.
Rectangle	 **Dikdörtgen:** Create rectangle	 **Rectangle:** Create rectangle.	- Click Dikdörtgen Icon. - Click on two different places to create a rectangle.	- Click Rectangle Icon - Click on two different places to create a rectangle.

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Table 7. (Cont.)

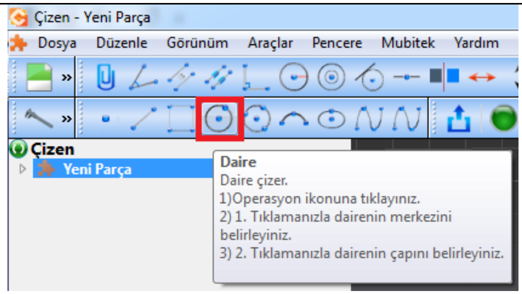
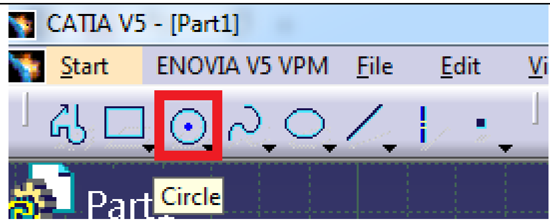
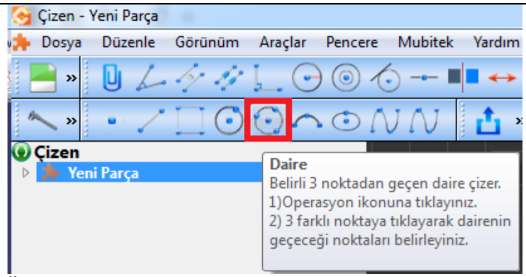
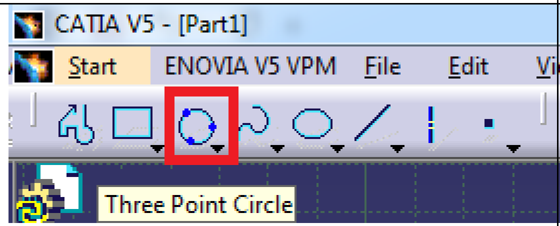
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Circle	 **Daire:** It forms a circle with a center and a certain point.	 **Circle:** It forms a circle with a center and a certain point. After clicking on the circle icon, the first click determines the center of the circle, the second click determines the diameter of the circle and forms the circle.	- Click Daire Icon - The first click determines the center of the circle - The second click determines the diameter of the circle.	- Click Circle Icon - The first click determines the center of the circle - The second click determines the diameter of the circle.
Three Point Circle	 **Üç noktalı Daire:** It allows us to form a circle passing through three points.	 **Three Point Circle:** It allows us to form a circle passing through three points. After clicking on the Three Point Circle icon, click on the three different points that the circle will pass through and create the circle.	- Click Daire Icon - Click on three different points.	- Click Three Point Circle Icon - Click on three different points.

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Table 7. (Cont.)

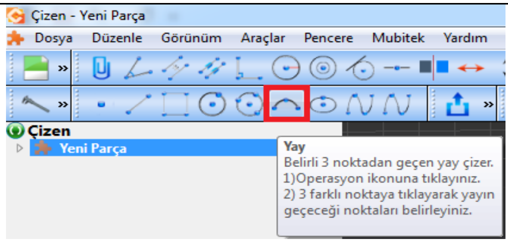
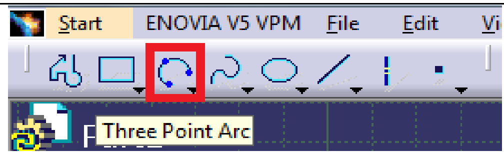
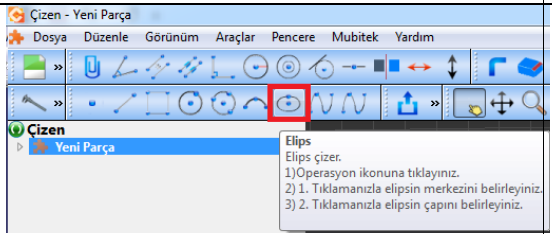
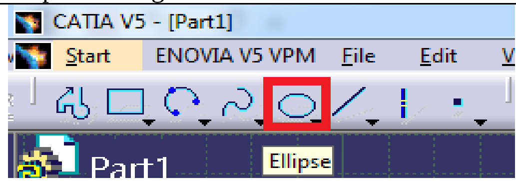
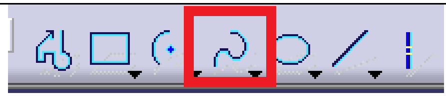
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Three Point Arc	 **Yay:** Create arcs.	 **Three Point Arc:** Create arcs. After clicking on the Three Point Arc icon, the arc is created by clicking on the three different points that the arc will pass through.	- Click Yay Icon - Click on the three different points where you want the broadcast to pass.	- Click Three Point Arcs Icon - Click on the three different points where you want the broadcast to pass.
Ellipse	 **Elips:** Create ellipse.	 **Ellipse:** Create ellipse. After clicking the ellipse icon, the ellipse is created by first determining the center of the ellipse, the second click determines the diameter of the horizontal and the third click the diameter of the vertical.	- Click Elips Icon - The first click determines the center of the ellipse - The second click determines the diameter of the ellipse.	-Ellipse simgesine tıklanır - The first click determines the center of the ellipse - The second click determines the diameter of the ellipse and the third click determines the vertical diameter of the ellipse
Draw Spline	 **Draw Spline:** It enables us to construct the spline which is tangent to the control points.	 **Spline:** It enables us to construct the spline which is tangent to the control points.	- Click Kontrol Eğrisi Icon - Select the points you want the curve to be tangent to - Right-click to end the process.	-Click on the operation icon -Create the points you want the curve to pass through -Click on the ESC.

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Table 7. (Cont.)

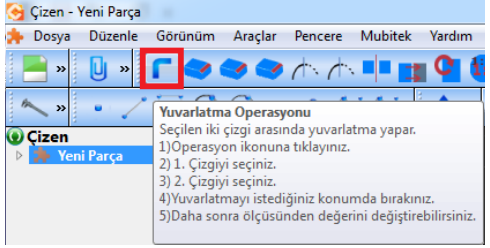
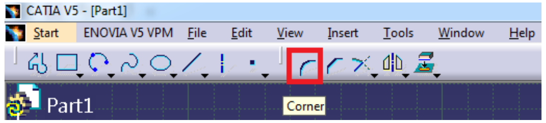
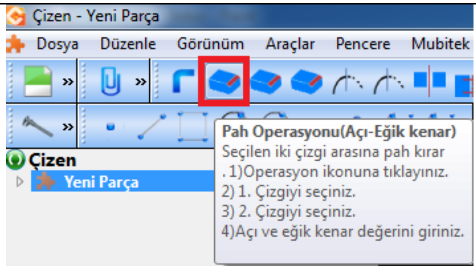
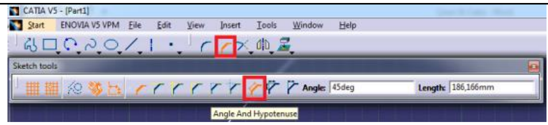
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Corner	 **Yuvarlatma:** Rounding between two selected lines.	 **Corner:** Rounding between two selected lines. Click the Corner icon and select the lines you want to round. Leave the rounding in the desired position. The value can then be changed over the measurement.	- Click Yuvarlatma Icon - Select the first line. - Select the second line. - Leave the rounding in the desired position. - Then enter the value on the measurement.	- Click Corner Icon - Select the first line. - Select the second line. - Leave the rounding in the desired position. - Then enter the value on the measurement.
Angle And Hypotenuse Chamfer	 **Pah Operasyonu (Açı-Eğik kenar):** Allows us to chamfer between two selected lines.	 **Chamfer (Angle And Hypotenuse):** Allows us to chamfer between two selected lines. Select the first and second line that you want to chamfer after clicking on the Chamfer icon. After selecting Angle And Hypotenuse from the Sketch tools menu that appears on the screen, click the shape from the position locator. Then change the angle and length values on the shape.	- Click Pah Operasyonu (Açı-Hipotenüs) - Select the first line and the second line. - Enter the angle and length of the hypotenuse.	- Click Chamfer (Angle And Hypotenuse) - Select the first line and the second line. - Choose Angle And Hypotenuse from the Skecth tools menu and select a location. - Enter the angle and length of the hypotenuse.

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Table 7. (Cont.)

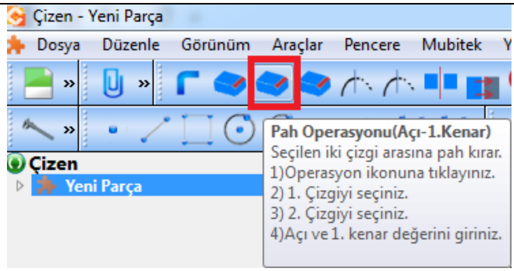
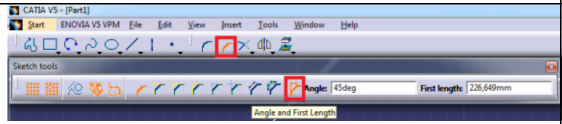
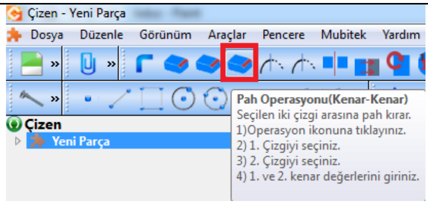
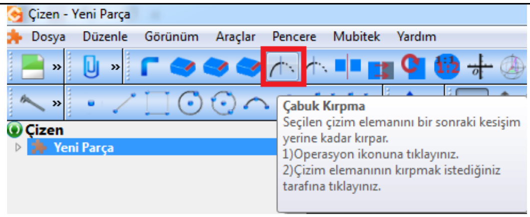
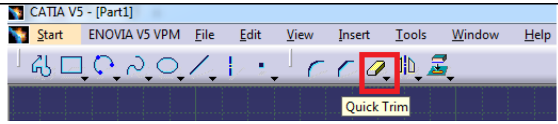
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Angle And First Length Chamfer	 **Pah Operasyonu (Açı-Kenar):** Allows us to chamfer between two selected lines.	 **Chamfer (Angle and First Length):** Allows us to chamfer between two selected lines. Select the first and second line that you want to chamfer after clicking on the Chamfer icon. After selecting Angle and First Length from the Sketch tools menu that appears, click on the figure from the position determination.	- Click Pah Operasyonu (Açı-Kenar) - Select the first line. - Select the second line. - Enter the angle and length of the first edge.	- Click Chamfer (Angle And Hypotenuse) - Select the first line. - Select the second line. - Select Angle And First Length from the Sketch tools menu and select a location. - Enter the angle and length of the first edge.
First and Second Length Chamfer	 **Pah Operasyonu (Kenar -Kenar):** Allows us to chamfer between two selected lines.	 **Chamfer (First and Second Length):** Allows us to chamfer between two selected lines. Select the first and second line that you want to chamfer after clicking on the Chamfer icon. After selecting First and Second Length from the Sketch tools menu that appears, click on the figure from the position determination.	- Click Pah Operasyonu (Kenar-Kenar) Icon - Select the first line. - Select the second line. - Enter the first and second edge length.	-Click Chamfer (Angle And Hypotenuse) Icon - Select the first line. - Select the second line. - Select First And Second Length from the Sketch tools menu and select a location. - Enter the first and second edge length.
Quick Trim	 **Çabuk Kırpma:** Crops the selected drawing elements to the next intersection.	 **Quick Trim**: Crops the selected drawing elements to the next intersection. After clicking the Quick Trim icon, click on the side of the drawing to be cropped.	- Click Çabuk Kırpma Icon - Click on the side of the drawing to crop.	- Click Quick Trim Icon - Click on the side of the drawing to crop.

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Table 7. (Cont.)

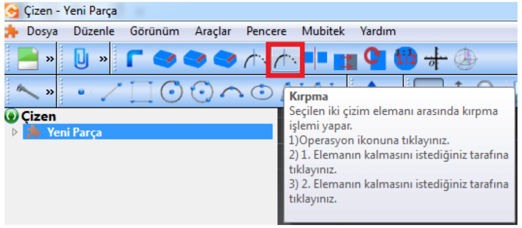
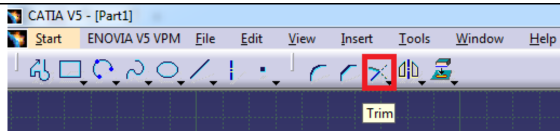
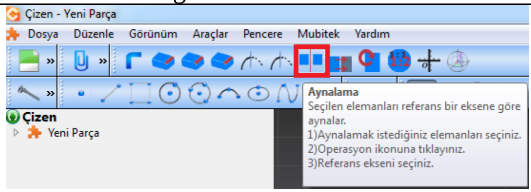
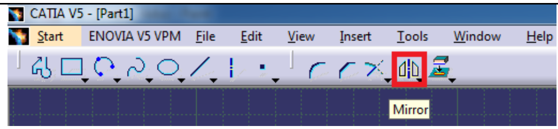
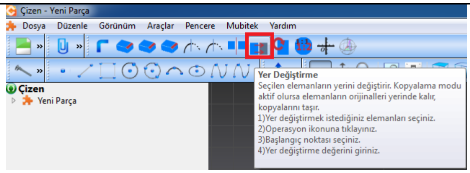
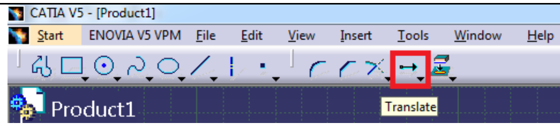
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Trim	 **Kırpma:** Allows us to crop between two selected drawing elements.	 **Trim:** Allows us to crop between two selected drawing elements.	- Click Kırpma Icon - Click on the side where you want the first element to remain. - Click on the side where you want the second element to remain.	- Click Trim Icon - Click on the side where you want the first element to remain. - Click on the side where you want the second element to remain.
Mirror	 **Aynalama:** Allows mirroring of selected elements with respect to a specific reference axis.	 **Mirror:** Allows mirroring of selected elements with respect to a specific reference axis. Click on the Mirror icon after selecting the drawing to mirror. Then the reference plane is selected and click OK.	- Select the part to mirror -Click Aynalama Icon -Select the reference plane.	- Select the part to mirror - Click Mirror Icon - Select the reference plane.
Translate	 **Yer Değiştirme:** Replaces the selected elements. If the copy mode is active, it allows us to leave the originals of the elements in place and move the copies.	 **Translate:** Replaces the selected elements.	- Select the drawing to change location. - Click Yer Değiştirme Icon - Select the starting point - Enter the amount of displacement.	- Select the drawing to change location -Translate simgesine tıklanır - Select the starting point - Enter the amount of displacement.

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Table 7. (Cont.)

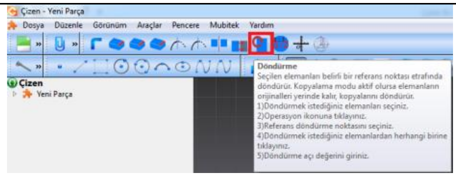
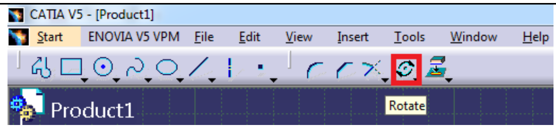
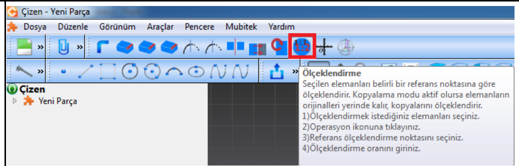
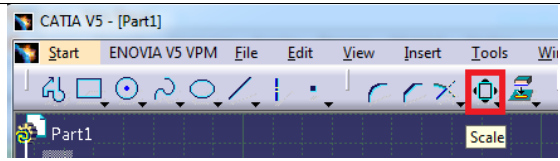
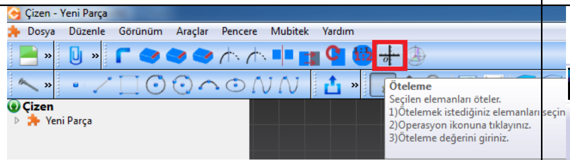
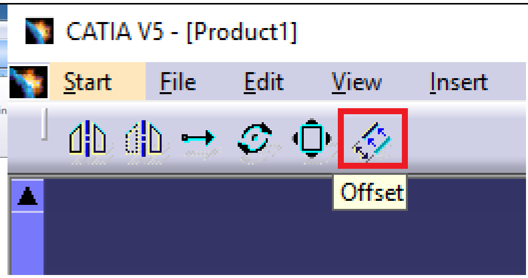
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Rotate	 **Döndürme:** Rotates selected elements around a specific reference axis.	 **Rotate:** Rotates selected elements around a specific reference axis.	- Select the drawing to rotate. - Click Döndürme Icon - Select the reference rotation point. - Click on any of the elements you want to rotate. - Enter the amount of rotation angle.	- Select the drawing to rotate. - Click Rotate Icon - Select the reference rotation point. - Enter the amount of rotation angle and pieces data.
Scale	 **Ölçeklendirme:** Scales selected elements to a specific reference point.	 **Scale:** Scales selected elements to a specific reference point.	- Select the elements you want to scale and Click Ölçeklendirme Icon - Select a reference scaling point - Enter the scaling ratio.	- Select the elements you want to scale - Click Scale and Select a reference scaling point and Enter the scaling ratio.
Offset	 **Öteleme:** Shifts selected elements.	 **Offset:** Shifts selected elements.	- Select the elements to be offsetted. - Click Öteleme Icon - Enter a offset value.	- Select the elements to be offsetted. - Click Offset - Enter a offset value.

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Table 7. (Cont.)

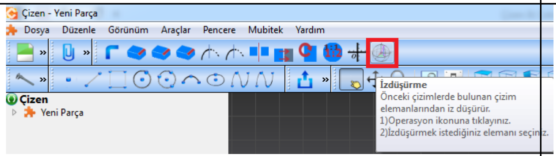
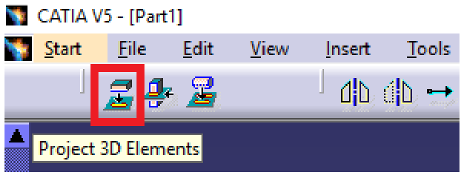
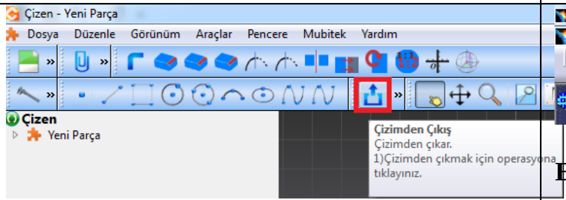
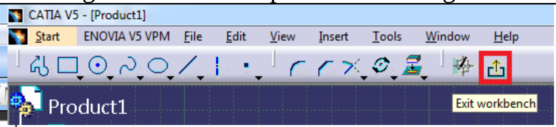
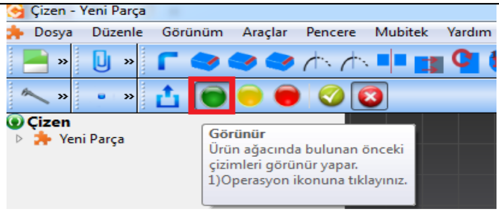
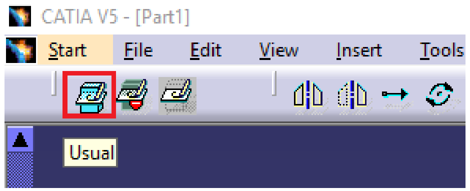
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Project 3D Elements	 **İzdüşürme:** It projectes from the drawing elements in the previous drawings.	 **Project 3D Elements:** It projectes from the drawing elements in the previous drawings.	- Click İzdüşürme Icon - Select the element you want to project.	- Click Project 3D elements Icon - Select the element you want to project.
Exit Workbench	 **Çizimden Çıkış:** Exits the drawing.	 **Exit workbench:** Exits the drawing.	-Click Çizimden Çıkış Icon	- Click Exit Workbench Icon
Usual	 **Görünür:** Makes the previous drawings visible in the design tree visible.	 **Usual:** Makes the previous drawings visible in the design tree visible.	- Click Görünür Icon	- Click Usual Icon

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Table 7. (Cont.)

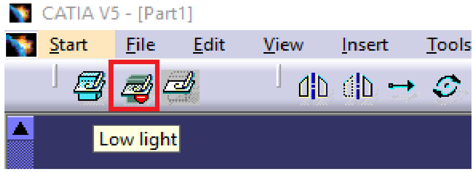
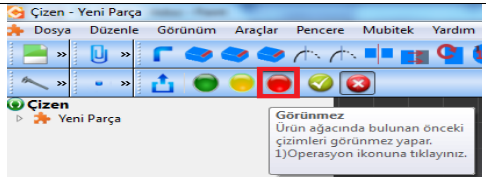
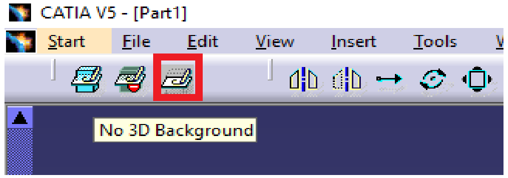
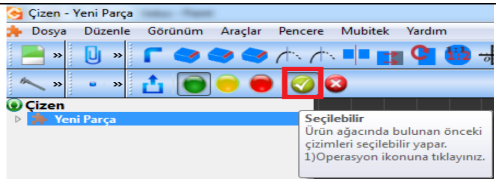
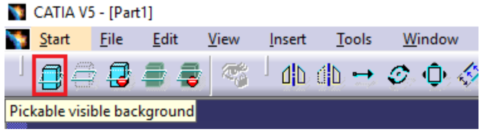
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Low Light	 **Saydam:** Makes the previous drawings in the design tree transparent.	 **Low Light:** Makes the previous drawings in the design tree transparent.	- Click Görünür Icon	- Click Low Light Icon
No 3D Background	 **Görünmez:** Makes previous drawings in the design tree invisible.	 **No 3D Background:** Makes previous drawings in the design tree invisible.	- Click Görünür Icon	- Click No 3D Background Icon
Pickable Visible Background	 **Seçilebilir:** Makes the previous drawings in the design tree selectable.	 **Pickable visible background**: Makes the previous drawings in the design tree selectable.	- Click Görünür Icon	-Click Pickable visible background Icon

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Table 7. (Cont.)

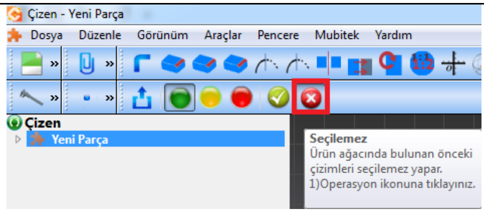
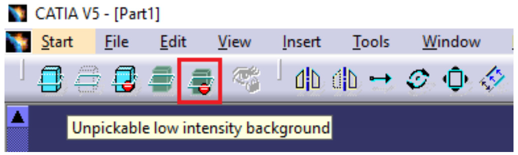
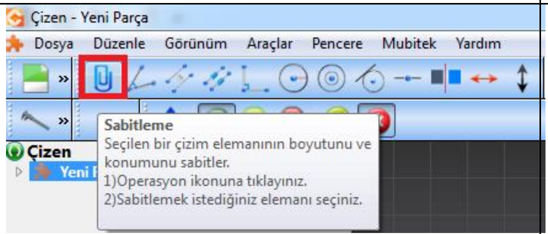
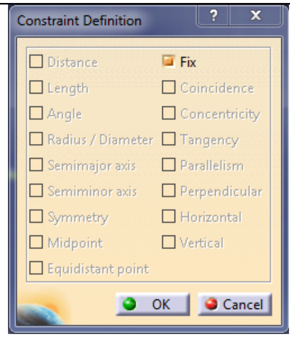
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Unpickable low intensity background	 **Seçilemez:** Makes the previous drawings in the design tree unselectable.	 **Unpickable low intensity background:** Makes the previous drawings in the design tree unselectable.	- Click Görünür Icon	- Click Unpickable low intensity background Icon
Fix	 **Sabitleme:** Fixes the selected drawing.	 **Fix:** Fixes the selected drawing. Click Constraints Defined in Dialog Box, check Fix and click OK.	- Select the drawing to be fixed - Click Sabitleme Icon	- Select the drawing to be fixed - Click Constraint Defined in Dialog Box, check Fix and click OK.

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Table 7. (Cont.)

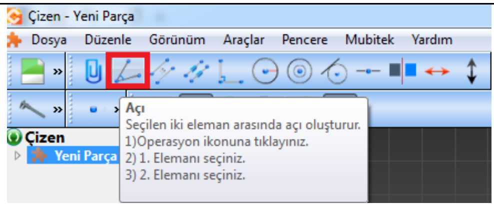
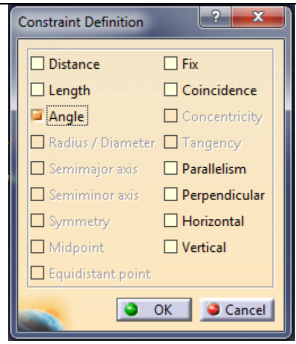
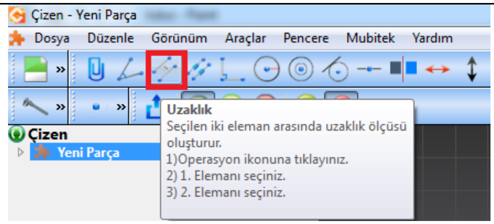
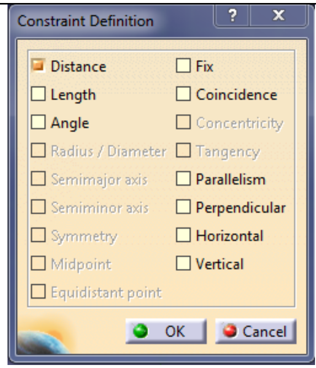
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Angle	 **Açı:** Creates an angle between two selected elements.	 **Angle:** Creates an angle between two selected elements. Click Constraints Defined in Dialog Box, check Angle and click OK. Then click the angle value on the figure and enter the desired angle value.	- Click Açı Icon - Select the drawings. - Give the angle value on the figure.	- Select the drawings. - Click Constraint Defined in Dialog Box, check Angle and click OK. - Give the angle value on the figure.
Distance	 **Uzaklık:** Gives the distance between the edges and surfaces selected on the drawing.	 **Distance:** Gives the distance between the edges and surfaces selected on the drawing.	- Click Uzaklık Icon - Select the drawings **- Change the distance value on the shape.**	- Select the drawings - Click Constraint Defined in Dialog Box, check Distance and click OK - Change the distance value on the shape.

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Table 6. (Cont.)

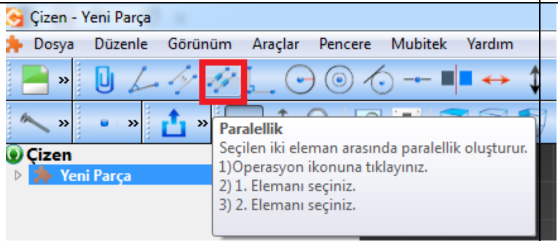
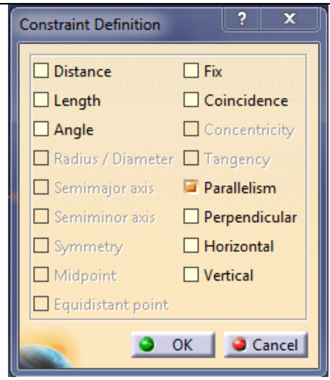
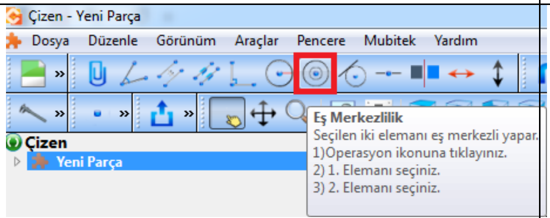
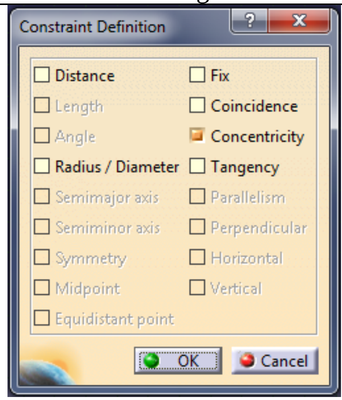
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Paralellism	 **Paralellik:** Makes the selected surfaces parallel to each other on the drawing.	 **Parallelism:** Makes the selected surfaces parallel to each other on the drawing.	- Click Paralellik Icon - Select the drawings to be parallelized.	- Select the drawings to be parallelized - Click Constraint Defined in Dialog Box, check Parallelism and click OK.
Concentricity	 **Eş Merkezlilik:** It makes two objects of circular cross section concentric.	 **Concentricity:** It makes two objects of circular cross section concentric.	- Click Eş Merkezlilik Icon - Select the circles you want to concentrate.	- Select the circles you want to concentrate. - Click Constraint Defined in Dialog Box, check Concentricity and click OK.

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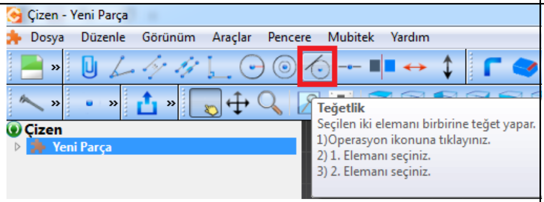
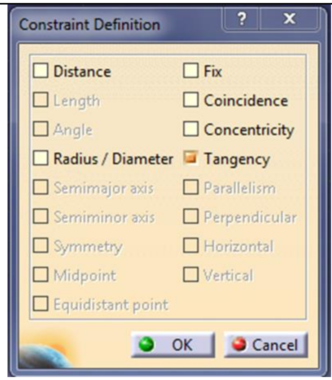
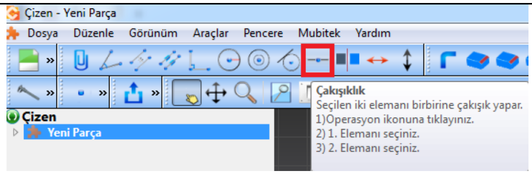
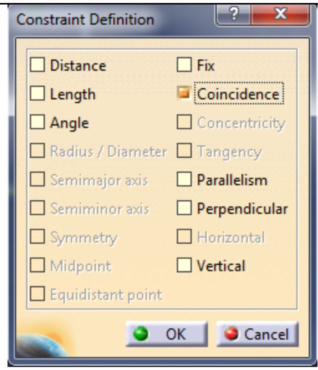
Table 7. (Cont.)

<table>
<tr><th>COMMAND</th><th>ÇİZEN ICON AND EXPLANATION</th><th>CATIA ICON AND EXPLANATION</th><th>ÇİZEN COMMAND USAGE</th><th>CATIA COMMAND USAGE</th></tr>
<tr><td>Tangency</td><td>

Teğetlik: It makes two objects tangent to each other, one of which is circular and the other is straight.</td><td>

Tangency: It makes two objects tangent to each other, one of which is circular and the other is straight.</td><td>- Click Teğetlik Icon
- Select the drawings to be tangent to.</td><td>- Select the drawings to be tangent to.

- Click Constraint Defined in Dialog Box, check Tangency and click OK.</td></tr>
<tr><td>Coincidence</td><td>

Çakışıklık: Overlaps selected surfaces on the drawing.</td><td>

Coincidence: Overlaps selected surfaces on the drawing. Click Constraints Defined in Dialog Box, check Coincidenceand click OK.</td><td>- Click Çakışıklık Icon

-Select the first element

-Select the second element.</td><td>- Select the drawings to be overlapped by the centers

- Click Constraint Defined in Dialog Box, check Coincidence and click OK.</td></tr>
</table>

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Table 7. (Cont.)

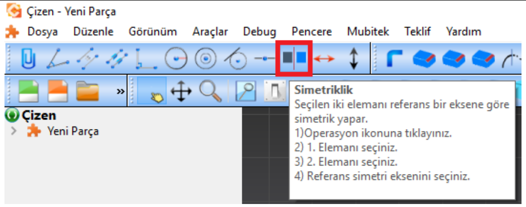
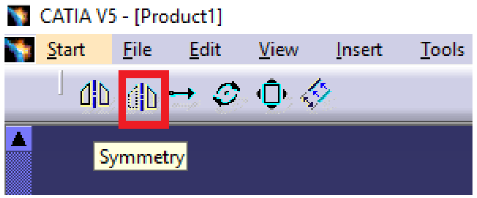
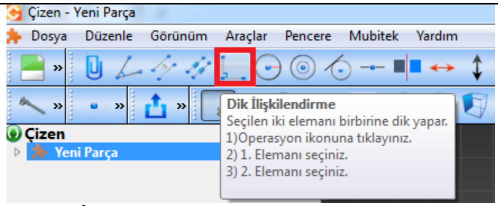
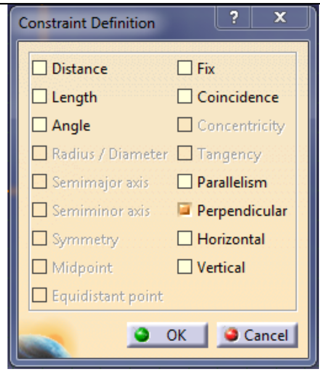
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Symmetry	 **Simetriklik:** It makes the selected two elements symmetrical with respect to a reference axis.	 **Symmetry:** It makes the selected two elements symmetrical with respect to a reference axis.	- Click Simetriklik Icon -Select the first element -Select the second element - Select the reference symmetry axis.	- Click Symmetry Icon - Select the element to be symmetry - Select the reference symmetry axis.
Perpendicular	 **Dik İlişkilendirme:** Makes the selected surfaces perpendicular to each other on the drawing.	 **Perpendicular:** Makes the selected surfaces perpendicular to each other on the drawing.	- Click Dik İlişkilendirme Icon - Select the drawings that you want to be vertical.	- Select the drawings that you want to be vertical -Click Constraint Defined in Dialog Box, check Perpendicular and click OK.

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Table 7. (Cont.)

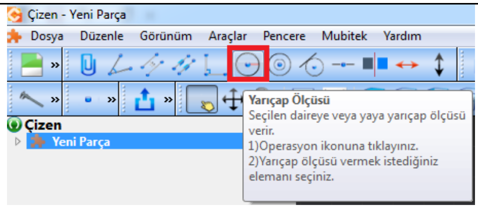
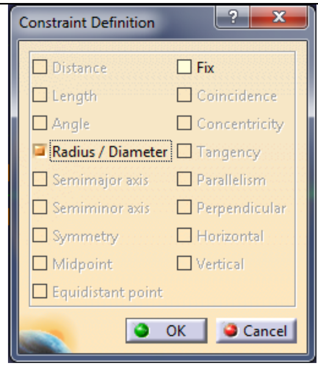
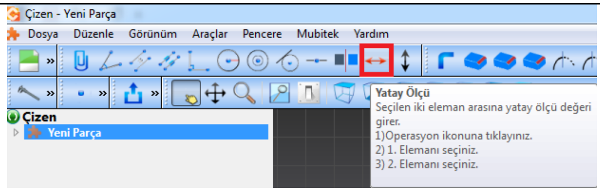
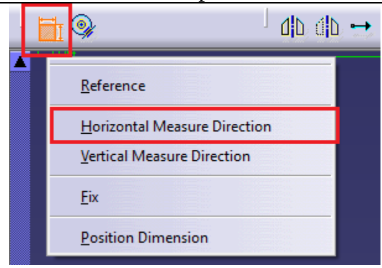
COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Radius/ **Diameter**	 **Yarıçap Ölçüsü:** Gives the radius of the selected circle or pedestrian radius.	 **Radius/Diameter:** Gives the radius of the selected circle or pedestrian radius.	- Click Yarıçap Ölçüsü Icon - Select the element whose radius is to be measured.	- Select the element whose radius is to be measured and Click Constraint Defined in Dialog Box, check Radius/Diameter and click OK - The radius value can be changed via the figure.
Horizontal Measure Direction	 **Yatay Ölçü:** Gives only horizontal dimension between two selected elements.	 **Horizontal Measure Direction:** Gives only horizontal dimension between two selected elements.	- Click Yatay ölçü Icon -Select the first element -Select the second element.	- Click Constraint Command - Select the drawing element - Right-click and select Horizontal Measure Direction.

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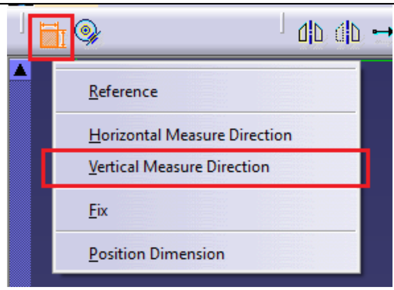
Table 7. (Cont.)

COMMAND	ÇİZEN ICON AND EXPLANATION	CATIA ICON AND EXPLANATION	ÇİZEN COMMAND USAGE	CATIA COMMAND USAGE
Vertical Measure Direction	 **Dikey Ölçü:** Gives only vertical dimension between two selected elements.	 **Vertical Measure Direction:** Gives only vertical dimension between two selected elements.	- Click Yatay ölçü Icon -Select the first element -Select the second element.	- Click Constraint Command - Select the drawing element - Right-click and select Vertical Measure Direction.

CHAPTER 4

USER EXPERIENCE RESEARCH AND RESULTS

In this chapter includes general framework of research, procederues of research, data collection methods and results.

4.1. General Framework of Research

User-centered design involves a different method for interface design, emphasizing the user’s experience. In broad terms, user-centered design is a process in which the needs, target tasks, and limitations of the end-user of an interface or document are considered at each stage of the design phase (Norman and Stephen 1986).

According to Wikipedia, “User-centered design can be characterized as a multi-stage problem solving process that not only requires designers to analyze and foresee how users are likely to use an interface, but to test the validity of their assumptions with regards to user behavior in real world tests with actual users.” To design a good user interface is a challenging task, needing multiple iterations from design and prototype to evaluation (Shiu and et all, 2007).

The term “User Experience” has been used by people involved in the Human-Computer Interaction of the 1970s and 80s, and in particular User-Oriented Design. In 1993, with Don Norman began to gain popularity while working in Apple as the “User Experience Architect”. The introduction of user experience into the literatüre has been made through the papers published at “Computer-Human Interaction” congress (Norman and et all, 1995).

User-centered design, user friendly systems, ergonomically designed systems, and so forth have been around for quite some time (Rouse, 1991). But a common mistake made by most of the companies is to ignore user tests while producing a new product. Mostly a product meets the users at the very end producing step or even after putting it on the market. They sometimes jump directly to the marketing and sales without even any test or content control. An idea may seem brilliant but it can be illusory, the user thoughts can never be truly estimated. Not letting users into the process usually leads failure, but the thing is very simple actually. From the first step, users‟ thoughts, advices and feedbacks should be observed till the end (Kantar, 2015).

4.1.1. Design and Participants of Research

The research is a case study with mixed design methodology, in which both quantitative and qualitative approaches were employed and combined into the research methodology of a single study (Tashakkori and Teddlie 1998). This study employed both triangulation and explanatory method type of mixed method design to validate and compare quantitative findings with qualitative ones and used qualitative data to expand on or explain findings of quantitative data.

In this study, persona methods was used. Eleven participants enrolled in the study. All of the participants are experienced sheet metal forming design. According to Nielsen (Nielsen, 1994) more than 75% of the usability problems can be found with five participants. As this number approaches 15, almost all of the usability problems can be identified (Nielsen, and Landauer, 1993). Therefore, eleven participants are sufficient to identify major usability problems on the inspected.

During the tests, participants were encouraged to think aloud as much as possible while completing the task, and the participants' performances were observed and recorded by the researcher. Used metrics in participants' eyes movements are below in the Table 8.

Table 8. Metrics Used In the Research

Metrics	**Aim of Utilization**
Interval Duration	It shows the time taken for users to complete their tasks.
Visit Duration	It shows the fixing time in the menu. The less fixation the easier he found the task.

(Cont. on next page)

Table 8. (Cont.)

Visit Count	It shows the number of visits to the menu. The more visits, the more difficult he is to find what he is looking for. Similar to visit duration.
Click Count	It shows the number of clicks on the menu. It indicates that, there may be a usability problem if there are many clicks on the menu.

Major phases of the study consist of quantitative and qualitative phases. The data collection phases of the study can be divided as beginning, during and after the study.
In the research we used a lot of Design Thinking Methods which is seen Table 9.

Table 9. Design Thinking Methods Used in the Research

	DESIGN THINKING METHODS	**REASON OF USE**
Methods of observing human experience	Interviewing	Used to capture the correct information and provide direct feedback from the user.
	Contextual inquiry	It was used to determine sheet metal forming design step by step.
	Buy a future	It has been used to reveal the features required from the users in the program.
	Think aloud testing	It was used to narrate how users do the design and the program.
	Critique	Used in the evaluation of the research results by the company.
	System usabilitiy scale	Used to measure the efficiency and quality of the program.
	Competitive product survey	It was used to analyze the features of the program by conducting competitive product research.
Methods for analyzing challenges and opportunities	Persona	Used to determine which users shold conduct the survey.
	Experience diagramming	It was used to map the design experiences of different designers.
	Affinity map	It was used to classify the similar suggestions that were obtained as a result of the research.
	Problem tree analysis	Used to elaborate issues into main headings.
Methods for envisioning future	Value priorization	It was used to separate the recommendations according to value-feasibility issues.
	Subject matter expert	It was used to determine future trends in the meeting with CAD distributors.

4.1.2. Prosederues of Research

All participants completed a pre-test questionnaire to get demografic information and sheet metal forming design methodologies. After that, the participants were informed about the test, user-test, think-aloud methods and the eye tracking system. After short explanation of the study, participants were asked to complete 16 tasks and they were encouraged to take time and we told to them “this was not a racing”.

4.1.3. Data Collection, Analysis and Instrument of Research

In this research data were collected through both quantitative and qualitative methods. The experiments were conducted in Mubitek and other sheet metal forming design companies located in Bursa. Eye movements of participants were recorded by Tobii X2-60 Eye Tracker. In addition to the eye tracker data, the participants were observed while they were working on the tasks and stopwatch was kept.

Figure 13. Tobii Pro X2-60 Eye Tracker Device

Data was collected by eye tracker, questionnaires and interviews. Eleven participants tested the usability of the ÇİZEN CAD with the help of eye-tracking technology, supported by observation of the participants, a pre-test questionnaire, which was used to assess participants'previous experience with CAD, a post-test questionnaire and a semi-structured individual interviews about the participants' perceptions at the end of the study. All data collection instruments were in Turkish.

At the end of the study, post-test questionnaire about the participants' perceptions was conducted. Post-test, System Usability Scale was adopted from Brooke.

Table 10. Data Collection Prosedures of Reserach

Phase of the Research	**Data Collection Procedures**
Beginning	Pre-test Questionnaire about demografic information and exploring sheet metal forming design methodologies
During	Eye Tracking, Think-aloud, Observations, Stopwatch, video recording.
End	Post-test Questionnaire Interview

The quantitative data analysis consisted of descriptive statistics. Tobii Studio software was used to analyze the data collected by eye tracker. Sessions were recorded and analyzed to identify the interval duration, visit duration, visit count and click count. Table 14 there are some photos from research.

Qualitative data were interviews and note taken during eye-tracking test. Qualitative data were analyzed together with eye-tracking data. After test, the participants and researcher was conducted in the form of a semi-structured interview in which the interviewees reflected on his feeling and perceptions about usability of the CAD.

This interview acted as an informal exchange of observations and views. Hence, no formal interview schedules were utilized. The interviews were tape-recorded after obtaining consent from the interviewees, and later, they were transcribed verbatim for data analysis.

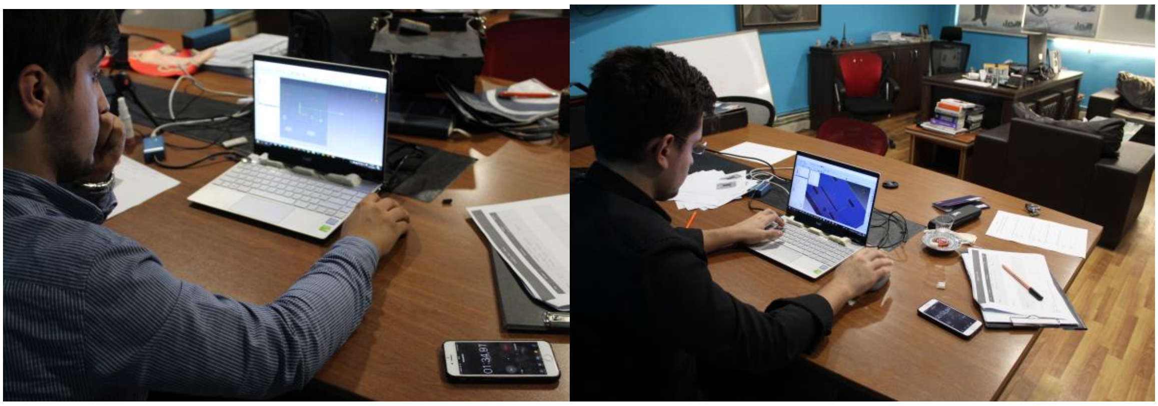


Figure 14. Photos from Research with Eye Tracker Device

4.2. Results

Results constist of three parts which are pret-test, eye tracking analysis and post-test.

4.2.1. Pre-Test Questionare Results

In this study, the pre-test questionnaire was used to evaluate participants' previous utilization and involvement details in the CAD program. Pre-test consist of two parts. First part having 6 questions is related Demografic Information. Second part is related sheet metal forming design methodology. It has 11 questions.

4.2.1.1. Demografic Information

The majority of participants age are 30 and under.

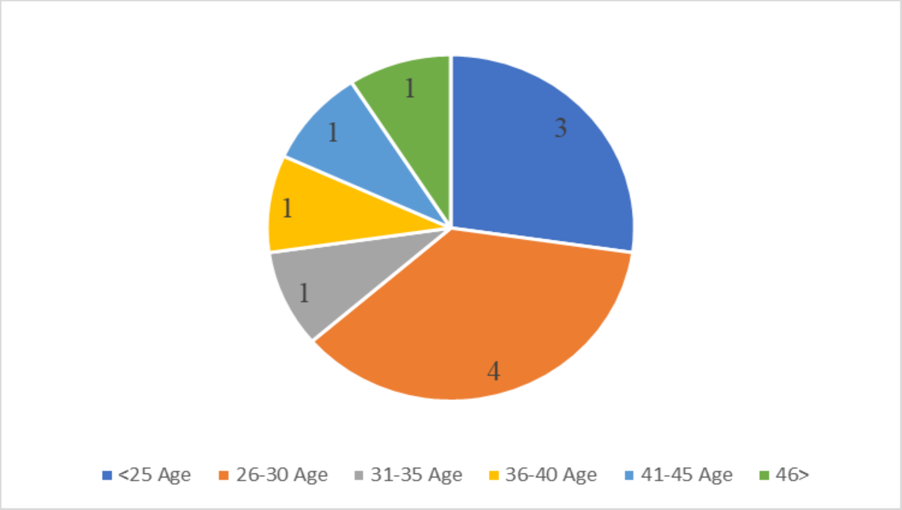


Figure 15. Ages of Participants

The majority of participants are graduate.

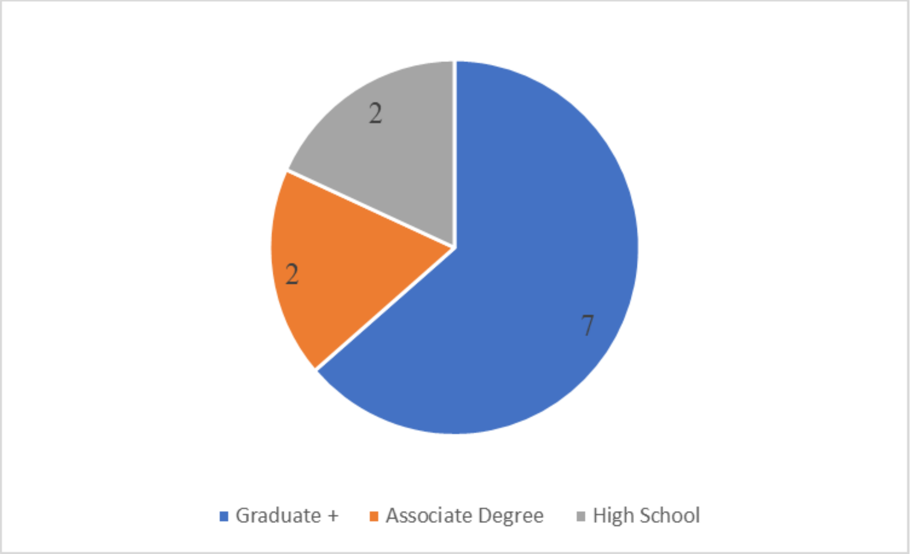


Figure 16. Graduation Level of Participants

The majority of participants job experience are 3 year and under.

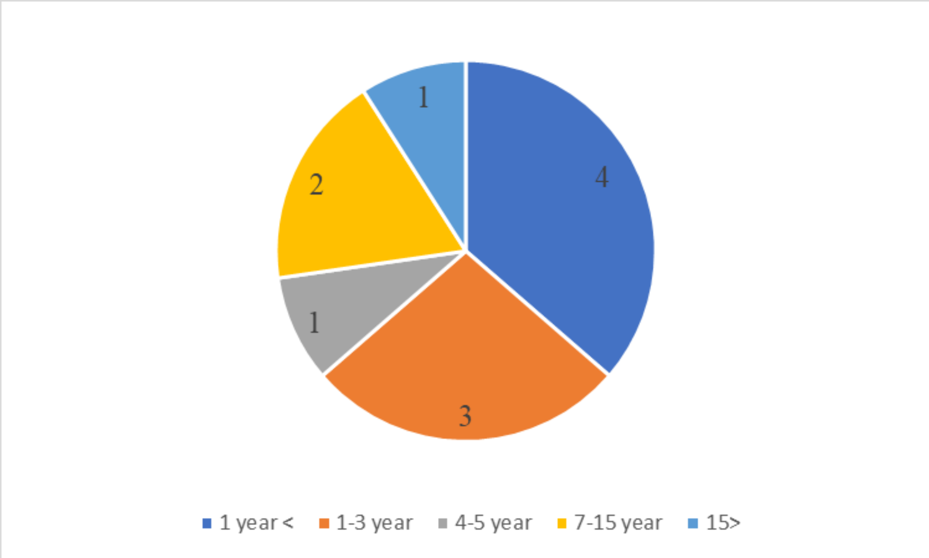


Figure 17. Job Experience of Participants

4.2.1.2. The Investigation of Sheet Metal Forming Design Methodology in Different Programs

In this research we previosly designed a mold design with different programs such as Visicad, Catia, Solidworks and Çizen. In Appendix B there is a mold designed. In the result of the mold design we compared all the prosess. Table 11 shows to us differences in the programs. You can see the detail result in Appendix I After evaluation Table 11, we prepared 11 questions to ask participations to investigate sheet metal forming design methods result of the participants design methodologies is shown table 12 and 13. We faced to participants using Catia and Visicad, not Solidworks.

Table 11. Sheet Metal Forming Design Methodology With Solidworks, Çizen, Visicad and Catia Program

	SOLIDWORKS	ÇİZEN	VISICAD	CATIA
Interface Evaluation	Modules are grouped horizontally on a single toolbar. Switching between modules is easily possible without the need for platform change.	Toolbars belonging to all modules are available on the same platform; easy access to commands can be performed from a single platform.	Modules are grouped horizontally on a single toolbar. Switching between modules is easily possible without the need for platform change.	When compared to other design programs, switching between modules was not considered usable.
Tree Structure	The product tree can be composed of two components: Parts and Assembly. The part component can be divided into bodies, allowing separate processing to be applied to each body. The body function (body logic, as same as other programs) is not useful. No commands, curves, surface grouping can be performed within the parts.	By forming separate parts, it can be made to add to each other and to subtract from each other. Geometries such as the imported curve and surface can be collected within the "Geometry Set" but cannot be grouped.	There is no tree structure. There is a layer system.	The product tree can be composed of various components such as part, product, and component. Its working ability with body logic within the part component makes the design easier. Via the Assemble command, grouping between commands can be performed within the design.

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Table 11. (Cont.)

	SOLIDWORKS	**ÇİZEN**	**VISICAD**	**CATIA**
Sketch Platform	The Sketch module is useful, but the sketch axes cannot be used for commands such as dimensioning or mirroring. (It is needed to draw the axis line into the Sketch.)	The Sketch module is useful, but some functions are missing. (E.g. projection, undo, etc.)	With Sketch, the operations can be performed easily. There is no separate sketch platform. All operations are performed on a single platform.	Sketch module was found useful. Sketch axes can be used effectively for all commands in a sketch, part, and product platform.
Creating Assembly and Sub-Assembly Files	An infinite number of parts and assembly files can be opened into the assembly file. The product tree is complex and has a visuality hard to discern.	An infinite number of parts and assembly files can be opened into the assembly file. There have been people who expressed their positive views on visuality during our presentations.	There is no assembly file. There is a grouping.	An infinite number of parts and assembly files can be opened into the assembly file. The product tree has a regular appearance and structure.

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Table 11. (Cont.)

	SOLIDWORKS	ÇİZEN	VISICAD	CATIA
Creating a Sheet Metal Forming Set	When designing a Sheet Metal Forming, demonstrating the components to be used in more than one place within the Sheet Metal Forming from a reference part makes it possible to greatly shorten the duration of the design. This feature in SolidWorks applies to sketches, curves, etc. However, because plane information, surface information could not be proclaimed, a complete Sheet Metal Forming set could not be created. Once components were added, each sheet metal forming group was designed sequentially and partially independently.	hen designing a Sheet Metal Forming, demonstrating the components to be used in more than one place within the Sheet Metal Forming from a reference part makes it possible to greatly shorten the duration of the design. Since this feature is not available in ÇİZEN, the Sheet Metal Forming set was not formed and the design was continued by creating part files one by one and putting them into the assembly file.	When designing a Sheet Metal Forming, demonstrating the components to be used in more than one place within the Sheet Metal Forming from a reference part makes it possible to greatly shorten the duration of the design. Since VISICAD does not have this feature, the Sheet Metal Forming set was not formed; all parts were drawn in a single platform.	Since the linked and parametric operation is available in the CATIA program, the components can be disseminated in the Sheet Metal Forming. For this reason, a Sheet Metal Forming set could be created in Catia.

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Table 11. (Cont.)

	SOLIDWORKS	ÇİZEN	VISICAD	CATIA
Use of Standard Parts in a Sheet Metal Forming	The parts were connected to the axes in the assembly platform. However, since the linked operation could not be performed, the creation of the component elements one by one, such as adding, milling, and drilling, made the design difficult.	Unable to connect standard parts to axes or points in the assembly platform is a major deficiency.	Processes for each standard part were created one by one.	Standard parts can be connected to references such as axis, line, and plane; and the elements of these parts such as adding, milling can be demonstrated in the Sheet Metal Forming through the publication command.
Moving and Rotating Operations In an Assembly Platform	The move/rotate command used to position parts in the assembly platform was not functional. There have been challenges in positioning the presses and the method file.	In the linking operations used to position parts in the assembly platform, some improvements are needed in terms of options and visibility of the options.	There is no assembly platform. All operations are performed on a single platform. Since the parts are not connected to each other, moving and rotating operations can be done easily.	The move/rotate command used to position parts in the assembly platform was functional. In addition, positioning and movement can be achieved with the caliper element.
Sufficiency and Functionality of the Commands	Commands are generally useful, but insufficient. Surface commands were inadequate.	Commands are generally useful, but insufficient.	Except for surface commands, the commands are useful and sufficient.	Commands are generally useful and sufficient.

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Table 11. (Cont.)

	SOLIDWORKS	ÇİZEN	VISICAD	CATIA
Implementation of Changes	Since no linked operation and parametric design were performed, the implementation of the changes is difficult.	Since no linked operation was performed, the implementation of the changes is impossible.	If the changes are on the surface, they are somewhat difficult to implement, however, results can be obtained quickly for other operations.	Since the linked operation and parametric design could be performed, it is possible to implement changes.
Parametric Operation	There is no parametric operation. (Or could not be used in the design.)	It has superior features regarding parametric design.	There is no parametric operation.	The parametric operation can be performed.
Linked Operation	The linked operation could not be performed.	There is no feature of linked operation.	There is no feature of linked operation.	There is a feature of linked operation.

Table 12. A Sheet Metal Forming Design Method of Catia Users

CATIA USER'S COMMENTS		
QUESTION TITLES	**PILOT PARTICIPANTS**	**PARTICIPANTS**
Tree Structure	• Users did not declare any deficiencies in the product tree structure of the program. • It has an understandable and functional product tree consisting of Part and Product components. • Because it is based on Body logic, it is easy to group. • The sorting of Assembly and Part components in itself, its transport and grouping can be achieved practically.	• It can be seen that the program can perform any process we want in sheet metal forming design. • All groupings can be easily done. • The Program does not allow enough flexible operation, there is one way to perform the desired operation, and it is not easy to do other than that. • In the program, the feature of being able to change the platform for module transitions and having the corresponding commands open, whichever module is operational, is useful. • There is no difficulty in searching for commands, it can be detected easily. • The use of the 'Replace Component' Command is not enough practical. • Link management is not efficient.

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Table 12. (Cont.)

CATIA USER'S COMMENTS		
QUESTION TITLES	**PILOT PARTICIPANTS**	**PARTICIPANTS**
Sheet Metal Forming Set	• All Catia users perform parametric design by creating a Sheet Metal Forming set. The designs made following this method have an impact on the design in the range of 20-50% in the case of any revision.	• All of the users design by forming a Sheet Metal Forming set. With the support of programs belonging to customer companies, the design is carried out with ready-made Sheet Metal Forming sets. • Through Sheet Metal Forming sets, time is saved by using the processes applied standardly in the design stages. • One of the users has come up with the idea that instead of a parametric Sheet Metal Forming set, by building a structure that can be managed with visual objects, it may be created a more useful set than parametric management. With the use of Sheet Metal Forming sets, the design workload decreases.
Parametric and Linked Operation	• It has been concluded that operating in this way has advantages and disadvantages, but its advantages, as well as its disadvantages, are negligible. • It leads to the beginning of the design takes some time. • It was observed that revisions were quickly and easily implemented in the design. • It provides a great convenience in implementing changes reaching standard parts, affecting almost the whole Sheet Metal Forming.	• Through the parametric and linked operation, the design is created with a more regular structure. • The changes are easy to implement. In the application of revisions, it is sometimes experiencing problems in the process of getting links. • In the application of revisions, the probability of the designer to make mistakes decreases. • Setting up parametric and linked operation at the beginning causes a waste of time. Making a design using parameters is a practical method. • When too many links are received, it leads to an increase in data size and a loop in design.

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Table 12. (Cont.)

CATIA USER'S COMMENTS		
QUESTION TITLES	**PILOT PARTICIPANTS**	**PARTICIPANTS**
Assembly Platform	• Standard parts, by using the assembly links, are connected to the points and planes that I have called into the design and created. • The manufacturing parts are positioned by drawing within the part they belong to and divided into parts with the 'Publication' command.	• All Catia users, after opening the standard parts in the program, connect them to plane, sketch, points by using the assembly links. • Manufacturing parts are drawn according to the coordinate axis by referencing the relevant part. • From a different point of view: A user has mentioned that he assembled standard parts by linking them to visual objects he created.
Sheet Metal Forming Design Revision	• All of VisiCad and Catia users have stated that they had no problems in terms of time and applicability in implementing Sheet Metal Forming revisions.	• In cases where the surface and contour are unsuitable, the fact that revising the Sheet Metal Forming takes too long causes time loss. • Problems occur when there are changes in surface and cutting lines. • When unnecessary commands are considered in the design and the designer does not work regularly, there are problems in implementing revisions. In such designs, there are difficulties in the product tree and link management. • Arising from the links giving too many errors, for the big changes made to the die, there is a lot of time loss in error removal.

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Table 12. (Cont.)

CATIA USER'S COMMENTS		
QUESTION TITLES	**PILOT PARTICIPANTS**	**PARTICIPANTS**
Challenges in The Sheet Metal Forming Process	• If any auxiliary program or library is not used for the standard parts, the process of drilling, milling, adding, removing elements of the parts is prolonged. • Special Cam designs extend the process. • As a suggestion, libraries should be used for the elements of standard parts such as drilling, adding, and subtracting. • If a sample Sheet Metal Forming is not available in the design process, it is difficult to form a Sheet Metal Forming set from the beginning every time.	• Many of the designers have been evaluated that surface commands are difficult to use. • Surface improvement and development processes cause a waste of time. It extends the design process. Surface and contour revisions are difficult. The issue of shifting surfaces and tangent faults slows down the design. • Building a structure based on parametric and linked operation has a challenging and process-prolonging effect for the beginning. • Although assembling standard and manufacturing parts to appropriate positions is a facilitator in the following processes, it takes time to create this layout at the beginning. • Forming of connection elements, hydraulic and electrical installations are the processes that take time.
Moving of Sketch, Part, Assembly Platform Elements	• In this context, the program was deemed sufficient by the designers. • The desired elements can be carried easily in all directions by the designers through the caliper element. • Positioning operations can be easily performed.	• All designers find the program sufficient in terms of carrying the elements in the assembly and sketch platform. Through the translate commands, element moving operations can be carried out easily in the assembly and sketch platform. • Apart from the Translate command, the desired elements can easily be moved independently (without restriction) by the formation called "caliper". • Besides caliper and translate commands, moving operations can also be carried out by providing assembly links.

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Table 12. (Cont.)

CATIA USER'S COMMENTS		
QUESTION TITLES	**PILOT PARTICIPANTS**	**PARTICIPANTS**
Commands Used	• Catia users do not consider that there is a missing command. Lacked features can be added with the formation of 'Powercopy'. • It will be useful to have Sheet Metal Forming wizard that can be used effectively for sheet metal forming. • Facilitating solutions should be developed in the formation of the Sheet Metal Forming set, which is an operation that extends the design process.	• Surface commands can be improved for ease of use. • ‘Powercopy’ and Macro support can be provided, especially for steel designs. • The Program should provide solutions for sheet metal forming design. • Applications for simply performed but time-wasting operations, such as drilling, positioning of standard parts, installation forming should be developed and the burden of the designer should be alleviated. • Surface commands should be developed to eliminate problems in development and revision operations, as the flexible operation cannot be available in surface commands.
Import-Export File Transfer	• All designers have found the program sufficient for file transfer.	• The program has been evaluated as sufficient for Catia file transfer by almost all users. • Compared to other programs, it has been considered more advanced. • Surface data processed with other programs give incorrect results when opened in Catia. • The extensions that the program supports are insufficient.

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Table 12. (Cont.)

CATIA USER'S COMMENTS		
QUESTION TITLES	**PILOT PARTICIPANTS**	**PARTICIPANTS**
Popular and Unpopular Features of The Program	**Popular Features:** • Link Management offers parametric design. • Having the 'Capture' command • There is body logic in Part • Ability to use 'Powercopy' • The surface module is rich in terms of variety of commands • There are commands to make Sheet Metal Forming checks **Unpopular Features:** • When the' Click Ok to Terminate ' error is received, it shuts down the program without allowing the Sheet Metal Forming to be saved and causes data loss.	**Popular Features:** • Easy access to commands, Interface design • Providing linked and parametric operation and body logic and 'Powercopy' features icon designs and Intelligible product tree structure Ease of application of the commands • Swift design elements and Functional Assembly commands **Unpopular Features:** • For the Sheet Metal Formings having increased design load, the performance of the program declines. The thickness command cannot be applied to complex surfaces that are split. • It causes a tangent error in adding and subtracting operations, and the appearance of the design is impaired. When solidification is performed using the 'Up To' function in the 'Pad' command, it does not fulfill this feature if there is drilling in the design. • 'Click Ok to Terminate' error occurs, Difficult and not a useful operation commands in the surface module. The Sheet Metal Formings drawn in the previous variations are very difficult to open in the new variation and data is not efficiently used. Links give errors when there is too much link usage. Licensing costs are very high.

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Table 12. (Cont.)

CATIA USER'S COMMENTS		
QUESTION TITLES	**PILOT PARTICIPANTS**	**PARTICIPANTS**
Interface Design	• Command icons are compatible with their functions. • The method of switching between modules was found to be challenging. • It is an additional feature that the interface design is personally organized and customizable. • Compared to other design programs, mouse movements considered compelling.	• More than one positive and negative view regarding the interface has been reported. Most designers find interface design successful. • The icons of some commands are similar, thus leading to confusion. • Since the interface is suitable for customization, it appeals to all users. • Lack of special applications for sheet metal forming design is a significant deficiency. • It is needed to enter more than one data to perform a command. This makes it difficult to implement the command. The designer must see and select only the relevant elements in the next step or option. If a method such as filtering is followed, more practical design is possible.

Table 13. Sheet Metal Forming Design Method Of Visicad Users

VISICAD USER'S COMMENTS		
QUESTION TITLES	**PILOT SCHEME**	**USER**
Tree Structure	• There is no product tree within the VisiCad program. The design is performed based on Layer logic. • It is not a Sketch-based program.	• It has been operating with the layer logic, and since the layers are independent of each other, it is easy to interfere in them separately.
Sheet Metal Forming Set	• A separate Sheet Metal Forming set is created for each Sheet Metal Forming.	• Sheet Metal Forming sets are used through programs provided by customer companies.
Parametric and Linked Operation	• VisiCad does not enable linked and parametric operations. • No components in the program are interlinked with each other, and the components must be moved one by one when any changes to the design are required. This increases the probability of making mistakes and extends the time for revisions to be implemented.	• VisiCad does not enable linked and parametric operations.
Assembly Platform	• There is only one platform in the program. There is no distinction such as Assembly and Part platform. Parts can be positioned practically.	• Since there is no assembly platform in the program, the link cannot be defined. None of the elements are connected to each other. The manufacturing parts are drawn to the position where they need to be drawn and taken into the group to which they belong. After the positions of the standard parts are determined, the parts can be placed from the desired points to the designated points.

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Table 13. (Cont.)

VISICAD USER'S COMMENTS		
QUESTION TITLES	**PILOT SCHEME**	**USER**
Sheet Metal Forming Revision	• It has been experiencing no problems in terms of time and applicability in the implementation of Sheet Metal Forming revisions.	• It has been experiencing no problems in terms of time and applicability in the implementation of Sheet Metal Forming revisions.
Challenges in The Sheet Metal Forming Process	• The Program is insufficient in terms of surface commands. Aside from its insufficiency, it does not have a simple and practical use.	• The most challenging issue in the program is surface operations. When it comes to changing formed surfaces, there is trouble to intervene. It is better to get the surfaces ready from an outside program.
Moving of Sketch, Part, Assembly Platform Elements	• There is no distinction of platforms such as Part and Assembly. All moving operations (parts, surfaces, etc.) could be performed via the ‘Translate’ command in a single platform. The application of the command is based on simple logic.	• There is no distinction of platforms such as Part and Assembly. All moving operations (parts, surfaces, etc.) could be performed via the ‘Translate’ command in a single platform. The program is sufficient in this regard.
Commands Used	• There are only deficiencies in surface commands.	• Surface commands are insufficient when evaluated in terms of being able to perform sheet metal forming design.

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Table 13. (Cont.)

VISICAD USER'S COMMENTS		
QUESTION TITLES	**PILOT SCHEME**	**USER**
Popular and Unpopular Features of The Program	**Popular Features:** • Being able to design without the Sketch platform. • Linked operation is not available. Mouse movement. **Unpopular Features:** • Surface commands.	**Popular Features:** • Easy access to commands. • Interface design. **Unpopular Features:** • For the Sheet Metal Formings having increased design load, the performance of the program declines.
Import-Export File Transfer	• The program is sufficient for File Transfer.	• The program is sufficient for File Transfer.
Interface Design	• The interface of the program has a feature that considers designer convenience. • The fact that the surface, Part and Assembly commands are merged into a single interface is a useful feature. • The visual appearance of the program icons is compatible with its functions.	• Although it was considered to have some deficiencies, the program has been found sufficient.

4.2.2. Eye Tracking, Interview and Observation Results

Eye tracking is a technique whereby an individual’s eye movemets are measured so that the researcher knows both where a person looking at any given tie and sequence in which their eyes are shifting from the location to another (Poole and Ball, 2005). Eye-tracking research has been increasingly used to supplement usability tests in both commercial, academic practice (Ehmke and Wilson, 2007). Eye tracking just is not a complete engineering approach, though it can make a significant contribution to the assessment of usability. A summary of 21 usability studies incorporating eyetracking was presented by Jacob and Karn (Jacob and Karn, 2003).

There are limited studies in literature to use eye tracking and its adventages.

Doğan and et all, the parameters of yacht hull design were examined with eye tracking device and visual evaluations were made. As a result of these evaluations remarkable opints were determined in order to improve the quality of the relationship between the parameters (Doğan and et all, 2018).

Aditya Thakur and Rahul Rai sketch and 3D drawings used in CAD software fort he use of the interface has made a study taking ino account movement. In the study qualitative and quantitative understanding of the various elements that form a gesture based CAD system. These studies mainly focus on just one of the aspect such as algorithm development, ergonomics/ biomechanics or linguistic viewpoint of hand gestures. The number of research studies focusing on the interaction aspect involved in a 3D CAD modeling task performed by the users is less. Some tasks were given to users and record camera. Movements in command were recorded. After that mapping of movements in CAD commands. This was made with 15 users (Thakur, A. Rai, R., 2015).

Mitchell J. Kelley and David W. Rosen had developed an interface that will increase the capabilities of the users and facililate the use of the tools. They developed an argument for reconfigurable CAD interfaces. The spesific purpose of this study is to enable software developers to develop software with the features required by CAD users. Fort his reason a system has been developed and tested more than the interface. We have only explored rCAD for mouse and keyboard access to a 2D CAD API. We look forward to adapting the concept to more input devices and 3D CAD APIs. Some high dimensional user interface devices such as cameras and motion trackers have not yet established widespread usage paradigms. Perhaps rCAD can help enhance usability for novel interface devices (Kelley, M., Rosen, D. 2016).

4.2.2.1. Tasks Completion Time of Pilot Participants

In our study all participants were given a total of 16 tasks to application on the CAD interface. But for pilot participants we did not apply eye traking recording because of cost. We just recorded task competion time and writed observations in the Usability test Notes Form given Appendix G.

The pilot application is designed in both ÇİZEN program and CATIA program designing the model given in the study. The result of tasks are below in Table 14. The average time of task in the CATIA program is 6,12 minutes, in ÇİZEN program is 12,50 minutes. As can be seen the table, there is almost 50% diffirence in design time. This is very big rate. But i mention that, there is a difference between Çizen and Catia program in doing design Task 4, Task 8, Task 12 and 14. And also Task 16 is only for Çizen program. Therefore it could not be implented in the Catia program.

Table 14. Çizen and Catia Program Tasks Completion Time Comparison Table

TASKS	PILOT-P1		PILOT-P2		PILOT-P3		PILOT-P4	
	ÇİZEN	CATIA	ÇİZEN	CATIA	ÇİZEN	CATIA	ÇİZEN	CATIA
TASK 1	0,4	0,26	0,49	0,48	0,4	0,26	1,3	0,25
TASK 2	1,1	0,45	1,4	0,45	0,33	0,45	1,38	1
TASK 3	0,14	0,11	0,05	0,1	0,05	0,11	1,2	0,12
TASK 4	1	0,31	1,2	0,43	0,33	0,31	1,46	0,44
TASK 5	1,1	0,56	2,2	0,42	2,17	0,56	1,37	0,29
TASK 6	0,43	0,26	0,19	0,18	0,47	0,26	0,23	0,56
TASK 7	0,11	0,07	0,15	0,07	0,11	0,07	0,29	0,23
TASK 8	0,13	0,07	0,38	0,09	0,11	0,07	0,55	0,1
TASK 9	1,28	1,5	2,17	1,29	2,4	1,5	2,23	1,04
TASK 10	1	0,37	0,4	0,22	0,36	0,37	1,37	0,38
TASK 11	0,23	0,9	0,03	0,07	0,11	0,9	0,2	0,12
TASK 12	1,55	0,41	2,29	1,49	1,46	0,41	3,17	1,35
TASK 13	0,38	0,25	1	0,28	0,53	0,25	0,13	0,26
TASK 14	1,01	0,2	0,08	0,21	0,08	0,2	1,31	0,2
TASK 15	0,14	0,22	0,18	0,24	0,14	0,22	1,15	0,25
TASK 16	0,1		0,02		1		0,3	
TOTAL TIME	**10,01**	**5,94**	**12,23**	**6,02**	**09,06**	**5,94**	**17,37**	**6,59**

4.2.2.2. Task Completion Time of Participants

I targed to include 10 participant to the research. But during the research i included one more participant. You can see the recording 11 participant in the Figure 18.

Recording 2 is a trial test. So this is not included in the assestment. While using program some participants changed interface menu locations. 7 participant used top menu, 3 participant used right menu, 1 participant both methods (Recording 13). But one of 7 participant was not assesment because of program problem (Recording 14). One participant using both methods also was not included to confuse. Therefore 9 participants were included in the analysis.

Recordings 12 Export Calibration data

Recording	Participant	Timeline	Duration	Date	Gaze samples
Recording2	ekremyazar1	Timeline1	01:18:16.680	22.10.2019 15:40	60%
Recording3	oguzhanmorova	Timeline1	00:40:03.594	23.10.2019 15:04	93%
Recording4	tolgafıcıcı	Timeline1	00:41:46.844	23.10.2019 15:50	78%
Recording5	ramazanon	Timeline1	00:33:16.329	23.10.2019 16:37	89%
Recording6	muratkara	Timeline1	00:29:50.551	23.10.2019 17:16	87%
Recording7	erkanbulan	Timeline1	00:33:36.384	24.10.2019 11:20	82%
Recording9	muhammetemre	Timeline1	00:26:19.406	25.10.2019 10:22	90%
Recording10	FiratSAGBAS	Timeline1	00:35:18.392	25.10.2019 11:24	82%
Recording11	mehmettunali	Timeline1	00:50:15.898	25.10.2019 15:55	65%
Recording12	mustafaguzel	Timeline1	00:36:39.370	25.10.2019 17:37	92%
Recording13	ekremyazar	Timeline1	00:22:34.702	28.10.2019 11:40	88%
Recording14	ibrahimerikgenoglu	Timeline1	00:25:37.915	28.10.2019 13:26	77%

Figure 18. Eye Traking Recording Screenshot

Pilot participant task completion average time is 12,17 minutes, participants task completion average time is 33,92 minutes.

Pilot Participants Task Completion Average Time	Participants Task Completion Average Time
12,17 minutes	33,92 minutes

In the below, there are task results figures called gazeplot and heatmap. Gaze; an eye tracking metric, usaaly the sum of fixation durations within a prescribed area. Also called “dwell1, “fixation cluster”, or “fication cycle”. In gazeplot there are saccades.

Heatmap is eye-tracking data visualization methods. It represents the values of a variable as colors, where the amount of “heat” is proportional to the level of the represented variable (Bojko, 2013). The main measurements used in eye tracking research are fixations and “saccades”, which are quick eye movements occcuring between fixations. More overall fixations less efficients search. More fixations on a particular area indicate that it is more noticeable, or more important, to the viewer than other areas.

More saccades indicate more searching. Larger saccades indicate more meaningful cues, as attention is drawn from a distance (Poole and Ball, 2005).

Between Figure 19 and Figure 32 show us top menu users. Between Figure 33 and Figure 46 show us right menu users. The colours in the gazeplot indicates to participants. Each participant has a colour.

In this eye tracking research we generally should take into account tap and right menu using. Because we want to see into how long participants find the commands and where they looking the commands. Focuses in the center of the program screen just indicate to us how participants design the tasks.

I also want to inform you some issues about the task. As we metioned before, there are 16 task. But while analyzing, some is divided into task. For example; task 1 includes 4 tasks, also task 4 includes 4 tasks, task 5, 12, 13 and 15 includes 2 task. In this case i analyzed not 16 tasks but 23 tasks.

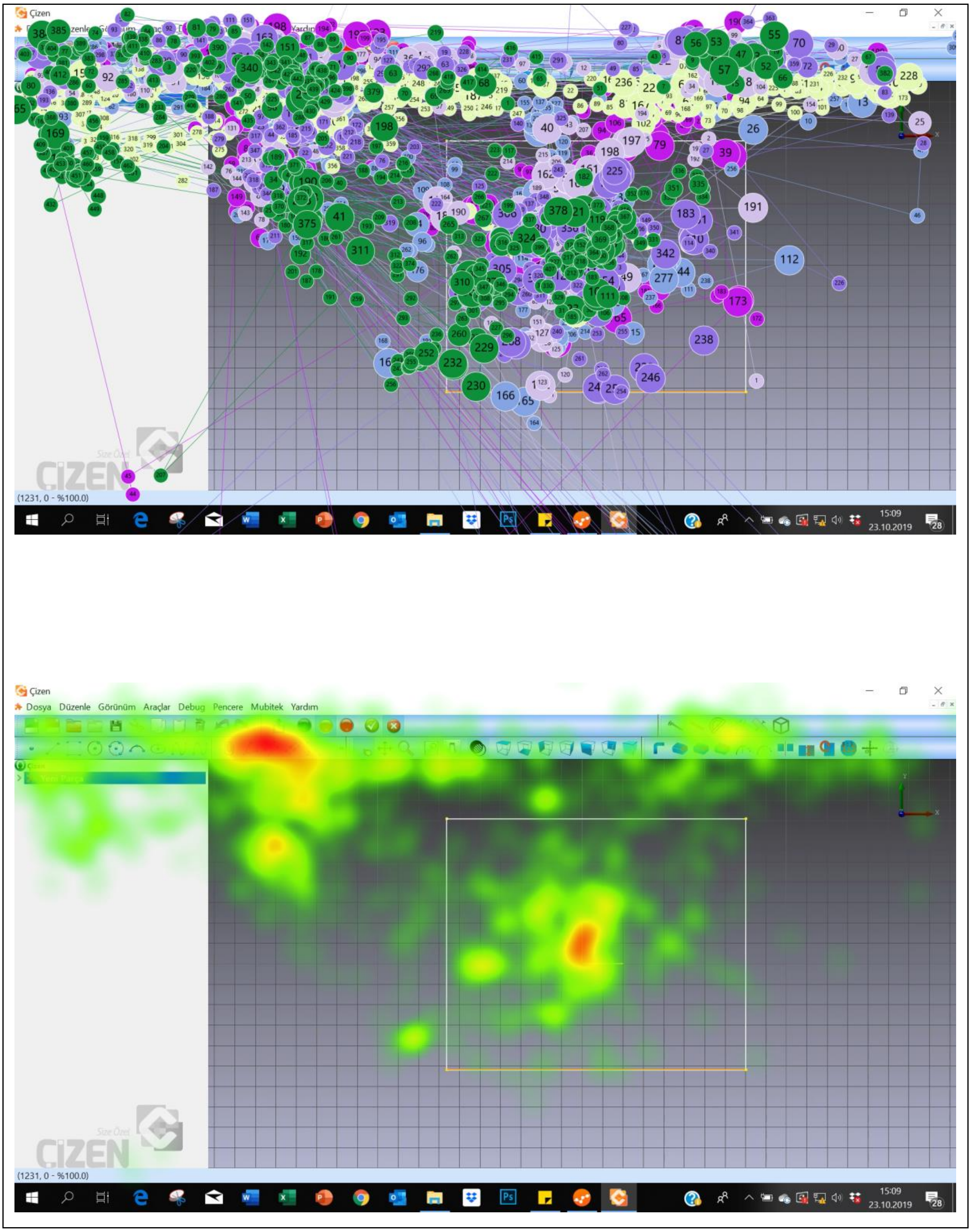

Figure 19. Task 1 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 19, Task 1 has so much saccades and fixations. This indicates that participants have difficulty performing the task. For example most of participants looked for in different tools and looked at different places. The task was completed in an average of 4,68 minutes.

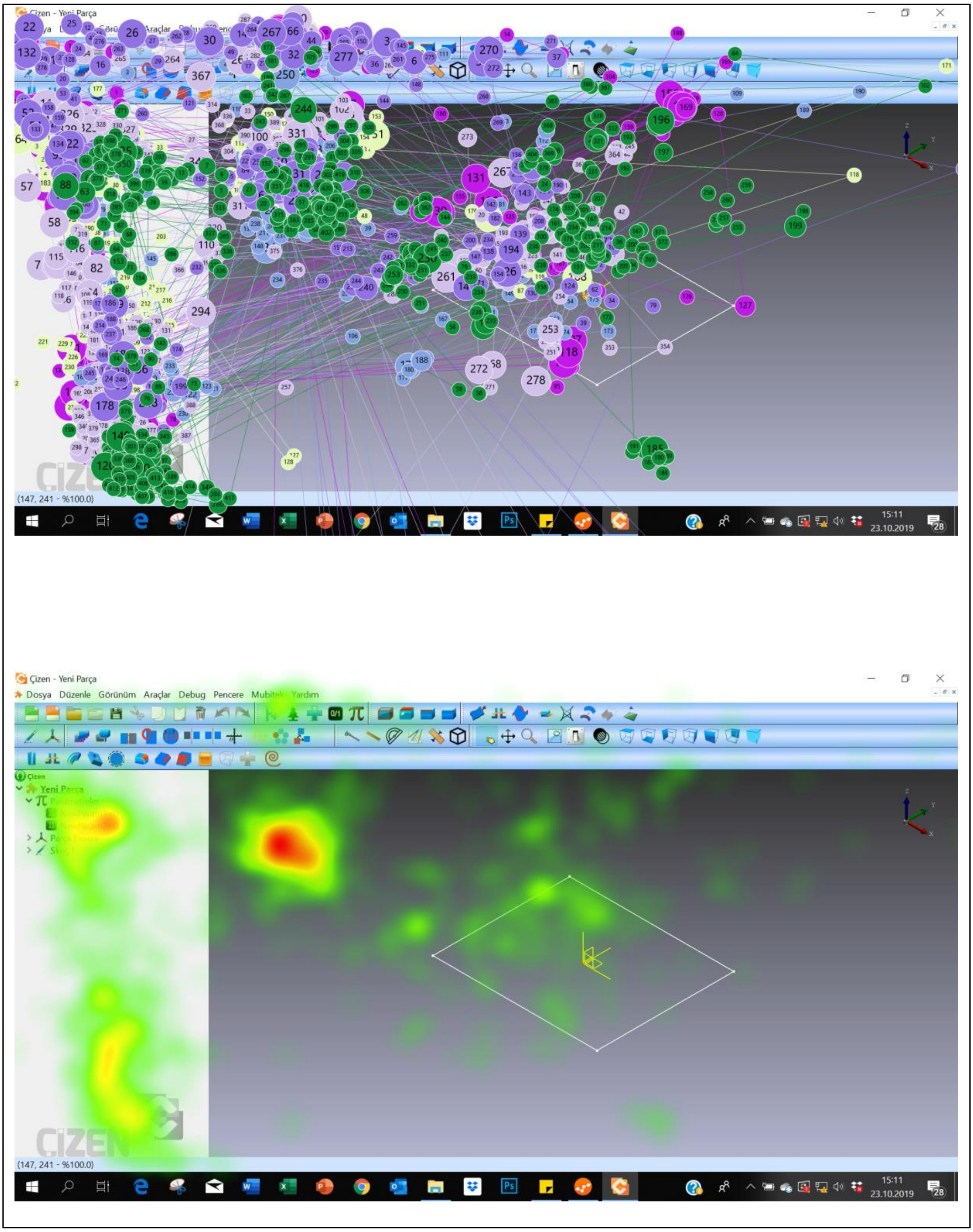

Figure 20. Task 2 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 20, Task 2 has so much saccades but normal fixations. This indicates that participants difficultly found the commands, icons but easily did the task. For example participants found the command in a short time. And they usually looked at the screen in where it can be given dimension and changed the information. The task was completed in an average of 1,25 minutes.

Figure 21. Task 3 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 21, Task 3 has so much saccades but normal fixations. This indicates that participants difficultly found the commands, icons but easly did the task. For example participants found the command in a short time. And they usually looked at the screen in where it can be given dimension and changed the information. The task was completed in an average of 1,79 minutes.

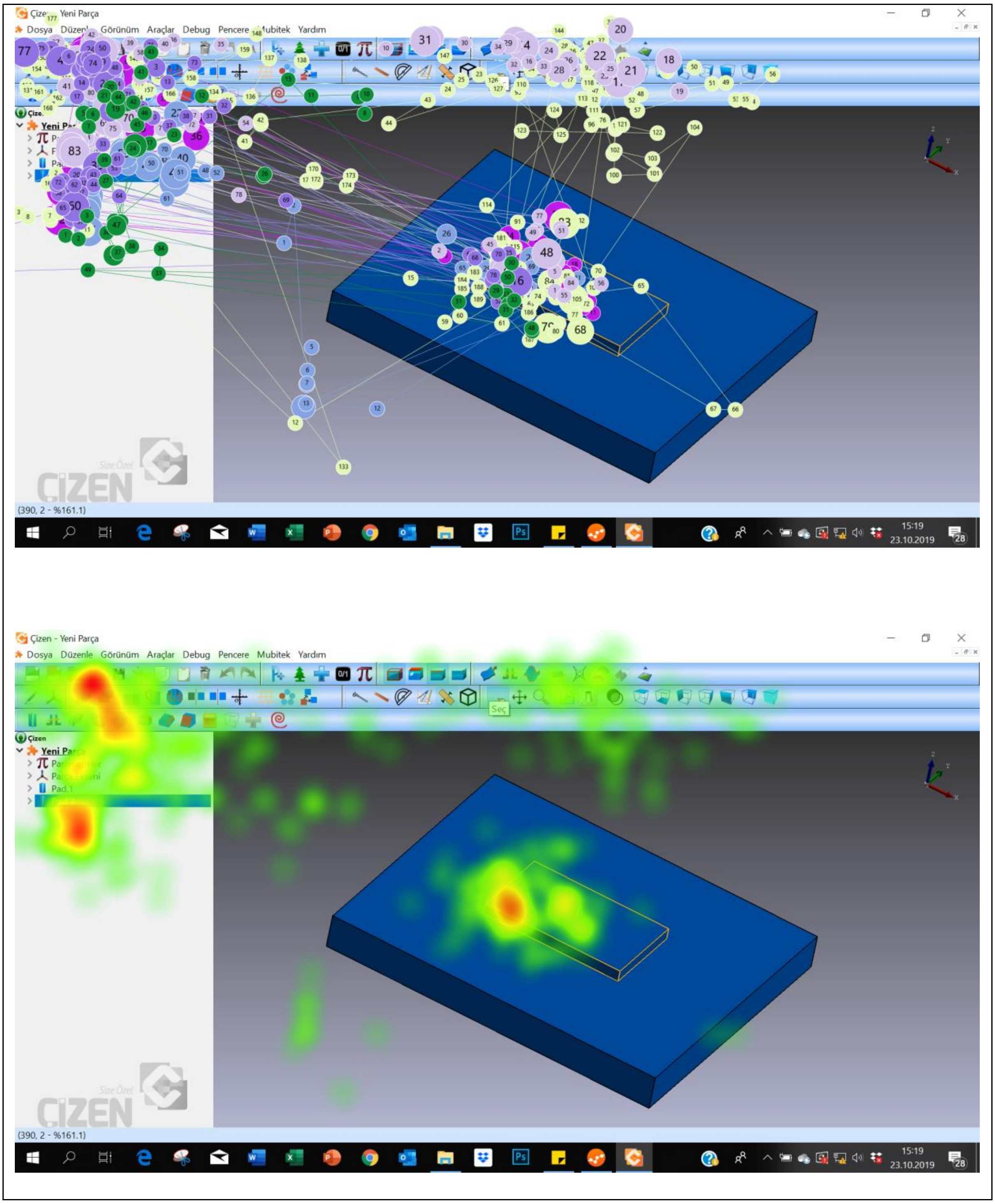

Figure 22. Task 4 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 22, Task 4 has so much saccades and fixations. This indicates that participants have difficulty performing the task. For example most of participants looked for in different tools and looked at different places. The task was completed in an average of 2,18 minutes.

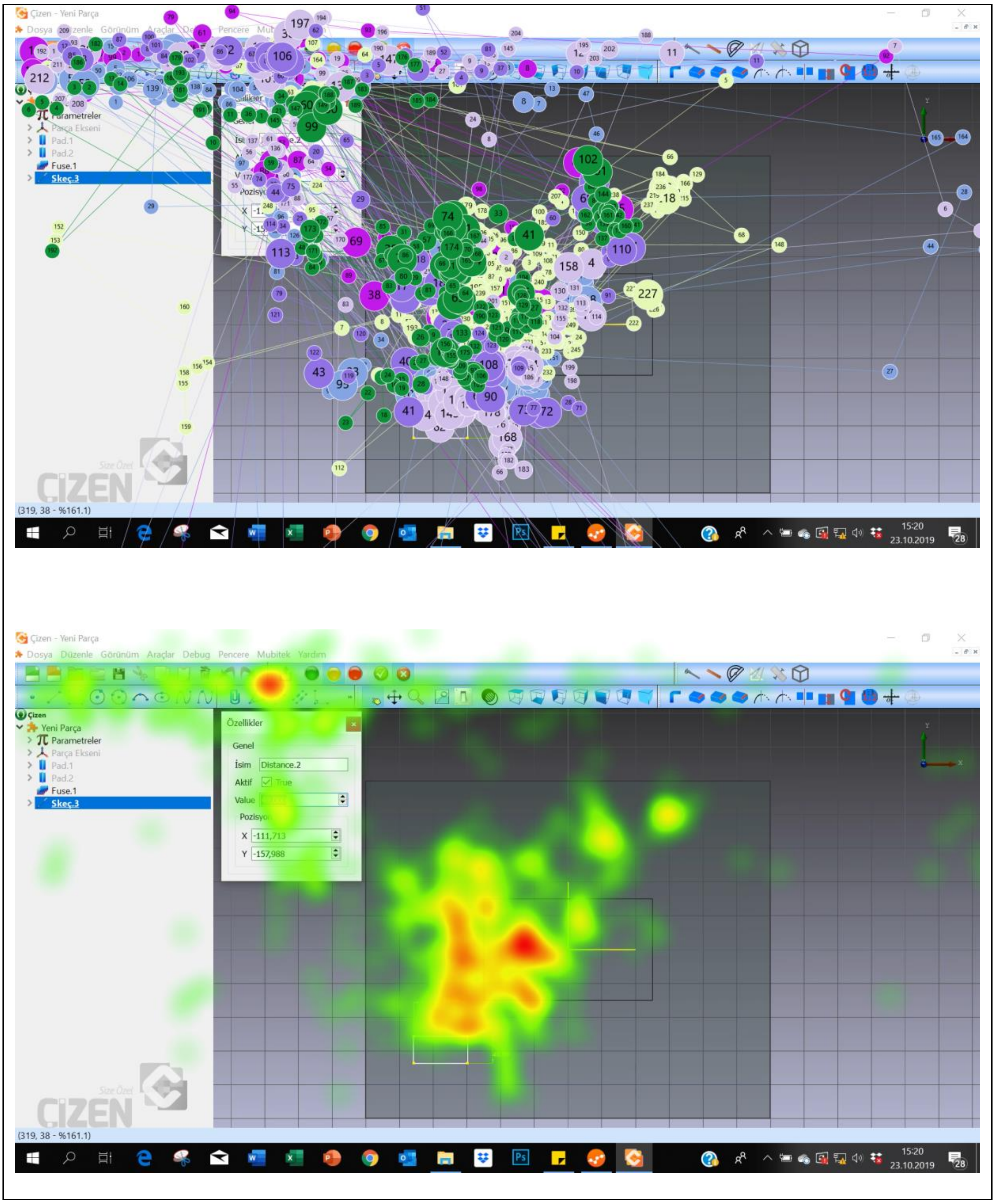

Figure 23. Task 5 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 23, Task 5 has so much saccades and fixations. This indicates that participants have difficulty performing the task. For example most of participants looked for in different tools and looked at different places. The task was completed in an average of 2,31 minutes.

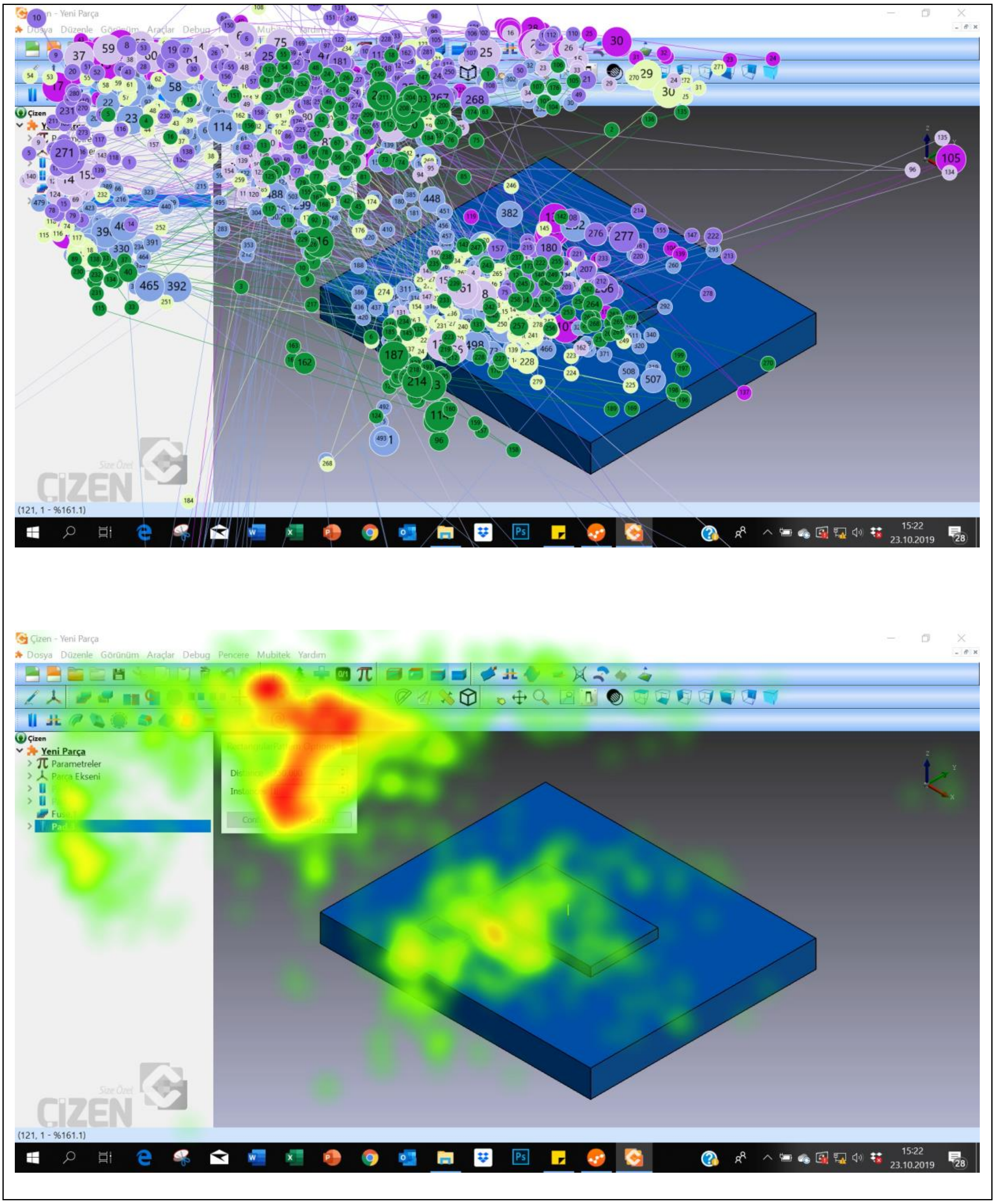

Figure 24. Task 6 Gazeplot and Heat Map (Top Menu Users)

As can be seen from Figure 24, Task 6 has so much saccades and fixations. Fixations were especially on dimension change measurement screen. This indicates that participants have difficulty performing the task and especially difficult to measure. The task was completed in an average of 2,23 minutes.

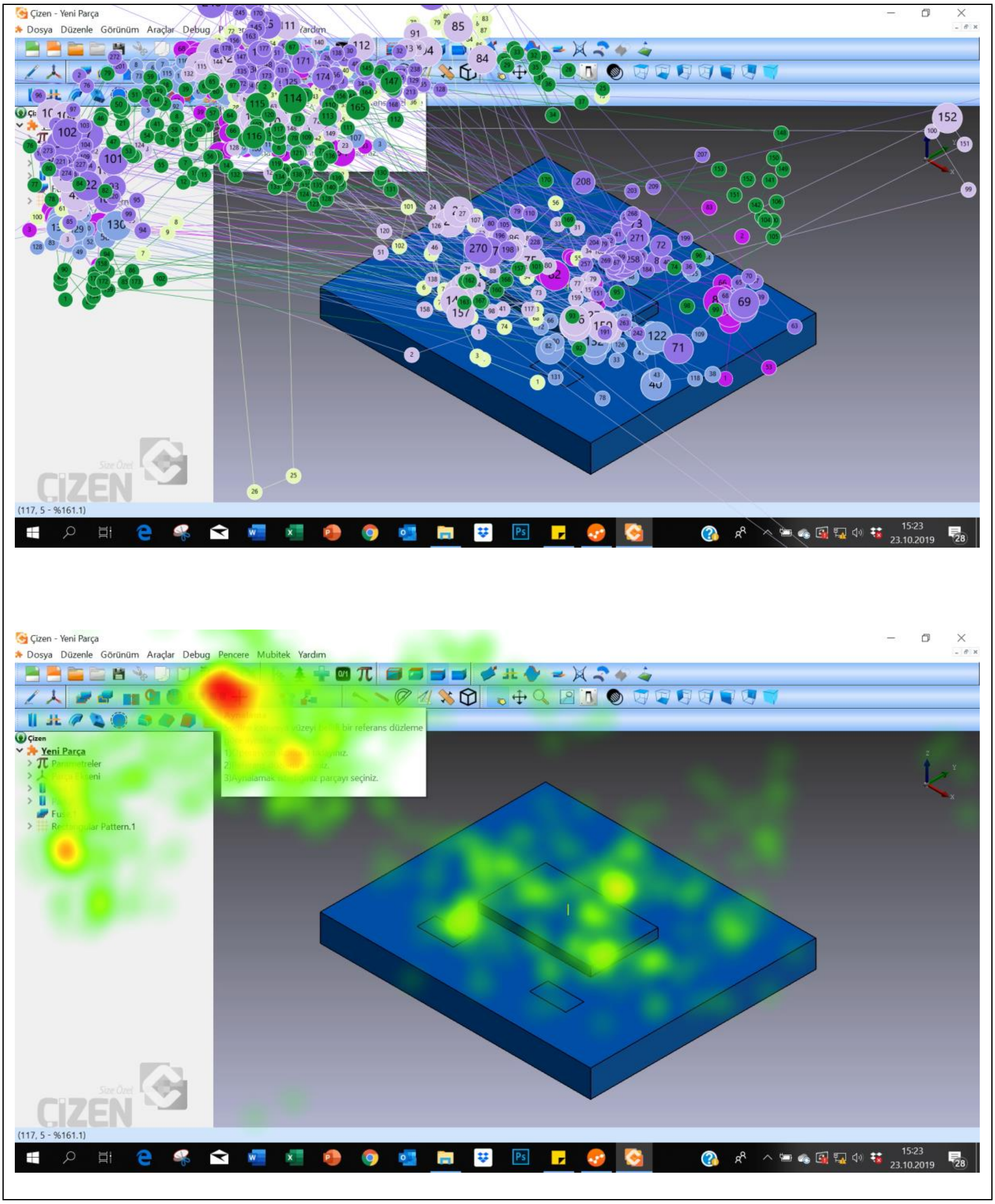

Figure 25. Task 7 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 25, Task 7 has so much saccades and fixations. Fixations were especially on dimension change measurement screen. This indicates that participants have difficulty performing the task and especially difficult to measure. The task was completed in an average of 1,20 minutes.

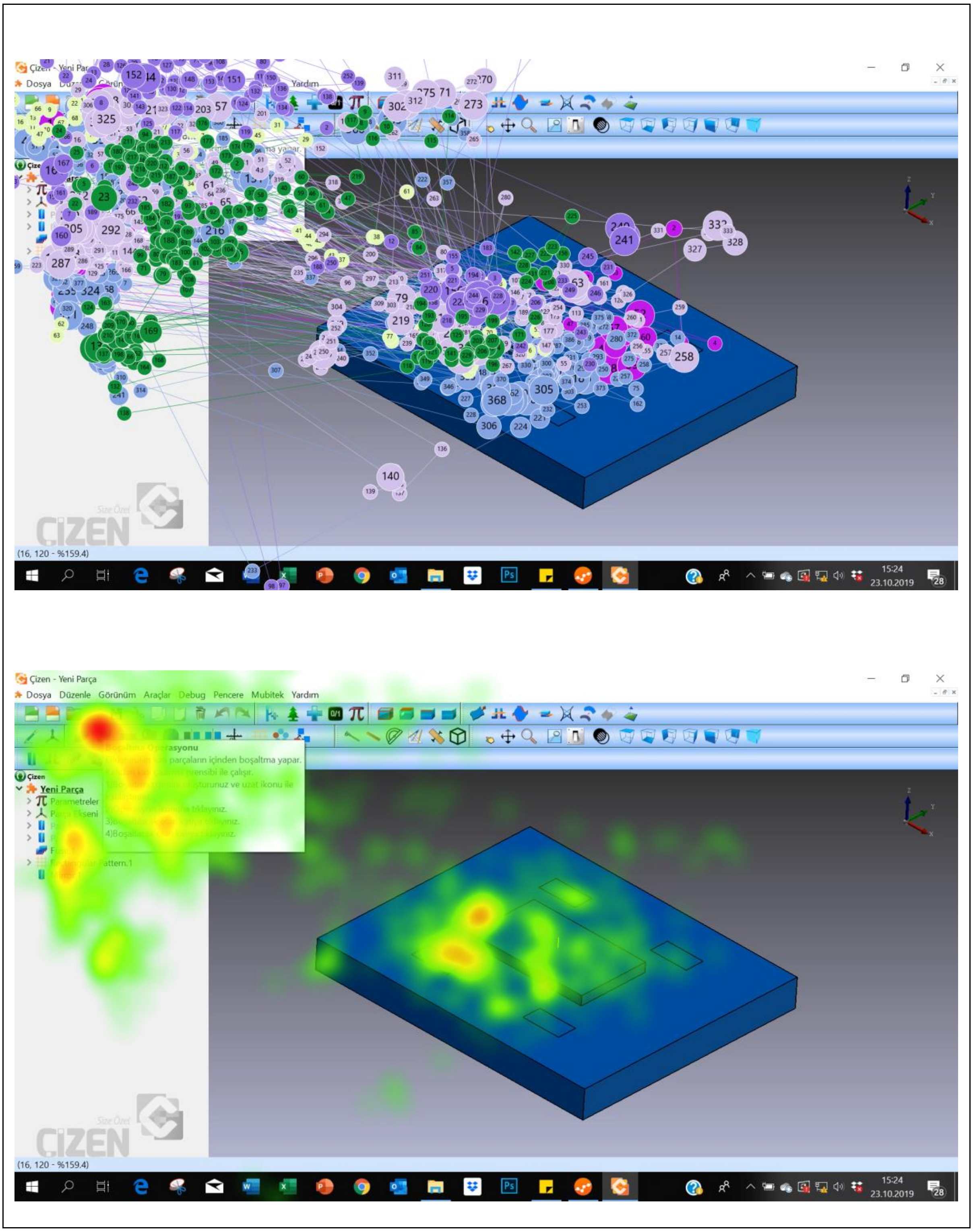

Figure 26. Task 8 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 26, Task 8 has so much saccades and fixations. This indicates that participants have difficulty performing the task. For example most of participants looked for in different tools and looked at different places. The task was completed in an average of 3,30 minutes.

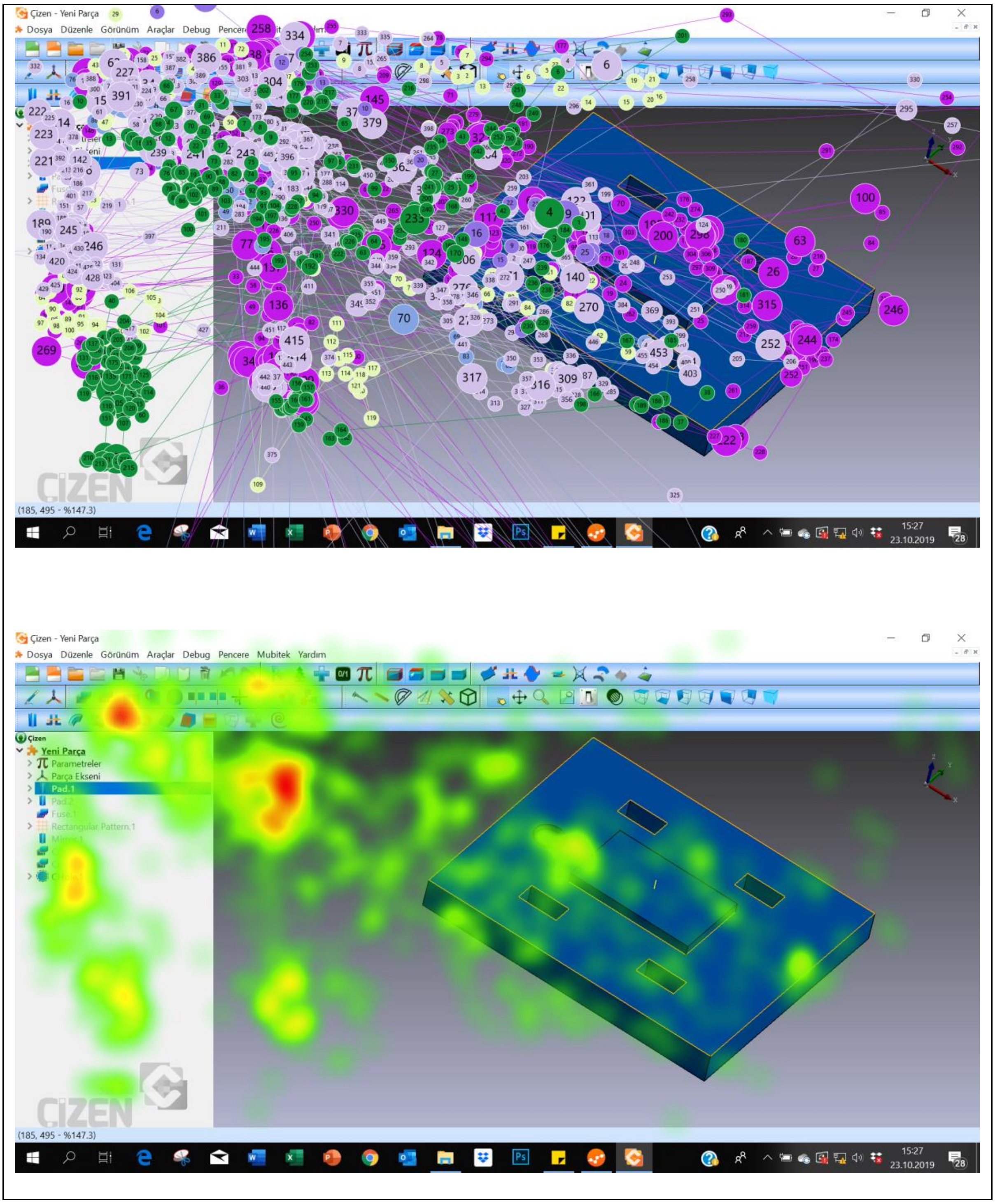

Figure 27. Task 9 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 27, Task 9 has so much saccades and fixations. Fixations were especially on dimension change measurement screen. This indicates that participants have difficulty performing the task and especially difficult to measure. The task was completed in an average of 2,19 minutes.

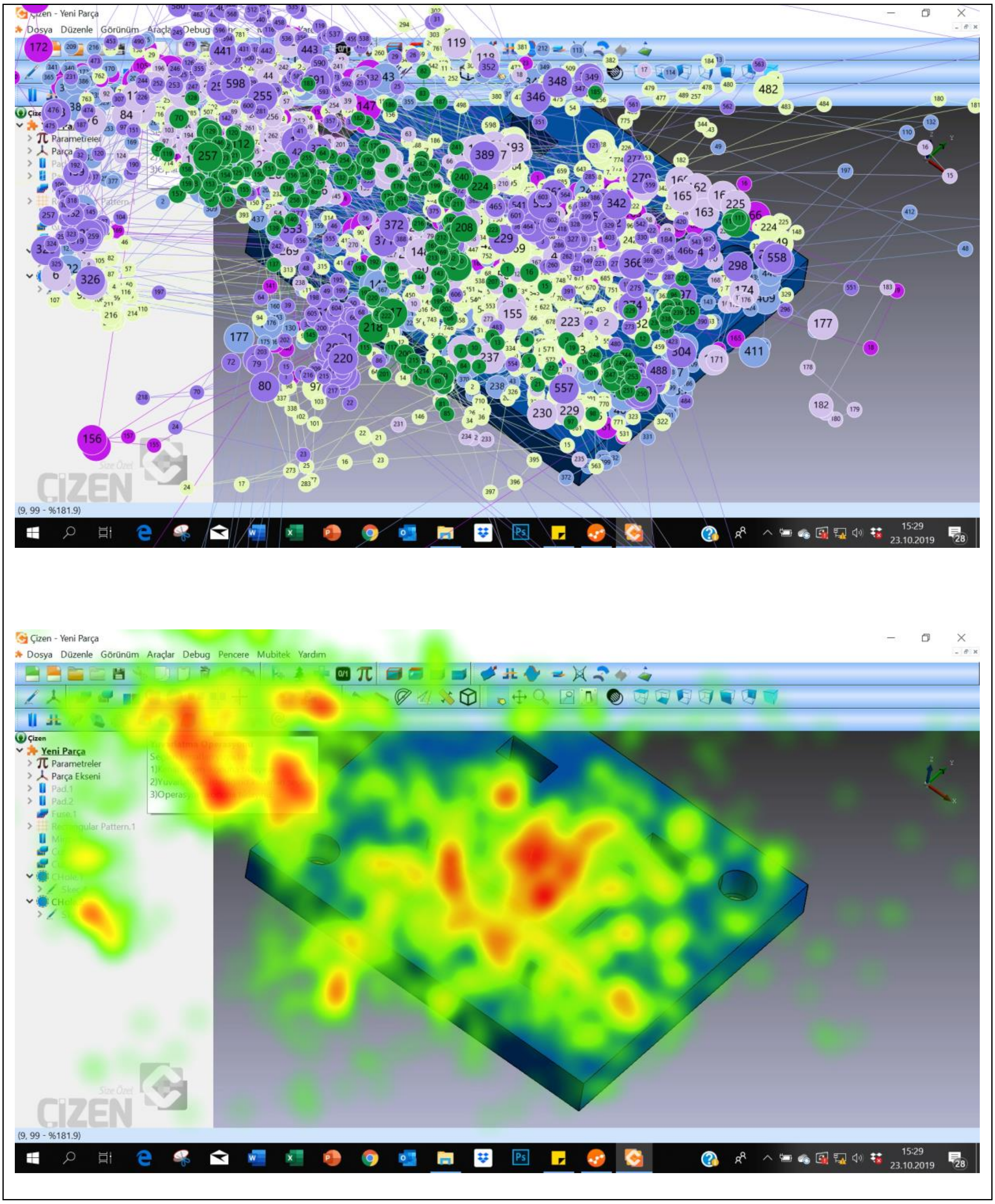

Figure 28. Task 10 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 28, Task 10 has so much saccades and fixations. This indicates that participants have difficulty performing the task. For example most of participants looked for in different tools and looked at different places and also fixed dimension measurement screen. The task was completed in an average of 2,11 minutes.

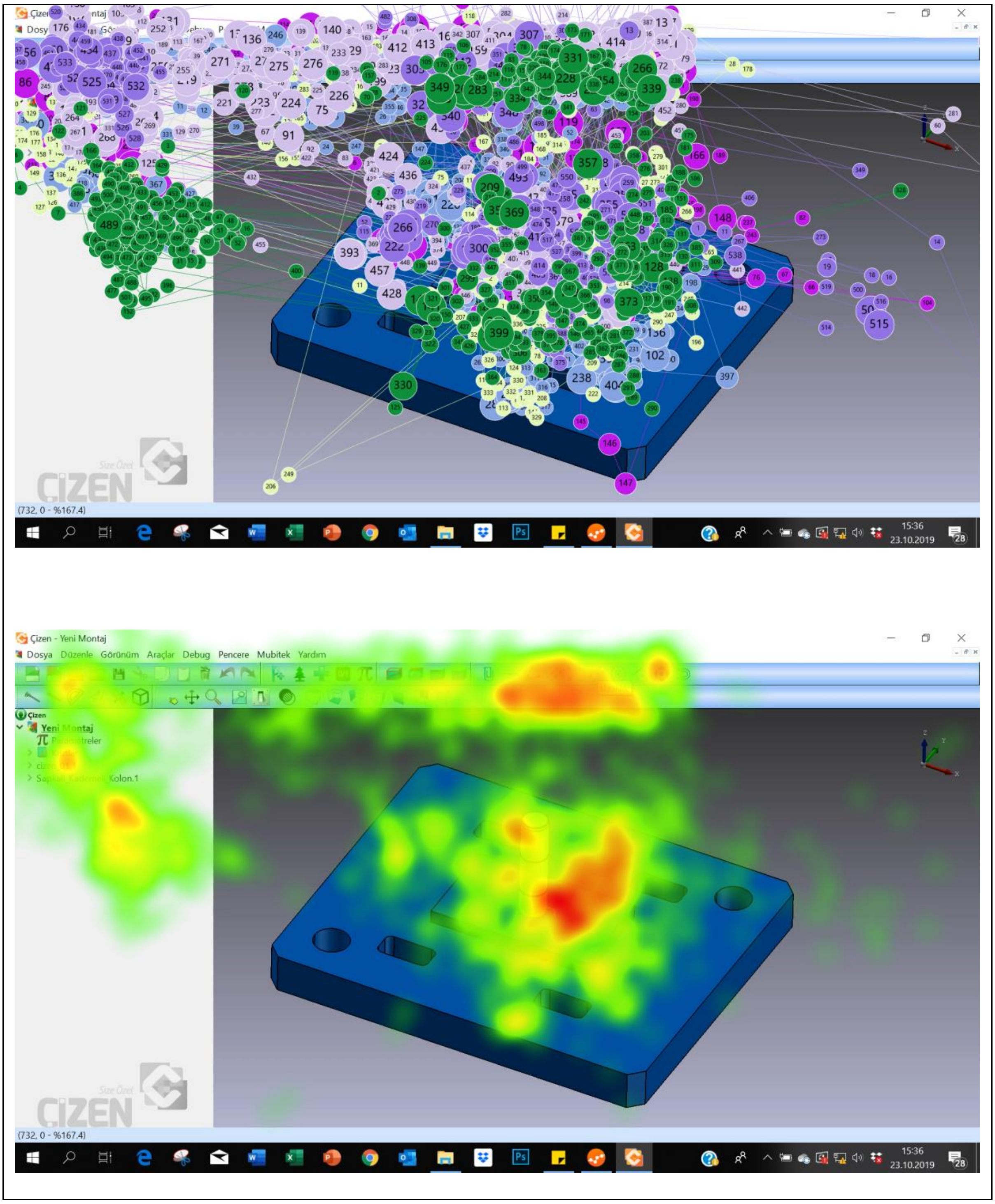

Figure 29. Task 12 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 29, Task 12 has so much saccades and fixations. This indicates that participants have difficulty performing the task. For example most of participants looked for in different tools and looked at different places and also fixed product design in which there was standart part. The task was completed in an average of 5,72 minutes. This is the longest task completed.

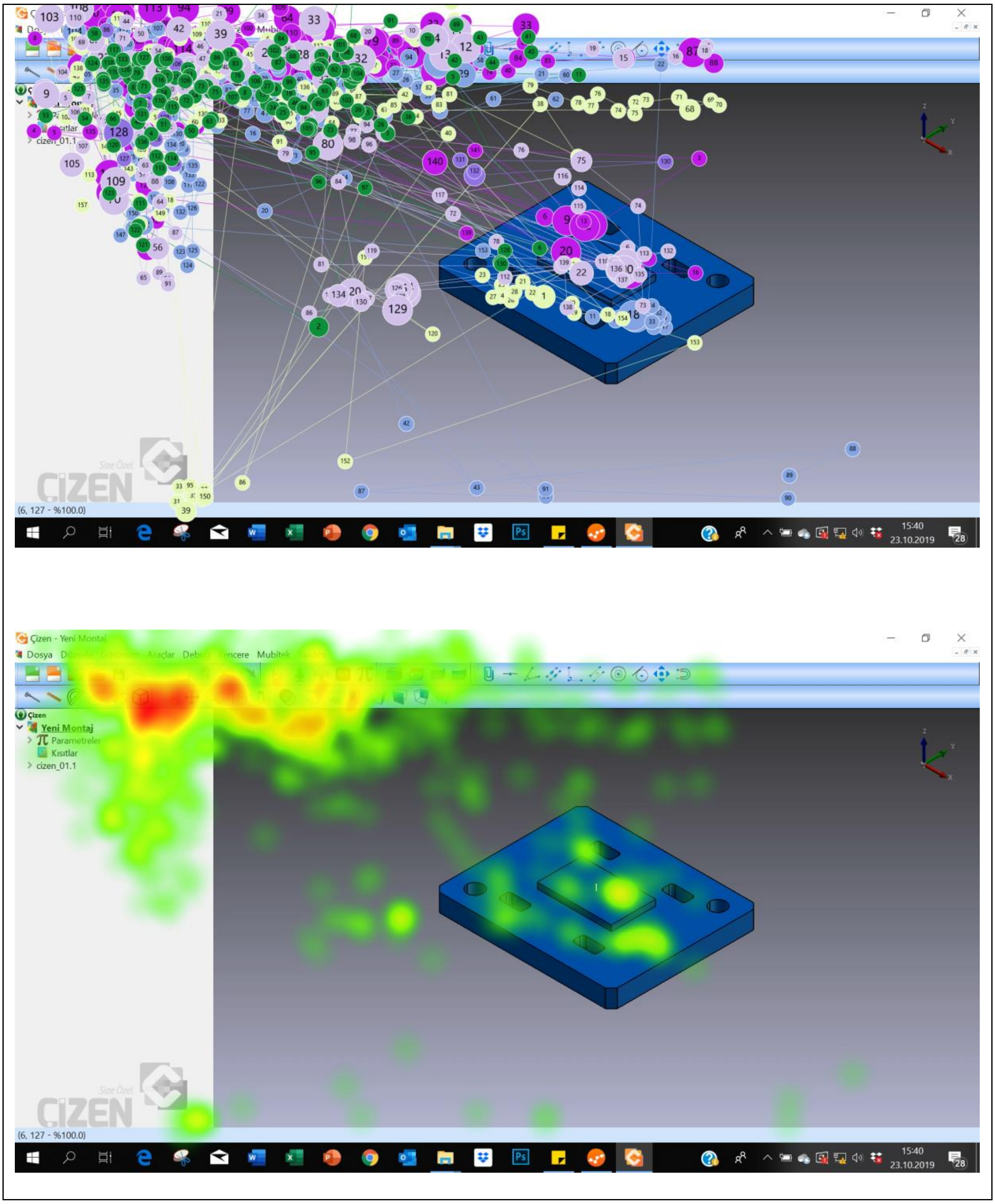

Figure 30. Task 13 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 30, Task 13 has much saccades and fixations. This indicates that participants have difficulty performing the task. The task was completed in an average of 1,27 minutes. In this task, there is remarkable point. One participant fixed different places on the screen. It is necesarry to understand why the participant fixed on insignificant points.

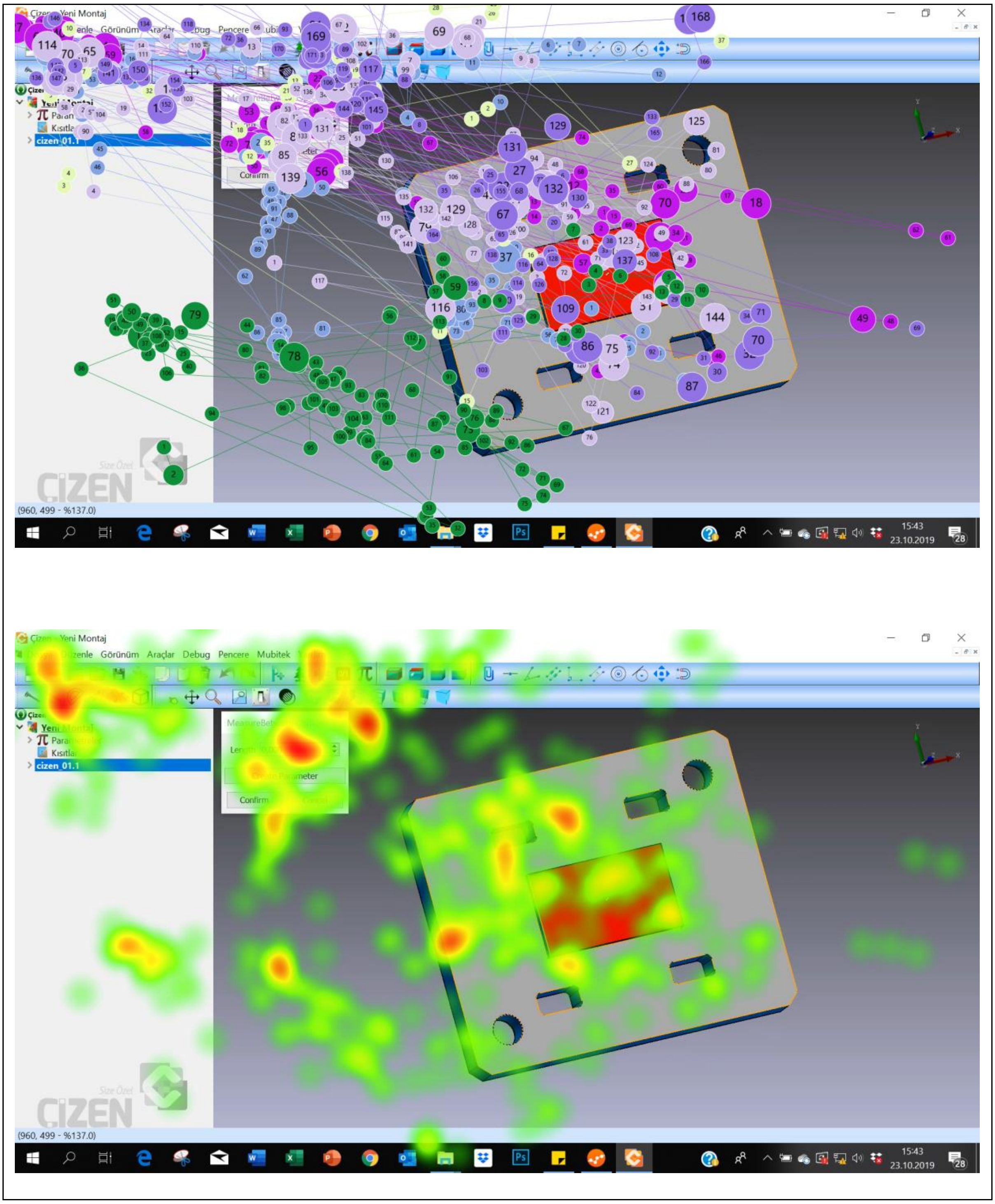

Figure 31. Task 14 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 31, Task 14 has much saccades and fixations. This indicates that participants have difficulty performing the task. The task was completed in an average of 1,73 minutes.

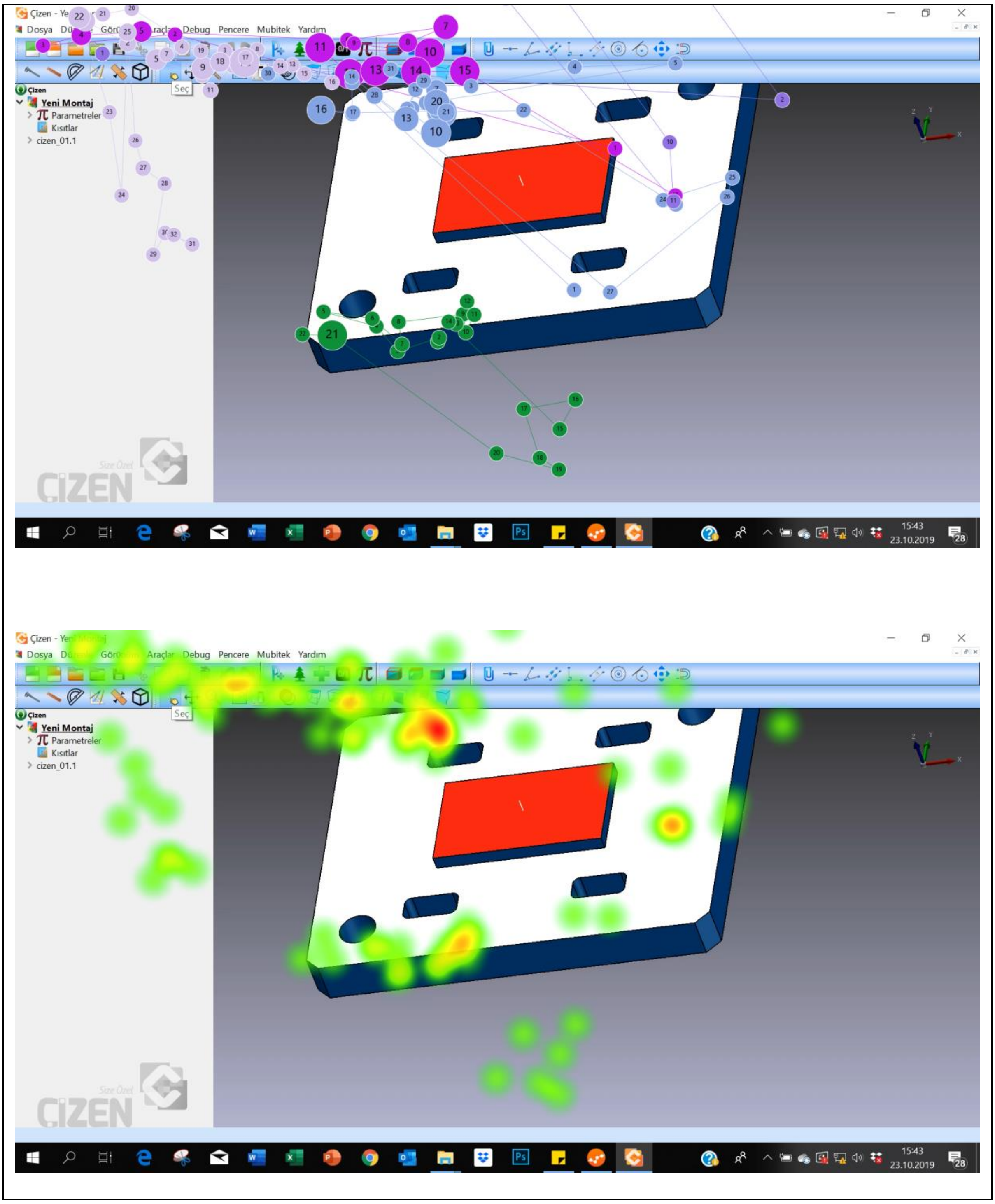

Figure 32. Task 15 Gazeplot and Heatmap (Top Menu Users)

As can be seen from Figure 32, Task 15 has expected saccades and fixations. This indicates that participants have successfully performing the task. The task was completed in an average of 1,08 minutes.

Right menu users and top menu users have similar results for gazeplots and heatmaps. All participants forced to perform similar tasks. There is no significant difference between the tap menu and the right menu users in the tasks.

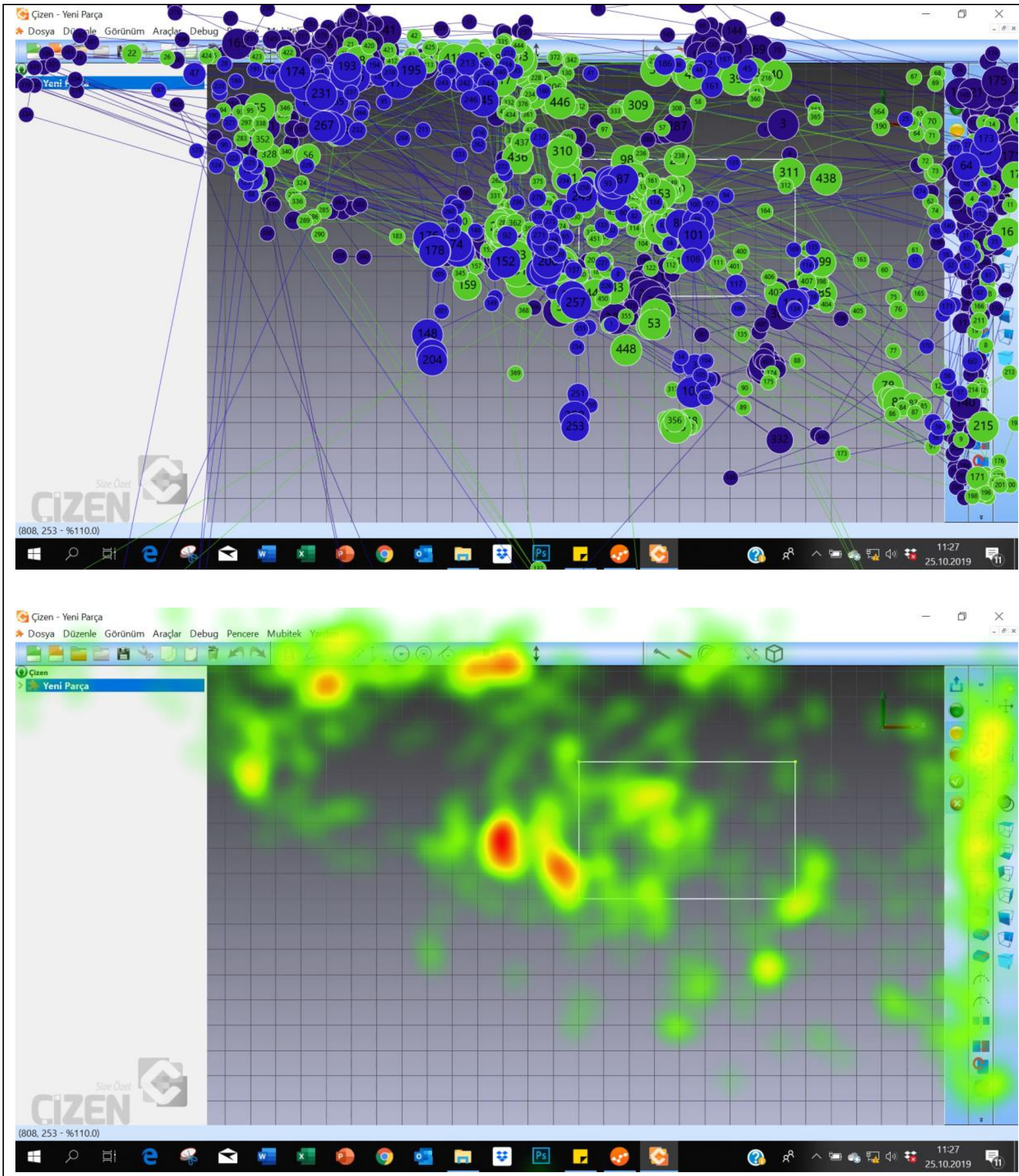

Figure 33. Task 1 Gazeplot and Heatmap (Right Menu Users)

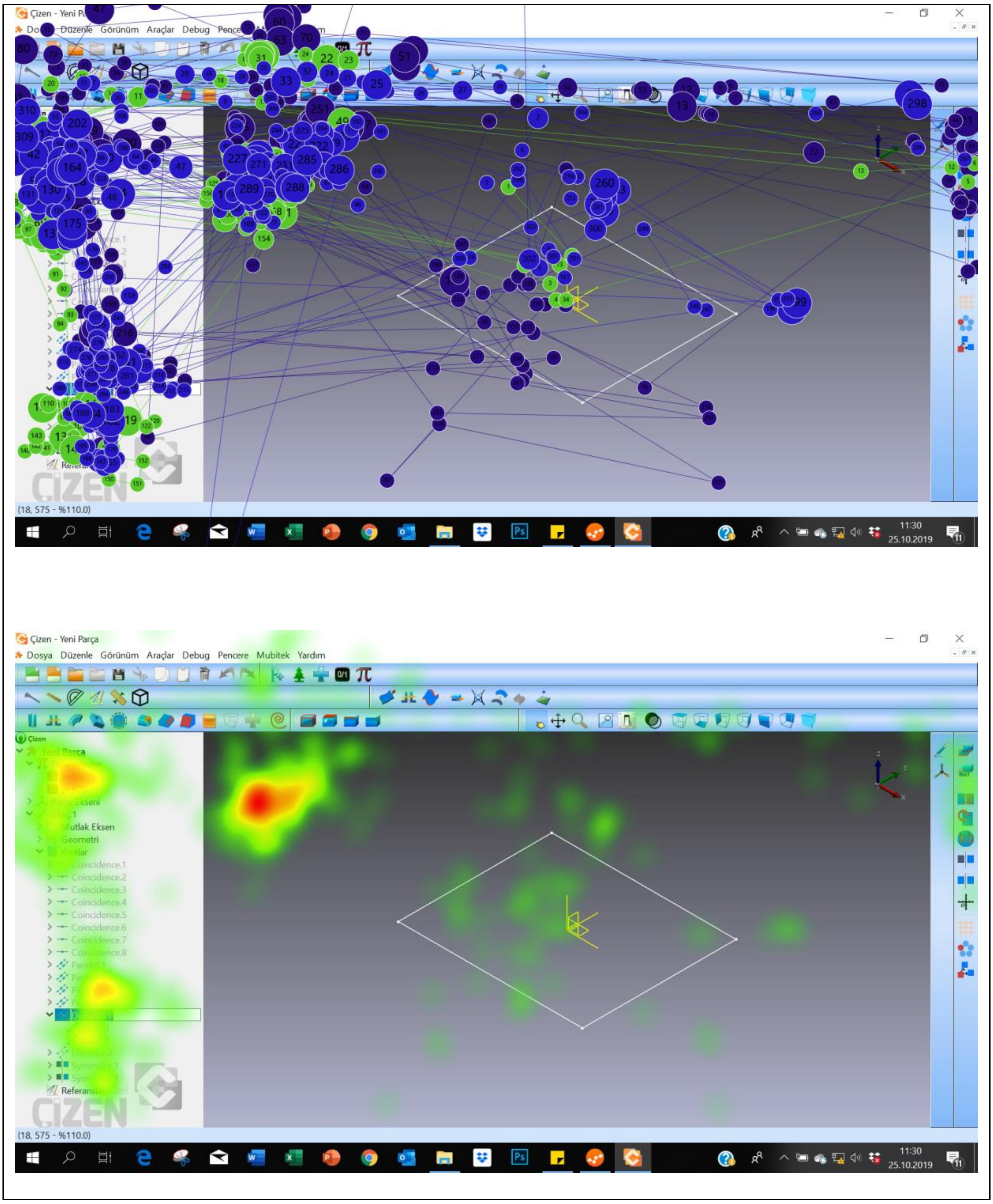

Figure 34. Task 2 Gazeplot and Heatmap (Right Menu Users)

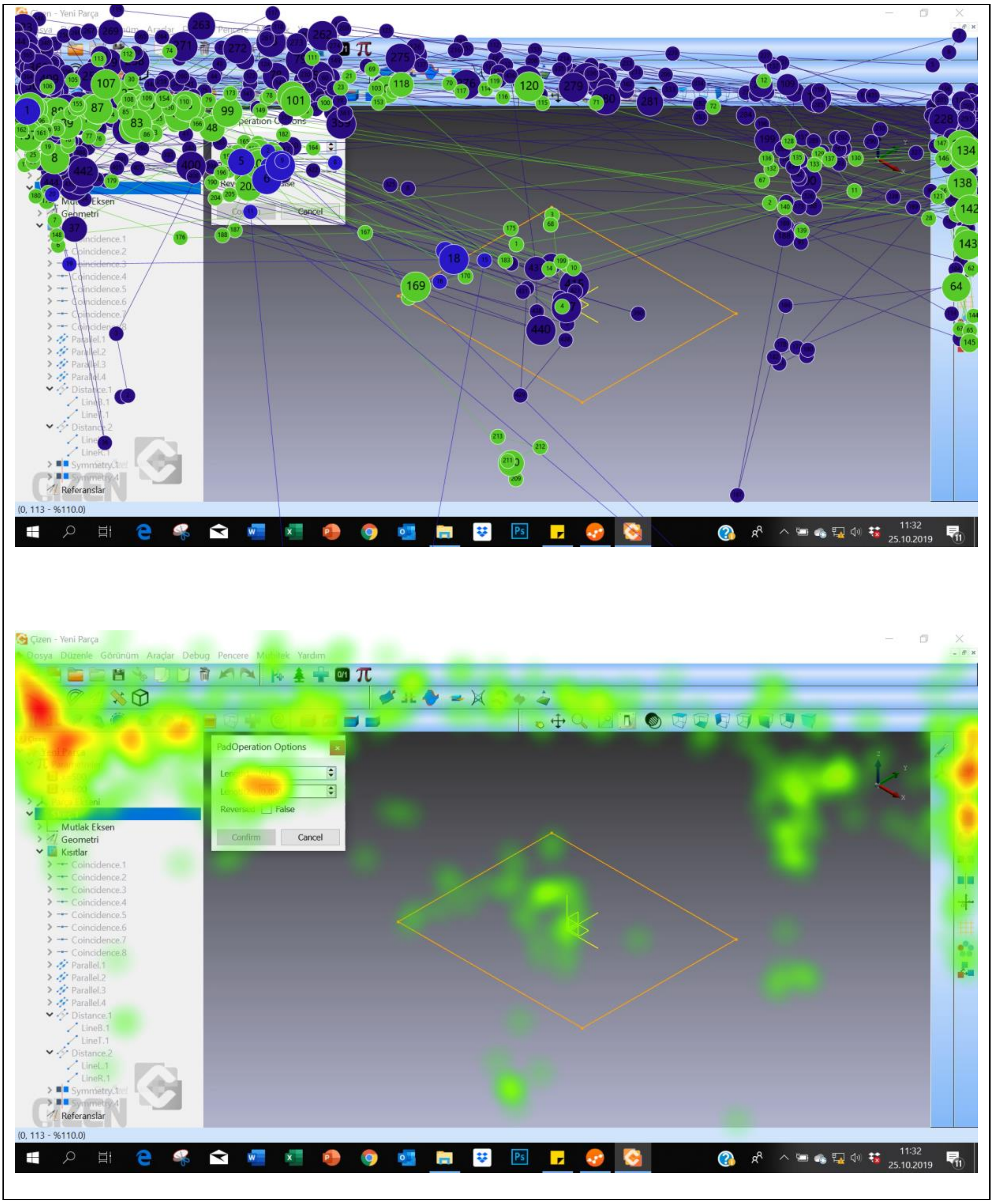

Figure 35. Task 3 Gazeplot and Heatmap (Right Menu Users)

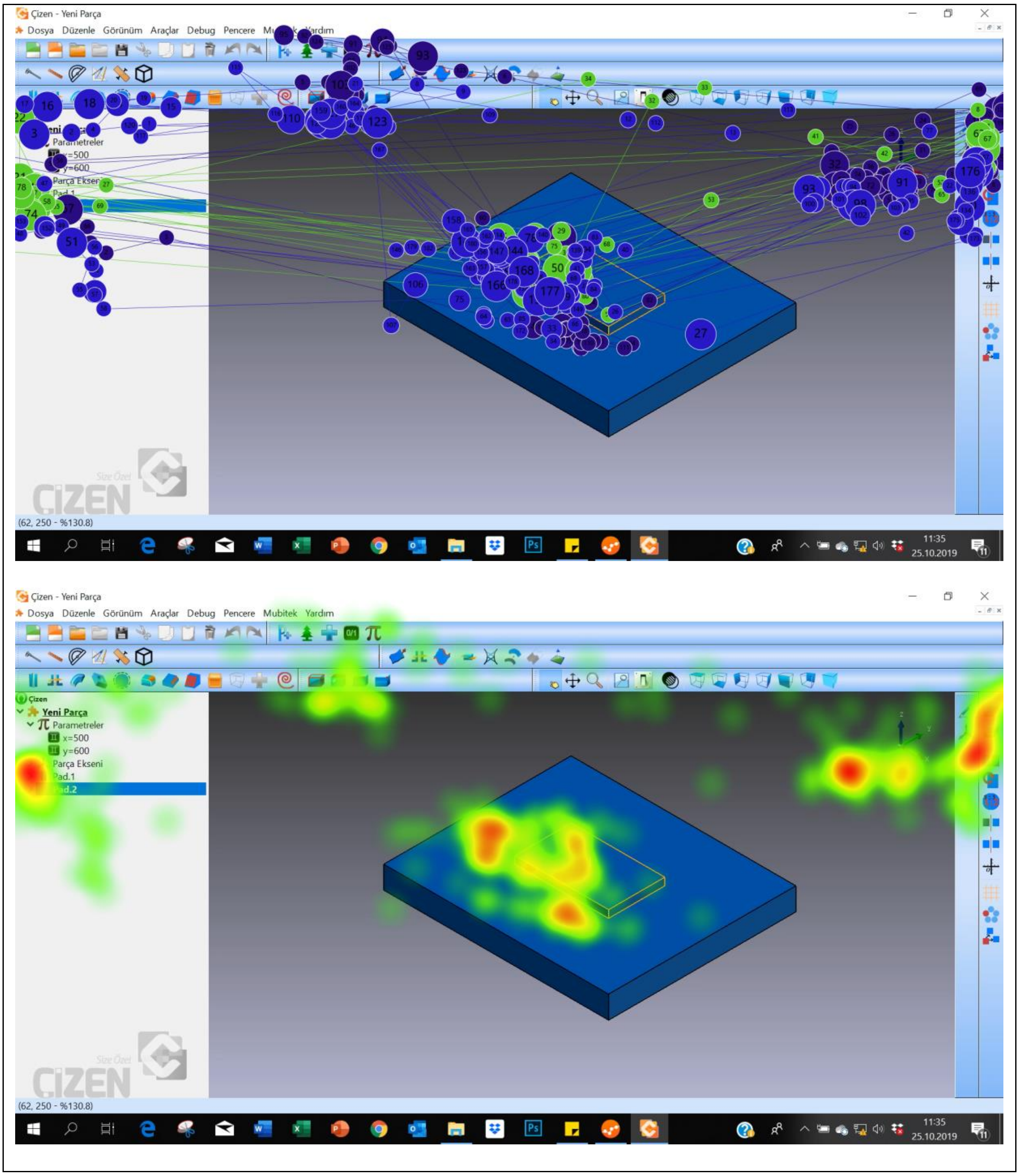

Figure 36. Task 4 Gazeplot and Heatmap (Right Menu Users)

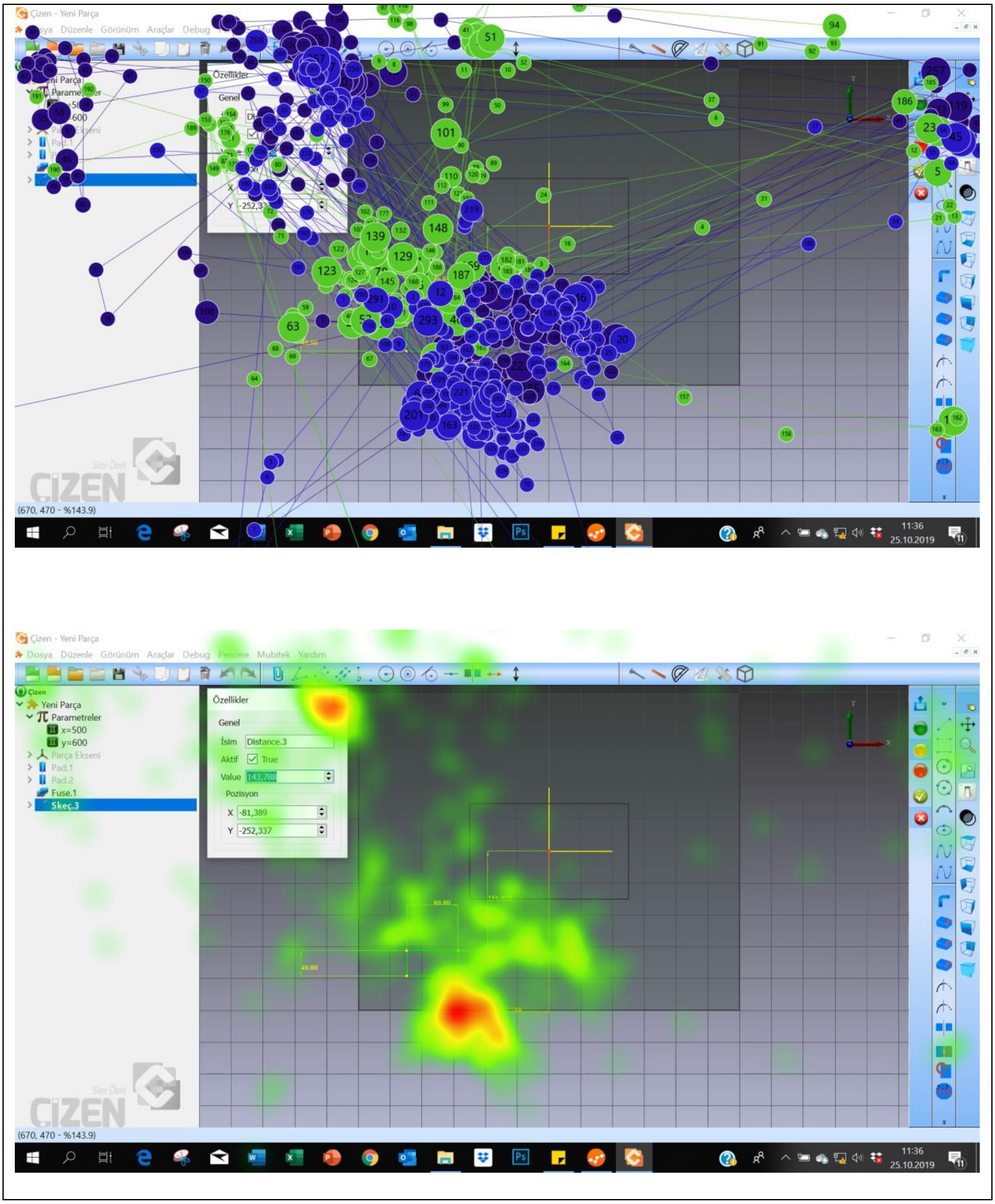

Figure 37. Task 5 Gazeplot and Heatmap (Right Menu Users)

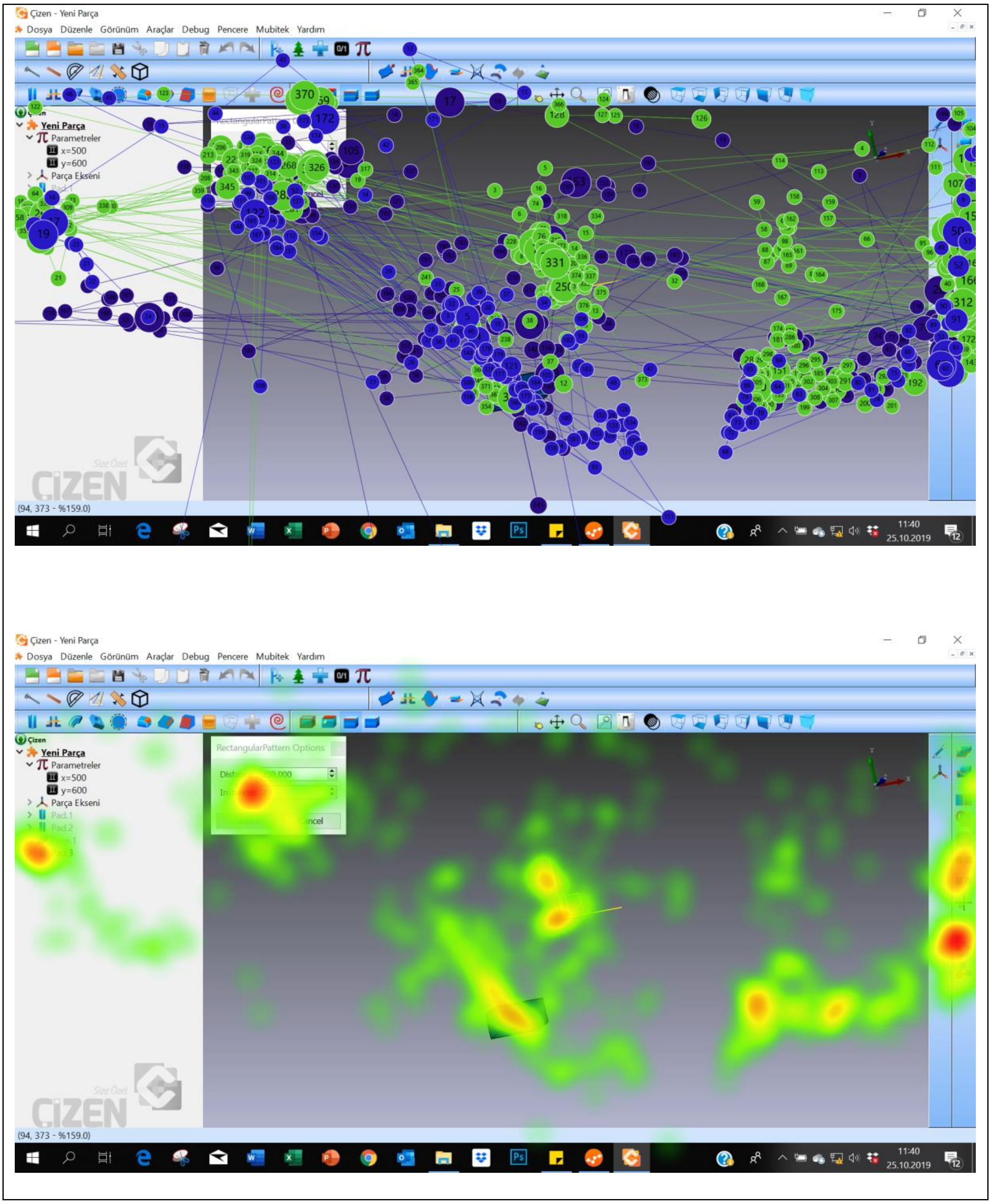

Figure 38. Task 6 Gazeplot and Heatmap (Right Menu Users)

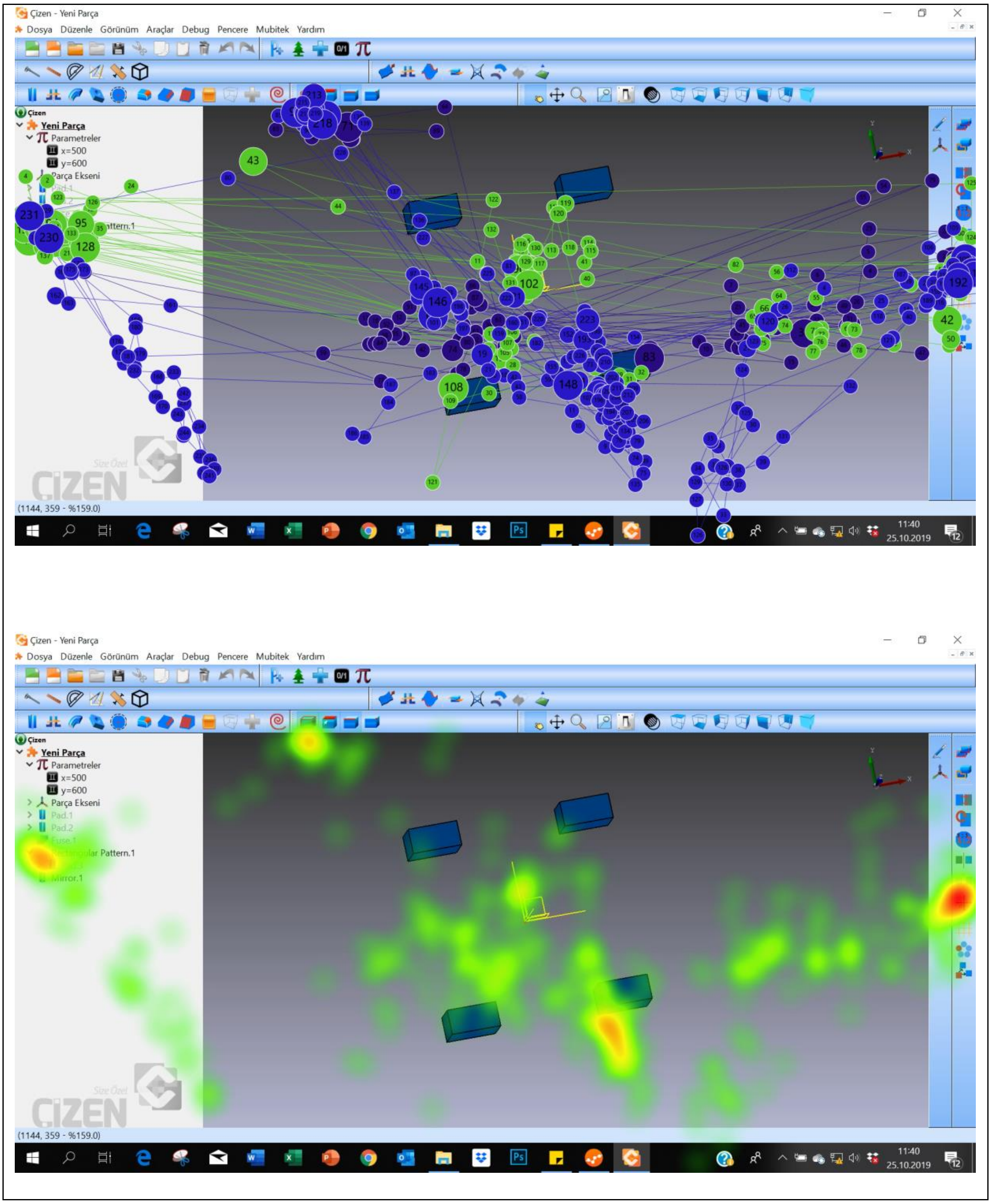

Figure 39. Task 7 Gazeplot and Heatmap (Right Menu Users)

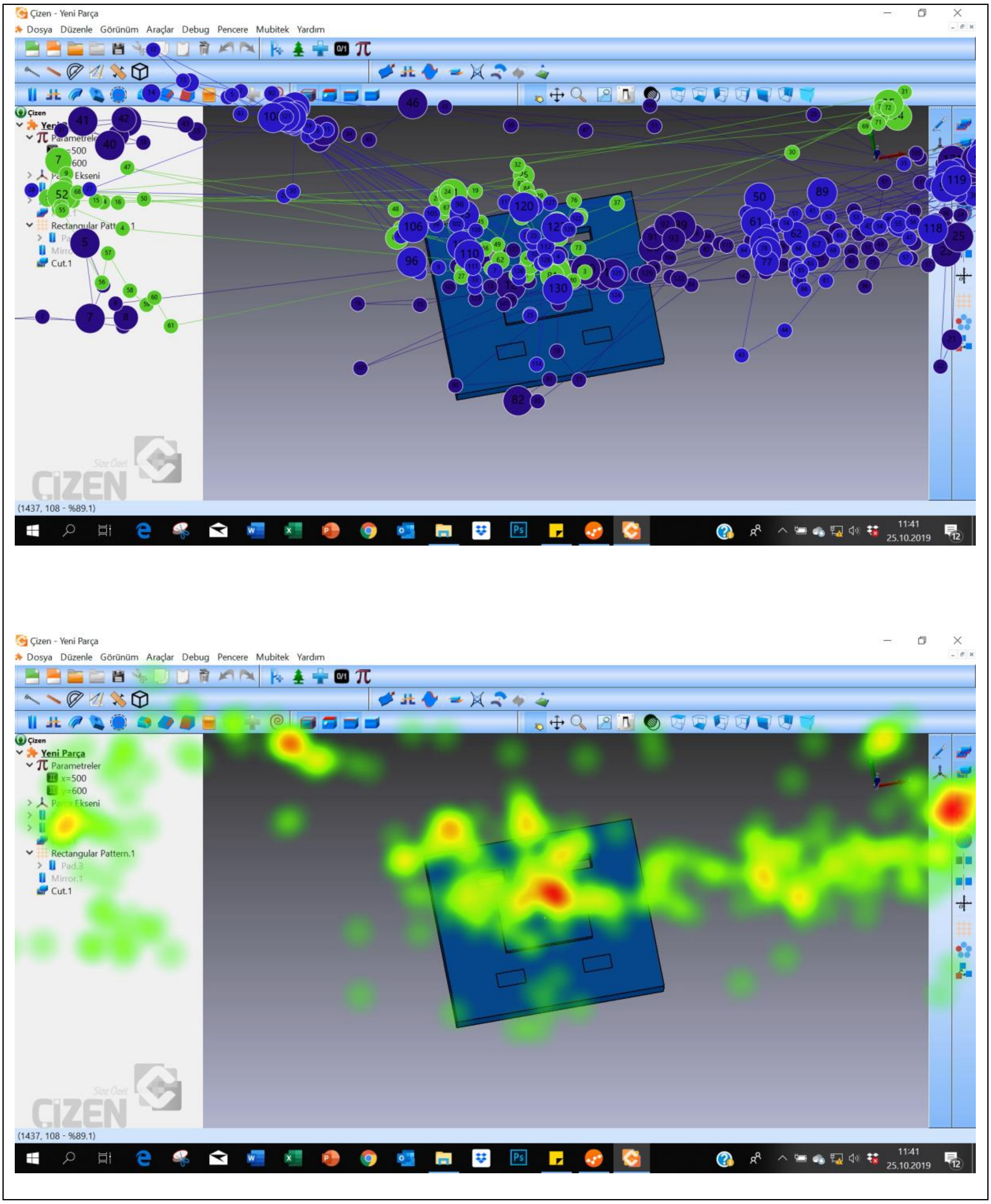

Figure 40: Task 8 Gazeplot and Heatmap (Right Menu Users)

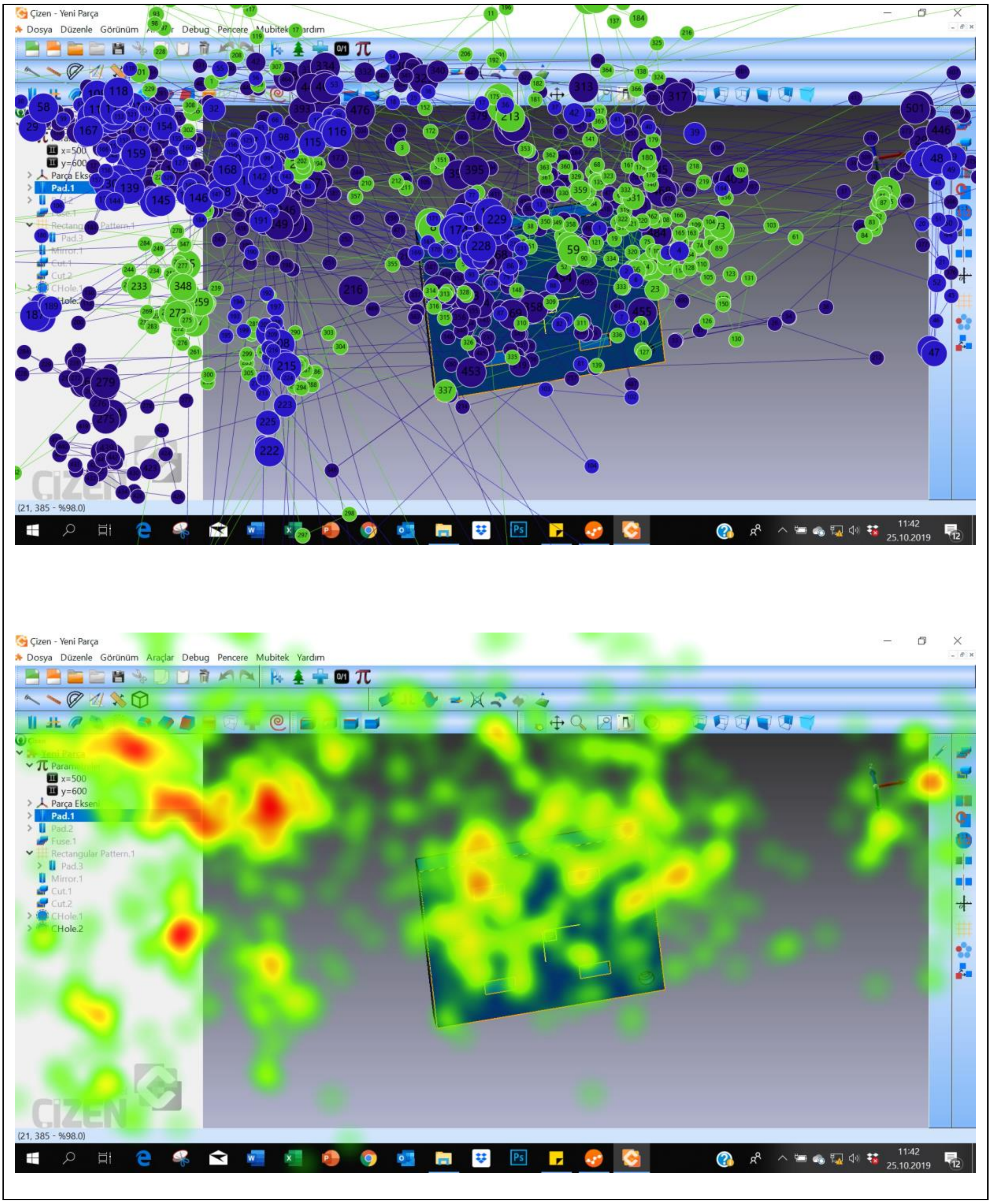

Figure 41. Task 9 Gazeplot and Heatmap (Right Menu Users)

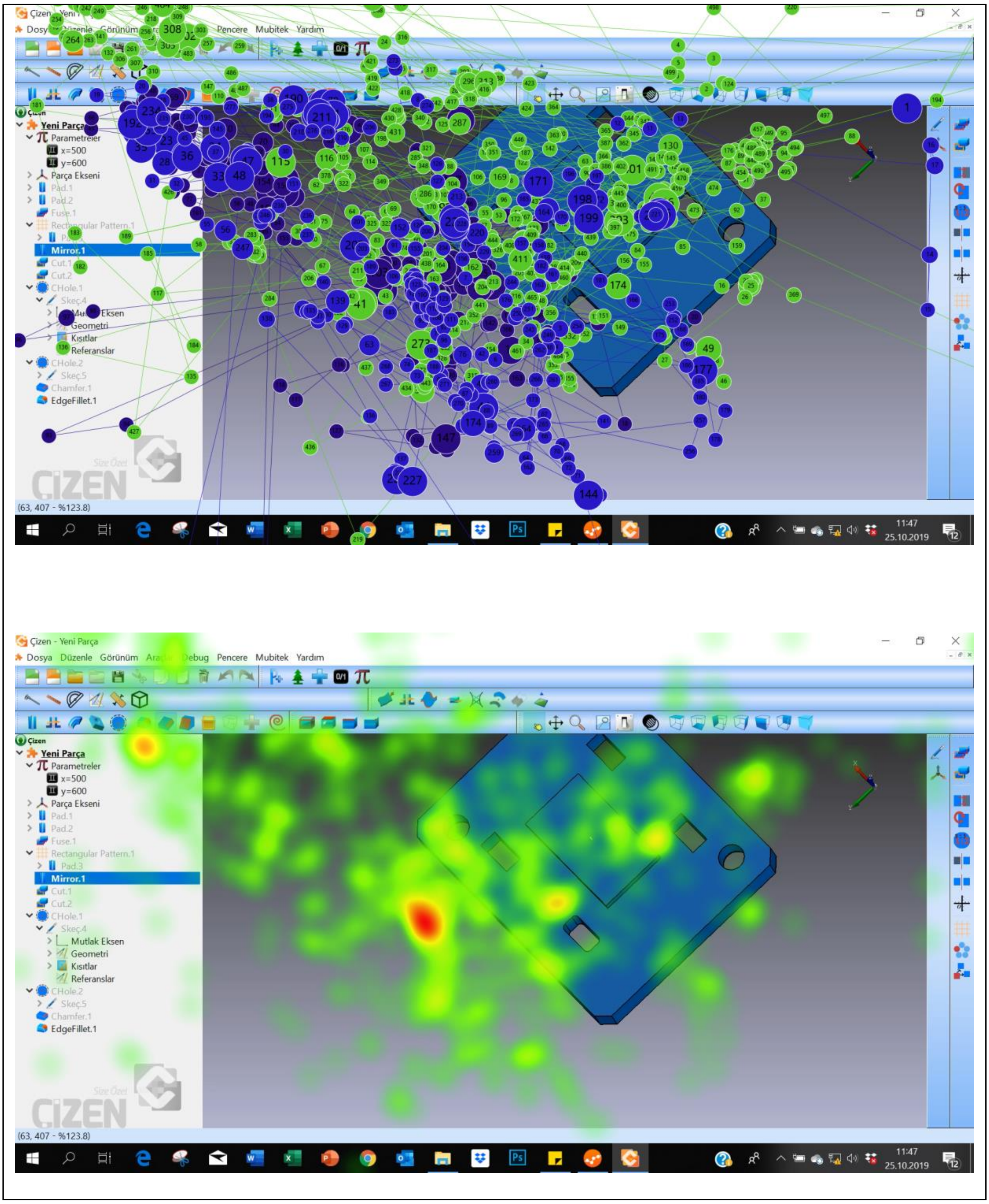

Figure 42. Task 10 Gazeplot and Heatmap (Right Menu Users)

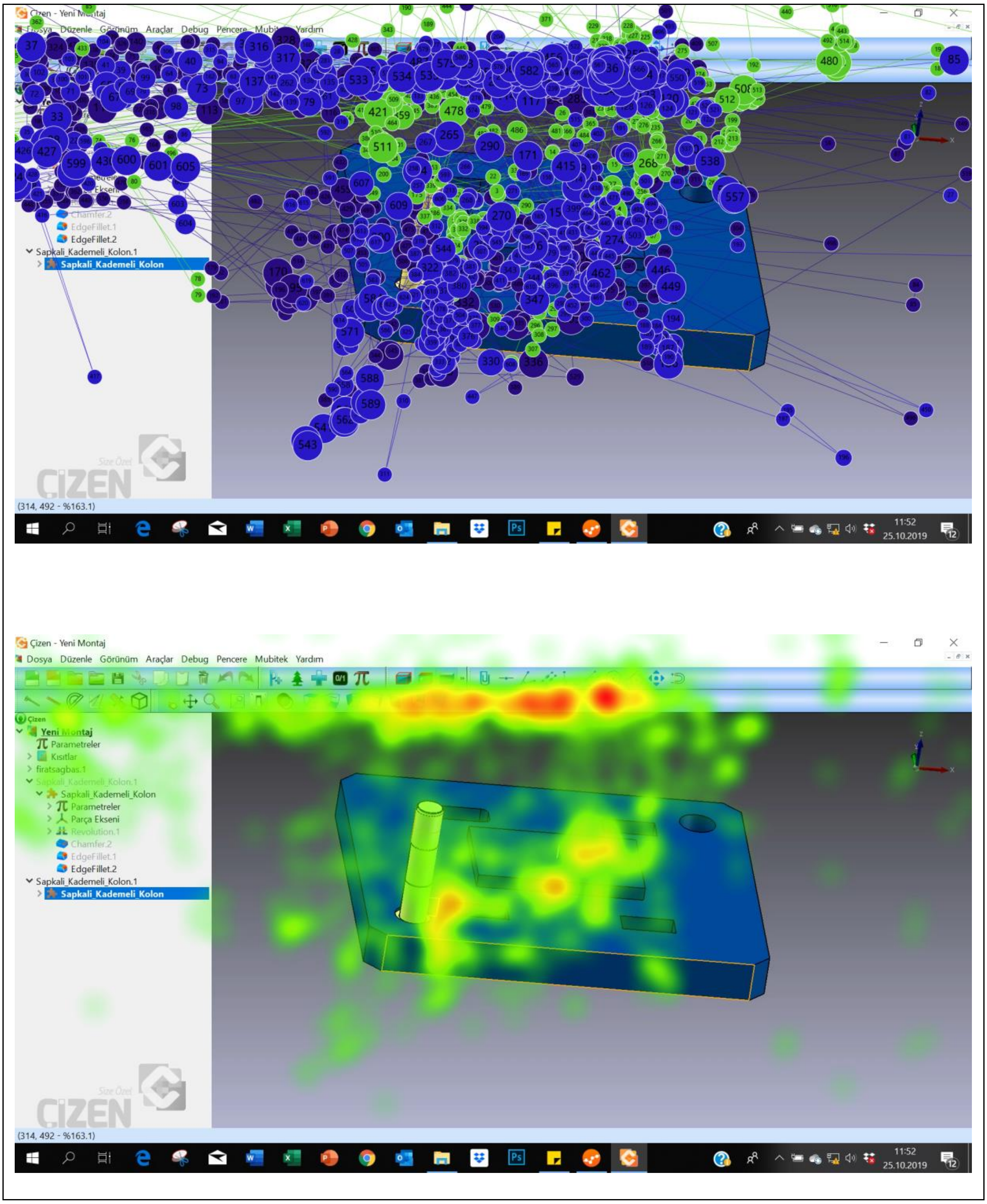

Figure 43. Task 12 Gazeplot and Heatmap (Right Menu Users)

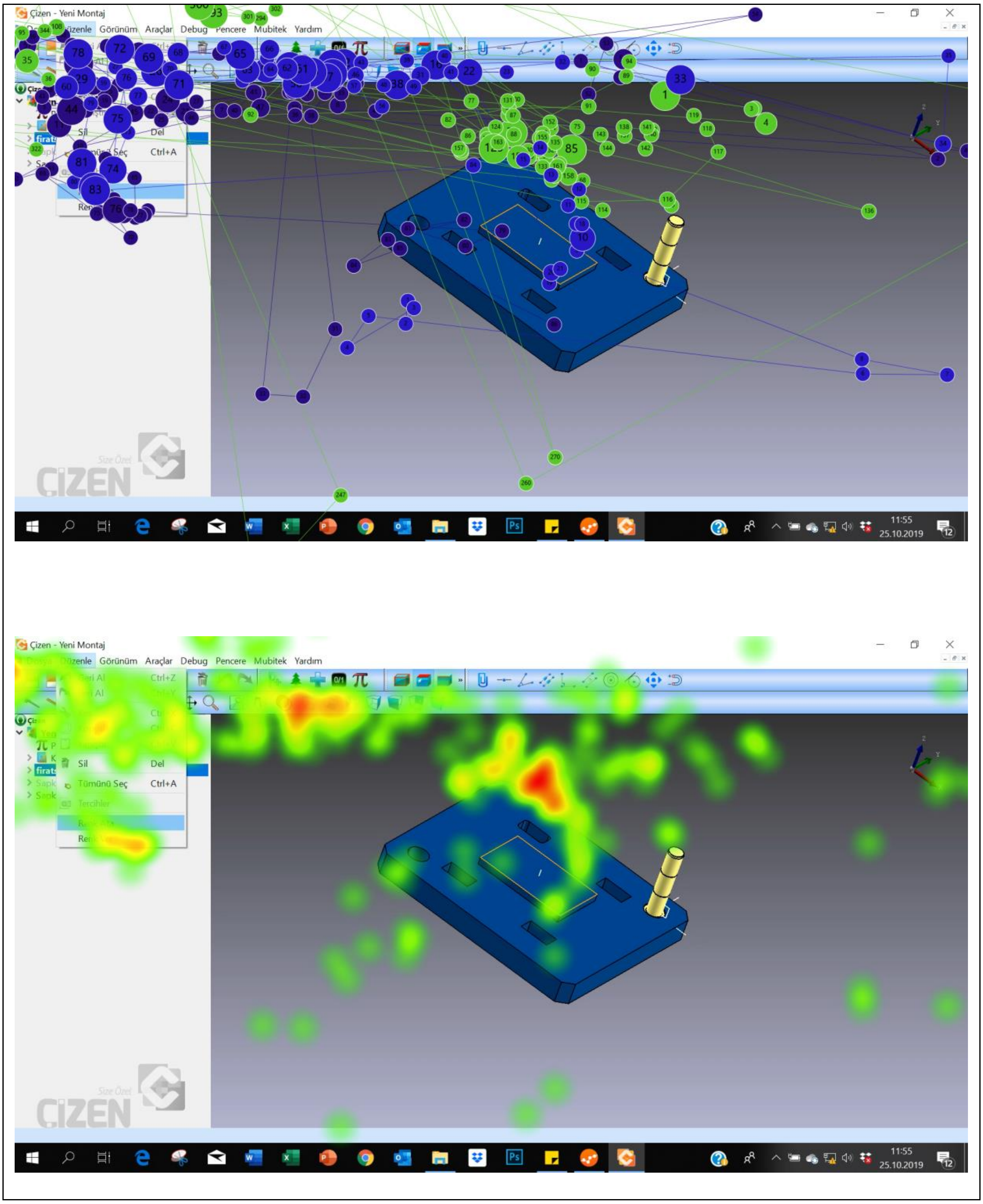

Figure 44. Task 13 Gazeplot and Heatmap (Right Menu Users)

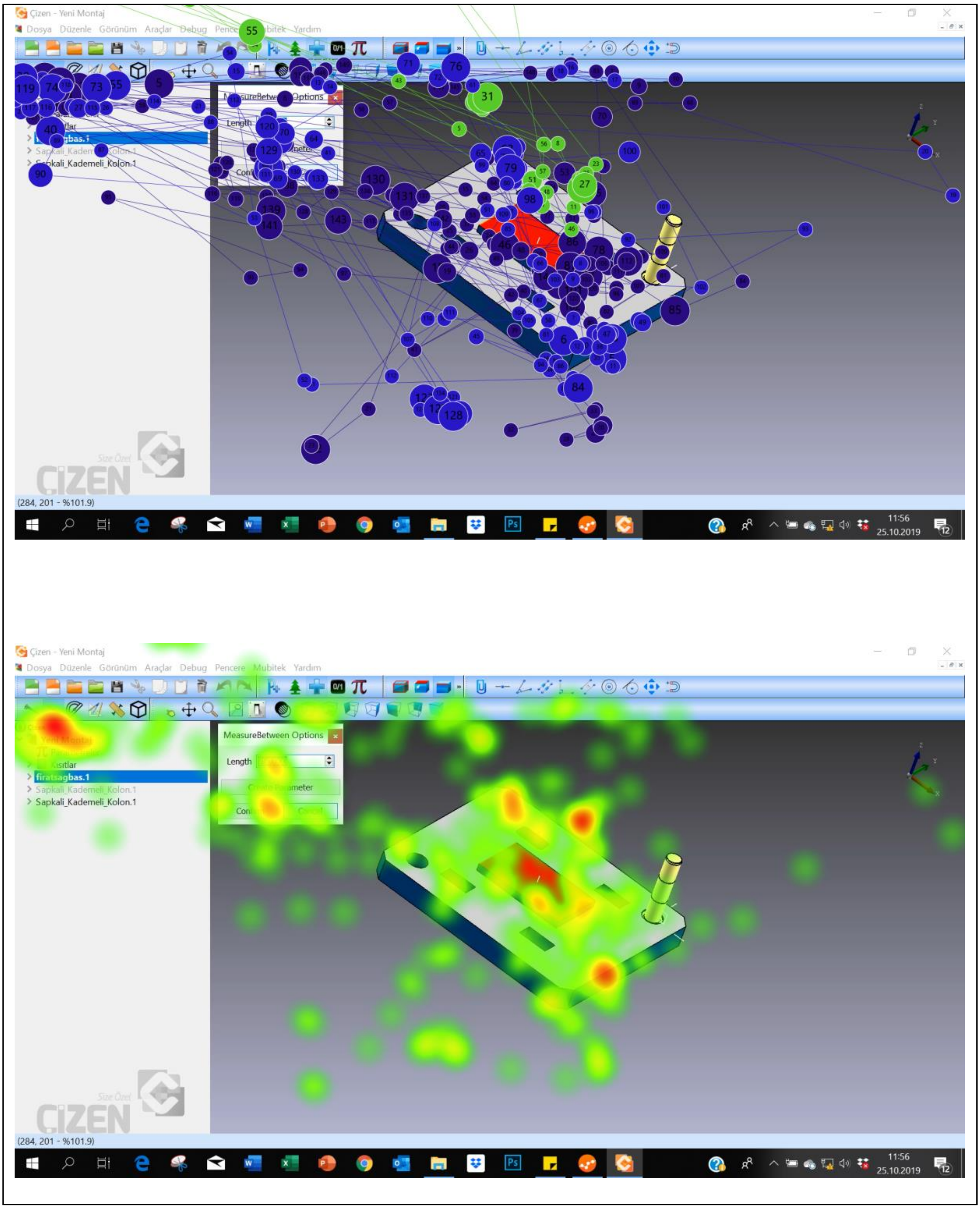

Figure 45. Task 14 Gazeplot and Heatmap (Right Menu Users)

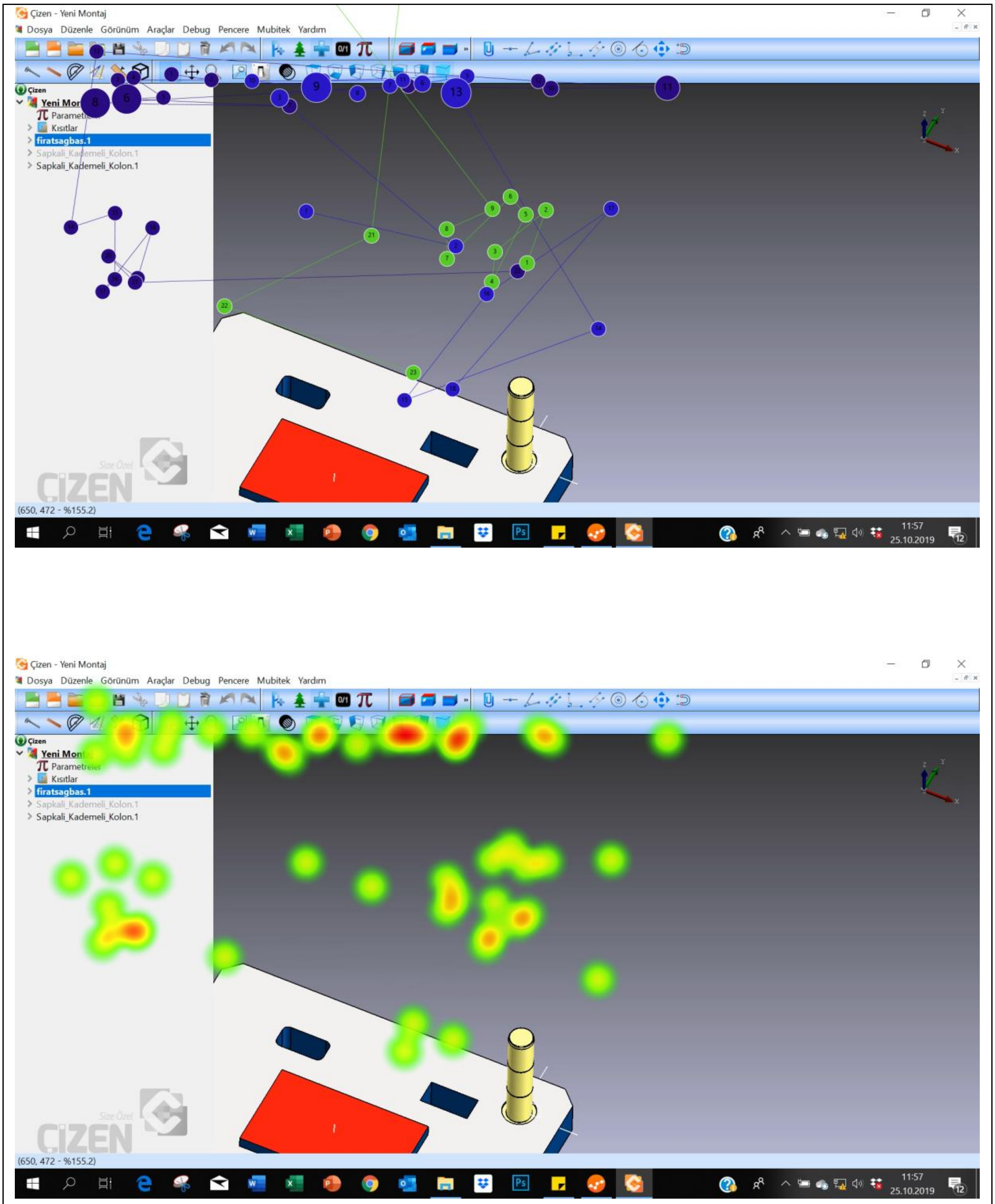

Figure 46. Task 15 Gazeplot and Heatmap (Right Menu Users)

Because of analyzing problem there is no gazeplot and heatmap in Task 11 and Task 16. In the research there is 14 Tasks gazeplot and heatmap.

Table 15. Gazeplots and Heatmaps Summary

Tasks	Explanation
Task 1	has so much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 2	has so much saccades but normal fixations. This indicates that participants difficultly found the commands, icons but easily did the task.
Task 3	has so much saccades but normal fixations. This indicates that participants difficultly found the commands, icons but easily did the task.
Task 4	has so much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 5	has so much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 6	has so much saccades and fixations. Fixations were especially on dimension change measurement screen. This indicates that participants have difficulty performing the task and especially difficult to measure.
Task 7	has so much saccades and fixations. Fixations were especially on dimension change measurement screen. This indicates that participants have difficulty performing the task and especially difficult to measure.
Task 8	has so much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 9	has so much saccades and fixations. Fixations were especially on dimension change measurement screen. This indicates that participants have difficulty performing the task and especially difficult to measure.
Task 10	has so much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 11	There are no gazeplot and heatmap in the task.
Task 12	has so much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 13	has much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 14	has much saccades and fixations. This indicates that participants have difficulty performing the task.
Task 15	has expected saccades and fixations. This indicates that participants have successfully performing the task.
Task 16	There are no gazeplot and heatmap in the task.

We wanted to find out the relationship between participants' task completion time and graduation status, job experience, CAD use and so on. But I didn't get a meaningful result. That is, the task completion time has nothing to do with other status. Table 16 shows us all the result about that.

Table 16. The Relationship Between Participants' Task Completion Time and Skills

Participant	**Task Completion Time**	**Graduation**	**Job Experience (year)**	**Program Using**	**Sector**
Participant 1	40,03	Graduate	1-3	Catia	Automotive
Participant 2	41,46	High School	1<	Catia	Automotive
Participant 3	33,16	Associate Degree	1<	Catia	Automotive
Participant 4	29,5	Associate Degree	1<	Catia	Automotive
Participant 5	33,36	Graduate	15>	Catia, Solidworks and Autoform	Automotive and Machine
Participant 6	26,19	Graduate	1-3	Catia	Automotive
Participant 7	35,18	High School	4-6	Catia	Automotive
Participant 8	50,15	Graduate	1<	Catia	Automotive
Participant 9	36,39	Graduate	15>	Catia	Automotive
Participant 10	22,34	Graduate	1-3	Visicad	Automotive
Participant 11	25,37	Graduate	7-15	Catia and Autoform	Automotive

Table 17. Result of Click Count Metric (All Tasks)

Number of Clicks in AOI	**Participant**	**Average**	**Median**
Recording3	oguzhanmorova	6,85	5,50
Recording4	tolgafıcıcı	5,30	3,00
Recording5	ramazanon	7,77	5,00
Recording6	muratkara	4,95	4,00
Recording7	erkanbulan	4,70	3,00
Recording9	muhammetemre	3,61	2,00
Recording10	FiratSAGBAS	3,16	3,00
Recording11	mehmettunali	4,93	3,00
Recording12	mustafaguzel	3,93	2,00
Average		5,02	3,39
Standard Deviation		1,49	1,22

Click count metric shows the number of clicks on the menu. It indicates that, there may be a usability problem if there are many clicks on the menu. When i examined in detail all the tasks, i saw that click count is ranging from 1 to 29. I think that more than 5 clicks are not acceptable.

Table 18. Result of Visit Count Metric (All Tasks)

Number of Visits (include zeroes)	**Participant**	**Average**	**Median**
Recording3	oguzhanmorova	4,69	0,00
Recording4	tolgafıcıcı	4,58	0,00
Recording5	ramazanon	3,95	0,00
Recording6	muratkara	5,10	0,00
Recording7	erkanbulan	3,58	0,00
Recording9	muhammetemre	3,05	0,00
Recording10	FiratSAGBAS	3,54	0,00
Recording11	mehmettunali	6,78	2,00
Recording12	mustafaguzel	3,27	1,00
Average		4,28	0,33
Percentage Fixated (%)			
Variance		1,36	0,50
Standard Deviation (n-1)		1,17	0,71

Visit count metric shows the number of visits to the menu. The more visits, the more difficult he is to find what he is looking for. Similar to visit duration. When i examine in detail all the tasks, i saw that visit count is ranging from 1 to 51. I think that more than 4 visits count are not acceptable.

Table 19: Result of Visit Duration Metric (All Tasks)

Total duration of Visit (include zeroes)	Participant	Average
Recording3	oguzhanmorova	8,20
Recording4	tolgafıcıcı	4,75
Recording5	ramazanon	4,97
Recording6	muratkara	4,46
Recording7	erkanbulan	2,83
Recording9	muhammetemre	5,96
Recording10	FiratSAGBAS	2,77
Recording11	mehmettunali	6,98
Recording12	mustafaguzel	3,80
Average		4,97
Share of Total Time (%)		
Percentage Fixated (%)		
Variance		3,32
Standard Deviation (n-1)		1,82

Visit duration metric shows the fixing time in the menu. The less fixation the easier he found the task. When i examine in detail all the tasks, i saw that visit duration is ranging from 0,08 to 102,21 second. I think that more than 4 visits duration are not acceptable.

Table 20. Result of Interval Duration Metric (All Tasks)

Duration of interval	Participant	Total Recording Duration
Recording3	oguzhanmorova	2403,59
Recording4	tolgafıcıcı	2506,84
Recording5	ramazanon	1996,33
Recording6	muratkara	1790,55
Recording7	erkanbulan	2016,38
Recording9	muhammetemre	1579,41
Recording10	FiratSAGBAS	2118,39
Recording11	mehmettunali	3015,90
Recording12	mustafaguzel	2199,37
Average		2180,75
Count		
Variance		179075,20
Standard Deviation (n-1)		423,17

Interval duration shows the time taken for users to complete their tasks. When i examine in detail all the tasks, i saw that interval duration is ranging from 1.790,55 to

3.015,90 second. As can be seen the interval duration results are not succesfull. So, there is a usability problem in Çizen.

Table 21. Participants' Task Completion Time and Comparison with Pilot Participants

TASKS	PILOT PARTICIPANT	PILOT PARTICIPANT	PARTICIPANT
	CATIA	ÇİZEN	ÇİZEN
TASK 1	0,31	0,65	4,68
TASK 2	0,59	1,05	1,25
TASK 3	0,11	0,36	1,79
TASK 4	0,37	1,00	2,18
TASK 5	0,46	1,71	2,31
TASK 6	0,32	0,33	2,23
TASK 7	0,11	0,17	1,2
TASK 8	0,08	0,29	3,3
TASK 9	1,33	2,02	2,19
TASK 10	0,34	0,78	2,11
TASK 11	0,50	0,14	0,47*
TASK 12	0,92	2,12	5,72
TASK 13	0,26	0,51	1,27
TASK 14	0,20	0,62	1,73
TASK 15	0,23	0,40	1,08
TASK 16		0,02	0,41*
TOTAL TIME	6,12 Minutes	12,17 Minutes	33,92 Minutes **

*In these tasks there are no gazeplots and heatmaps.

**Result of 11 participants tasks. If we take just 9 participant for whom analyzing eye tracking records, total time result is 36,34 second.

As can be seen Table 21, There is usability problem all the tasks without Task 2. Task 2 is related parametric design.

4.2.3. Post-Test Questionare Results

In the post test we asked 6 open ended questions, 1 multiple choice question and applied System Usability Scale and Nonverbal Pictoral Scale.

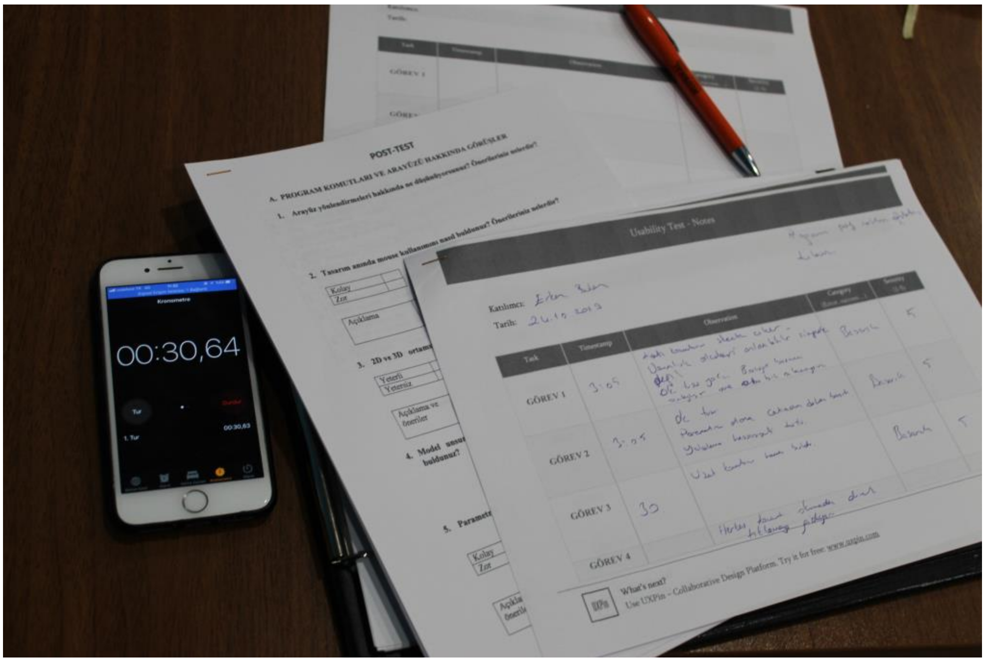

Figure 47. Photos of Post-Test Questionare and Usability Test Notes

4.2.3.1. Result of Open Ended Questions

In this test we analyzed all the answers and categorized to interface-technical infastructure and icon-command suggestions and problems using Affinity Mapping Methods. Table 22 shows to us Affinity Mapping results.After we categorized with affinity map, we did value based prioritization with technical staff working in Mubitek. During the meeting we examined all the suggestions and evaluated the importance-value and feasibility.

Figure 48. Photos of Analysis (Using Affinity Mapping and Value Priorization Methods)

Table 22. Affinity Mapping

ICONS & COMMANDS	INTERFACE AND INFRASTRUCTURE	PROBLEMS
*Center oriented square should be a rectangular *Should be added Compass *Commands should be grouped into modular *The command window should be open for command redirects *Icon designs should be sorted by frequency for use *Some icons rename have to be change *For sketch environment should be pencil not a paper *Appearance of assembling association commands should be change *The active command should be seem like active *Polygon should be added in 2D *The command window preview cancel button should be change as a ok button *Should be a command for Assign Color *Should be a command for Coloring *Should be a command for Lamba *Should be added projection command in Catia *Should be a command for take a screenshot tool *The assign color and the coloring commands are confusing *Measuring-analysis toolbar and set measure are confusing	*The new assembly and the new part icon locations should be change *User Interface can be customize *The solid command must be remove to sketch *The constraint is not simple, guiding should be added *Catia-NX programs can be taken as examples for guiding *Dimensions should be written next to Discart *Edge, face, solid selections are hard *Available operations should be added to the right-click menu *Operations should be enabled on right click *Measure operation should be enabled to do with digits *Feedback mechanism should be in help menu *The program should be have to guide for tricks *Should be change Mubitek Menu *Can repeated command be different color ? *Should be write to program version number *On the command can to inside to another command *The measure window should open while clicking but not with double-click *When 2 pads are selected, right click can be made for add and remove operations *The last entered dimensions should be in the memory *The parameter can be selected any modules. *When dimensions entered parameter can be set on tight click *Parameter assignment should be done with right click	*The part call command should appear in the interface in the assembly *Edge, surface, solid selection icons using is difficult. It have to be fixed. *Zoom in and zoom out commands should have their guides *Help file should have page number *It is hard to hide while doing Mirror and Empty operations *Interface routing is not good *While doing Radius-part operation if an edge is forgotten the whole operation must be done from scratch. Add-Remove Edges operation should be added. *When information in the product tree is deleted previous ones hide *Column hole operation have an error *The mirrors and symmetry commands are confusing *The screenshot tool icon have magnifying glass *Assembly operations are not good. *Some commands do not have explanations. Ex: overlap operation *Screenshot tool should be has icon *The Line selection has sensitivity issue *The Icons are similar to each other *Scale and define measure commands are confusing *In Sketch environment the vertical and horizontal line lengths are assigned together. *Offset projection is not working correct

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Table 22. (Cont.)

ICONS & COMMANDS	INTERFACE AND INFRASTRUCTURE	PROBLEMS
*Should be cross replication and move *Symmetry and parallelism commands are similar *Should be Publication *Snap command should be added *Icons views should change	*X-y-z should be written on the sketch *Navigation among the measures should be done by "Enter" key instead of "Tab" key *For constraint command should be has help menu on the sketch *Direction information should be on the part axis *An empty operation should be done by Extend, Empty operations as in CATIA *Command windows should contain the "OK" button *Face and plane selections should be focused on the measure *The Çizen logo can be remove on the product tree *While Mirroring a line the measure should be given by keys *Parameter assignment should be simple *Mouse control should be develop *Mirror operation should be enabled for a section of a part on a point *Geometrical shapes can be more in the 2D *Toolbars can be open and close and have names *The Part file and Assembly file place should be changed the product tree *Step, IGES, DXF and STL file transfer should be.	*Adding new parts while assembling, clicking no adds new parts *It is very difficult to memorize the contents of the commands *+ - does not detect when changing dimensions *Concentricity command does not work with surfaces *Rotate and surface rotate icon are the same *Difficult to find wanted command when product tree operations are extend *There are missing items in the interface guide *Pad command 'Extend' in Turkish version is not clear *Back-forward must be corrected *Difficult to find exit command from sketch *When the number of parameters increases, it may be difficult to find the relevant parameter from the list. *Mouse usage not good on the sketch *While creating constraints for parameters, it is hard to find the related length unit among the units *When dimension change true-false not working *Difficult to working with shift key *The icon for Layered Rotation operation is not related *Some commands in the command application are not detected when selecting from the product tree *Mouse sensing distance too short *Command and icon not match *Distortion in appearance during working (part of part not visible).

4.2.3.2. Result of System Usability Scale and Nonverbal Pictoral Scale

From all the participants we took System Usability Scale answers. After we calculated all answers. Lowest score is 47,5 point and higher score is 92,5 point. From some references I have read, the standard average of SUS score is 68. If the product reaches less than that, it have had to change the product usability (Hadi, Alathas, 2018).

Table 23. SUS Score Interpretation

Sus Score	Grade	Adjective Rating
>80,3	**A**	**Excellent**
68-80.3	**B**	**Good**
68	**C**	**Okay**
51-68	**D**	**Poor**
<51	**F**	**Awful**

Table 24 shows us the participants and pilot application SUS score results. Average result of participants is **68,4 point (Okay).** Average of pilot participants is 80,6 (Good) point. If we take the average, the result is 74,51 point (good)

Table 24. The Participants and Pilot Participants SUS Score

Participant	SUS Score	Pilot Participant	SUS Score
p1	60	p1	80
p2	87,5	p2	82,5
p3	92,5	p3	70
p4	85	p4	90
p5	82,5		
p6	47,5		
p7	47,5		
p8	55		
p9	52,5		
p10	62,5		
p11	80		

We also measuered the participants experience with a pictoral scale validity and reliability of the research. And some participant can not explain their feelings. Nonverbal Pictoral Scale helps explain feelings emotions. Figure 49 shows us the result. As can be seen from the result, most of participants like ÇİZEN program.

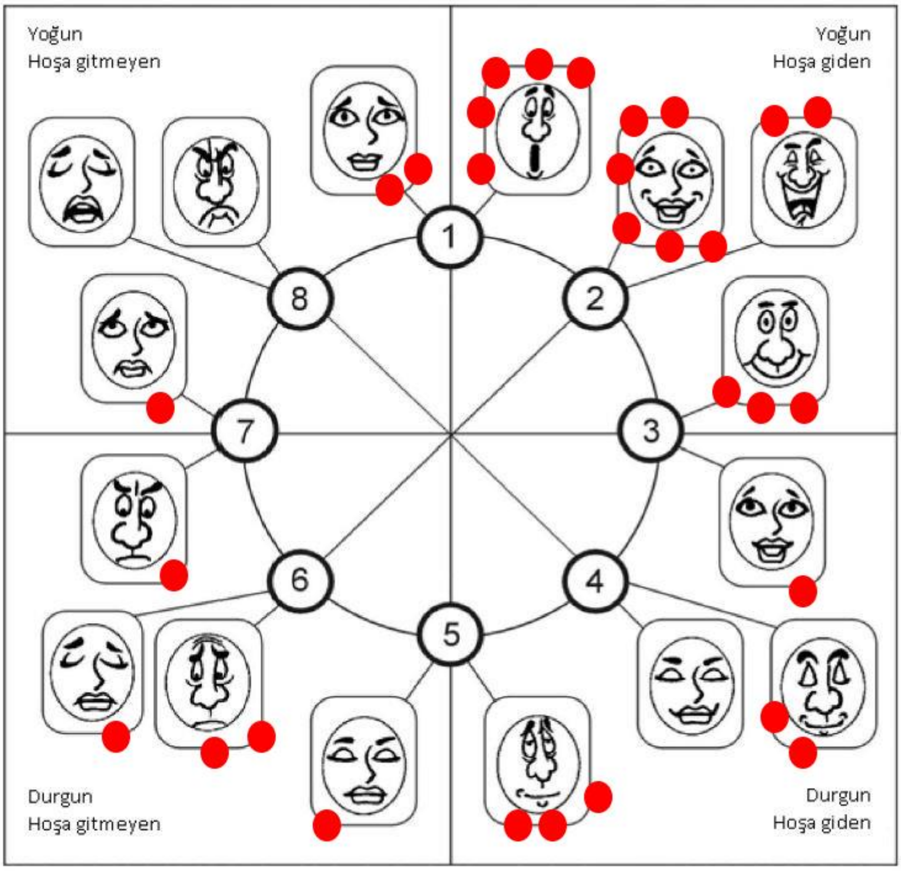

Figure 49. Nonverbal Pictural Scale Result

CHAPTER 5

DISCUSSION AND EVALUATION

Regardless of the interface design, the architecture of the game/program/software, the process flow, should be planned simultaneously with the user. However, the interface design is usually put at the end of these stages (Keş and Kara, 2015). But it is not the right aprroach. Because it may face many usability issues in the future. In that reason, in this research, we see that there is usability problem.

Use of eye tracking tools adds different aspects to the usability studies by providing objective and quantitative evidence to investigate participant's cognitive processes. So in this study uses so much methods to be objective and quantitative. Also usability research has benefits such as; Increased productivity, Increased sales and revenues, Decreased training and support costs, Reduced development time and costs, Reduced maintenance costs, Increased customer satisfaction. (User Experience Professionals Association, 2013)

I will present my evaluation result to ensure the availability of the program and improve the smart and lean interface, under three main headings: productivity, simplicity/functionality, and interface. But before than I would to explain my general opinions about research results seen from Table 25 and the problems experienced on the basis of participants seen from Table 26.

Table 25. General Views of Researcher About Research and Results

In the General Research Process	*****I wanted to choose a model to use a lot of menus in Çizen program. For analizing I set 16 tasks in the model. But 16 tasks forced participants. *****In analyzing stage 16 tasks were divided in to tasks. Eventually there were 23 tasks to analyze. For 9 participants to analyze 23 tasks would take so much time. So that we tried to put forward the most important results. *****During the performing of some tasks, problems occourred due to Çizen Program software infrastructure and affected completion of tasks.

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Table 25. (Cont.)

In the General Research Process (Cont.)	*Starting to record eye tracking, also problems occurred due to Tobii X2 Pro device I don't know why. Therefore from some participants we couldn't take recording. So we tried new participants. *Some of participants we selected were not willing to participate, so that, we gave them Sodexo Gift Check to thank.
In the Process of Performing Tasks	*In trial application, Participant did the tasks reading his own. But it took a lot of time and coused data loss on eye tracking device. So we read the tasks to other participants. *We want to start to design without giving information about Çizen Program. Because the program is Turkish, we believed participants could. But in most of tasks participants had difficulty understanding general logic of program. I think it would be better if we introduced the program before application. *During the application we have found that the translation in some menus from English to Turkish incorrect. This makes it difficult for participants to understand. *We found that participants using Catia were more successful in using Çizen Program. *Tasks completion time difference between pilot participants and research participants is so high. Pilot participants completed in 12,17 minutes. But research participants completed 33,92 minutes. There is almost 3 times difference. I think, this result shows us there is a usability program in the Çizen program. We can normally accept the difference for not introducing the program before. But this difference is still so high. *We also compared Çizen Program with Catia with pilot participants. Mubitek company wants to use Çizen Program instead of Catia in sheet metal forming sector. At this stage it does not seem possible. Because tasks completion time is 12,17 minutes in Çizen and 6,12 minutes in Catia. There is 2 times difference.

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Table 25. (Cont.)

In the Post-Test and Interview Process	*Although the results of analysis are not good, overall views are positive because of national and domectic program. *For example System Usability Scale result is 68,4. This is acceptable result. Also Nonverbal Pictural Scale result is fine. I think, the program can be successful if developing process continues. *Of course, many suggestions and criticism came from the participants. I examined in detail and got the opinions of software developers and designers working in Mubitek. After that I took out the suggestions that can be done.

Table 26. Participants-Based Problems in The Tasks Observed by Researcher

Participant	Comments
Participant 1	Task 1- Forced to exit from sketch Task 2 – Forced to put parameter relationship Task 4-Forced to find Simmetry Icon Task 8 – Forced to do user pattern Task 12-Did not find the standart part because of menu name (menu name is mubitek) Task 16-Forced to find “feedback menu”
Participant 2	Task 1- Forced to give measure Task 6- Forced to do Rectangular Pattern Operation Task 13- Give Colour Operation Task 16-Forced to find “feedback menu”
Participant 3	Task 1- Forced to exit from sketch Task 7-Confused Mirror and Pattern Operation Task 12-Did not do Assembly Operation Task 13-Confused Give Colour Operation

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Table 26. (Cont.)

Participant	Comments
Participant 4	Task 3- Forced to do Pad Operation Task 4- Forced to exit from sketch Task 6- Forced to do Rectangular Pattern Operation Task 13-Confused Give Colour Operation Task 14-Find “Yardım Menu”, under all the menus Task 16-Forced to find “feedback menu”
Participant 5	Task 4- Forced to do Add Operation Task 6- Forced to do Rectangular Pattern Operation Task 7-Confused Mirror and Pattern Operation Task 8-Forced to do Remove Operation Task 13-Confused Give Colour Operation Task 16-Forced to find “feedback menu”
Participant 6	Task 1-Forced to do Sketch Operation Task 4- Forced to do Add Operation Task 6- Forced to do Rectangular Pattern Operation Task 12-Forced to do Assembly Operation Task 13-Confused Give Colour Operation Task 14-Forced to find “Yardım menu” Task 16-Forced to find “feedback menu”
Participant 7	Task 1- Forced to give measure Task 3- Forced to find Pad Operation Task 6- Forced to do Rectangular Pattern Operation Task 14-Forced to find “Yardım menu” Task 16-Forced to find “feedback menu”

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Table 26. (Cont.)

Participant	Comments
Participant 8	Task 1-Forced to do Sketch Operation Task 3- Forced to do Pad Operation Task 4- Forced to do Add Operation Task 6- Did not do Rectangular Pattern Operation Task 8-Forced to do Remove Operation Task 15- Looking for “Screenshot” under view menu Task 16-Forced to find “feedback menu”
Participant 9	Task 6- Forced to do Rectangular Pattern Operation Task 12-Forced to do Assembly Operation Task 13-Confused Give Colour Operation Task 15- Looking for “Screenshot” under view menü Task 16-Forced to find “feedback menu”
Participant 10*	Task 1-Forced to do Sketch Operation Task 4- Forced to do Add Operation Task 12-Did not find the standart part because of menu name (menu name is mubitek) Task 13-Confused Give Colour Operation Task 15- Forced to find “Screenshot” Task 16-Forced to find “feedback menu”
Participant 11*	Task 1- Forced to exit from sketch Task 9-Forced to find Gradual Hole Operation Task 16-Forced to find “feedback menu”

*The reason I mentioned earlier Participant 10 and Participant 11 were not included eye traking analyze.

As can be seen Table 26, participants were forced on almost all tasks. But on some tasks all participants were forced such as task 1- task 6, task 13. Also almost all participants clicked the number to give measure all time. But in Çizen, measurement can be given just to click line. So, giving measurement operation must be changed in Çizen.

When all analyzes were completed, we evaluated all the results with Technical Manager (Mechanical Engineer and designer) and Software Manager (Computer Engineer). After evaluation, i summarized evaluation result under 3 main headings the needs to be done for the program. If the three main headings are effectuated by software developer company Mubitek, Çizen program will be successful. But now there is a usability problem in Çizen.

Table 27. Evaluation Result

Productivity	*Mirror Command axis should be picked part surface *Mouse usage should be customizable to the user *The page number in the Help File should be written *A guide should be added how to use the parameter operation *Measurement change operation should be done automatically *Commands in the toolbars should be arranged according to frequency of use *With double click on the icon, command repetition should be done *Right-Click window should be created for constraints *Copy-paste command should be created *Undo-Redo operation bug should be fixed *In concentricity operation, surface should be picked
Simplicity/ Functionalty	*To add description window for Zoom-in Zoom-out command *The previously entered measurement values in the Hole Command should be remain in the command window *New geometric shapes should be added in 2D operation *In product tree, easily see which parameter is assigned in parameter operation *Give colour and assign colour commands should be combined *Move command should be developed *.dxf extension should be added *Publication property should be added *All the chages in design should be done not only product tree but also on design *Commands group should be hidden and displayed at any time *Solid and surface commands shoulb be in seperate interface

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Table 27. (Cont.)

Interface	*During openin Çizen Program, which version of the program can be written *Remove the Çizen Logo from the product tree *Feedback menü should be added *All description Windows related commands should be updated *The toolbar name Mubitek which is related standart part library should be changed *The names of toolbars translated into Turkish should be reviewed, some of them should be chaged. *The interface colour should be customizable *The background colour should be distinct which command is active *Preview window should be added for commands *Most icon images should be chaged. *On the part axis, x-y-z direction information should be written *There should be user guide in HTML format *Toolbars should be named

CHAPTER 6

CONCLUSION

Sheet metal forming design is one of te most important issues in terms of time and cost savings in automotive industry. Failure or prolongation of sheet metal design steps adversely affetcts the cost and delivery times of a new product. Quality, time and error-free design is possible with CAD Programs. Nowadays, as a result of the rapid progress of technology, technical infrastrcture and features of CAD programs are developing and user-orientied interfaces emerge. All of CAD programs used in Turkey is foreign products. Lack of a domestic program and high licensing fees push the sector to unlicensed use. The Çizen program, which has been developing from this point of view is gain for sector because of a sale price and Turkish language support. It will be easy to expand the program to small enterprises due to the financial problems of small businesses and the foreign language problem of designers. However, as a result of the negative feedback coming form the users of the program, the need for user-oriented research on the interface emerged. Within this scope, firstly, the future vision of CAD Technologies was investigated. Evaluate the Çizen program efficiency and the design methodologies of different designers were extract and program interface research with task based was conducted.

As a result of the research, it is seen that CAD technology is more oriented towards cloud based operation and licencing fees are transferred to leasing model. Also, sheet metal forming design program is already offered for sale in Turkey. It was reported that the authorized distributor will start advertising activities in the year 2020. When Çizen program was compared with Catia V5, it is found that the Çizen program is performed in less steps in some commands. About 120 commands and features compared in 5 sections. However, although Çizen performed some operations/commands with fewer steps, it was observed that the tasks completion time was 2 times longer than Catia V5. In this study, a diemold was also designed with 4 CAD programs. These programs are Solidworks, Visicad, Catia V5 and Çizen. The design was examined under 11 headings. As a result of design it had been that Catia V5 and Çizen were similar design methodologies. Solidworks program was found to be insufficient in progressive sheet metal forming design.

In the user experience research, a basic sheet metal forming was designed by potential ÇİZEN program users. As a result of this design, it has been determined that, there is approximately 3 times difference between the pilot participants and research participants in the completion time of the tasks. In addition, in the majority of tasks, research participants have lost a lot of time, searched the relevant command for a long time, made too many clicks on the commands and found it difficult to understand the design steps.

The results were evaluated and to develop a smart, lean interface, 11 task were identified under productiviy, 11 task were identified under Simplicity/Functionalty and 13 task were identified under Interface. Although System Usability Scale and Nonverbal Pictural Scale results were "Okay" the issues identified showed that there is a usability problem in the ÇİZEN Program. If evaluation results under 3 headings are applied to the Çizen program, the Çizen program usability increases and can reach its goals described in the Background Section.

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APPENDIX A

SUMMARY OF RESEARCH LITERATURE

Related Topic	Focus	Aim	Referances
Sheet Metal Forming Design Technology	Productivity	They also entered the information of the standard elements in the prepared parameters so that they quickli create material list. They have created mold design modules with this technique. Actually this technique is a computer aided parametric design.	Lee, R.S., Q.C. Hsu., S.L. Su. (1997). Development of a Parametric Computer-Aided Die Design System for Cold Forging. Department of Mechanical Engineering, National Cheng Kung University, 1 University Road, Tainan 701, Taiwan. p. 80-89
Sheet Metal Forming Design Technology	Expert System Approach	Machine elements are assembled with the dahabase they prepared. To do this, they used expert system approach technique.	Myung, S., S. Han. (2001). Knowledge-Based Parametric Design of Mechanical Products based on Configuration Design Method. ICAD Laboratory, Department of Mechanical Engineering, KAIST (Korea Advanced Institute of Science and Technology), ME3080, 373-1 Kusong-dong, Yusong-gu, Taejon, South Korea, p. 99-107
Sheet Metal Forming Design Technology	Using CAD Programs	They used CAD program for prograsive sheet metal forming design	Duffey, M.R., Q. Sun. (2003). Knowledge-based design of Progressive Stamping Dies. Department of Mechanical Engineering University of Massachusetts at Amherst, MA 01003, USA. p. 221-227

Sheet Metal Forming Design Technology	Using CAD Programs	They had developed a macros running with AutoCAD program for making prograsive sheet metal forming design	Kumar, S., R. Singh. (2006). An Intelligent System for Automatic Modeling of Progressive Die. Journal of Materials Processing Technology, Department of Mechanical Engineering, Hindu College of Engineering, Sonepat, Haryana, India. p. 176-183
Sheet Metal Forming Design Technology	Using CAD Programs	Using a user interface they designed a mold design for automobile tires without preparing technical drawing.	Chu, C.H., M.C. Song., (2004). Computer Aided Parametric Design for 3D Tire Mold Production. Computers in Industry, Department of Industrial Engineering and Engineering Management, National Tsing Hua University, Taiwan, p. 11-25
Sheet Metal Forming Design Technology	Using CAD Macros and Parametric Design	The rules were assigned to the drawing sheet metal forming set prepared in this study, so that the sheet metal forming could be adapted to new situations. This set, which is prepared in the “Pro / E” solid model design program, takes the information about the press and other sheet metal forming design information from the parameters which are a feature of cad program. This technique is a computer aided parametric design approach.	Lin, B.T., S.H. Hsu. (2006). Automated Design System for Drawing Dies, Expert Systems with Applications. Dept. of Mechanical and Automation Engineering, National Kaohsiung First University of Science and Technology, Taiwan. p. 1586–1598
Sheet Metal Forming	Using CAD	With an interface have been	Kim, C.W. , C.H. Park.,

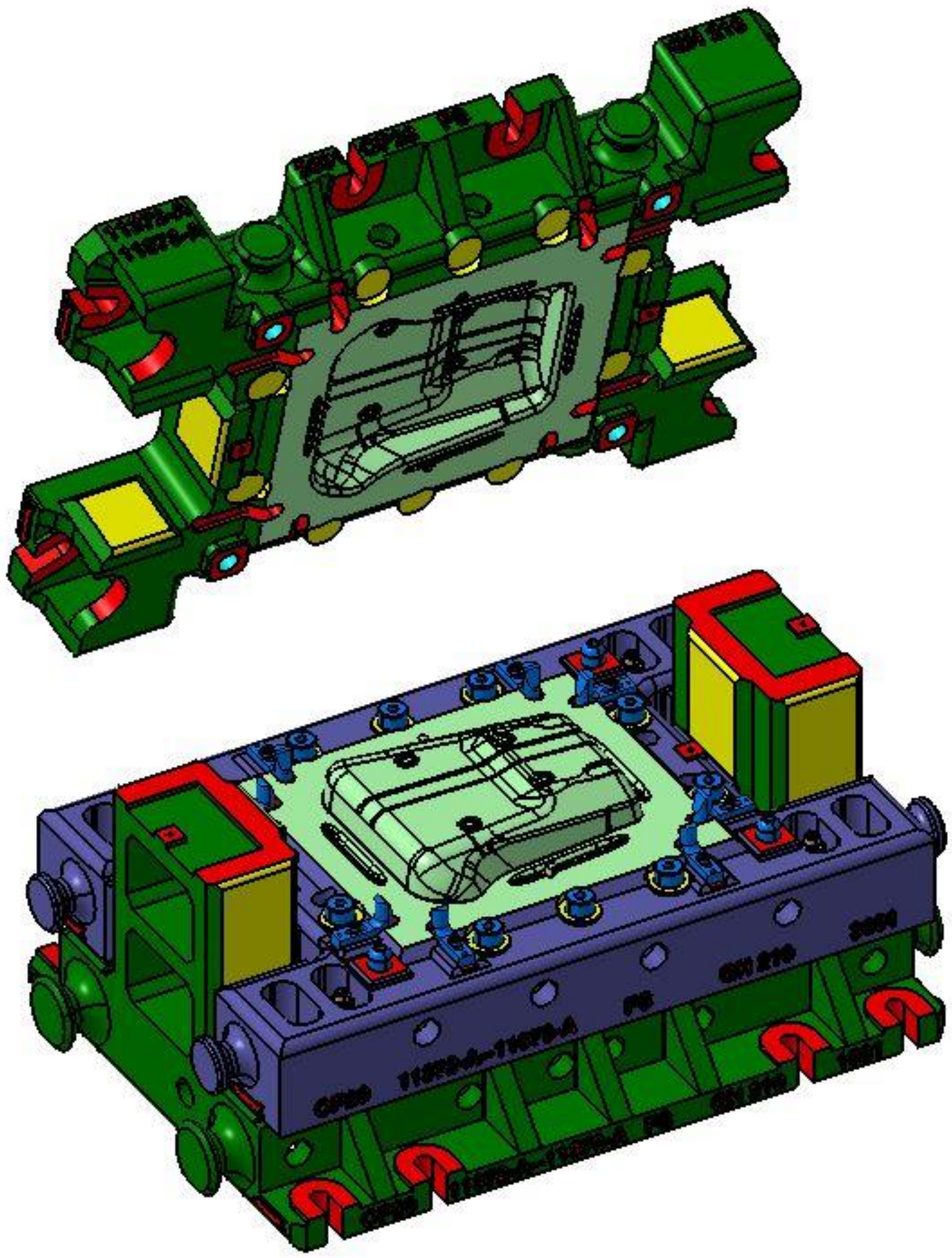
Design Technology	Macros	able to prepare to control a small and simple cutting pattern. In this study, Basic Visual Basic 6.0 ”programming language is used. With this study, the sheet metal forming set was standardized. Pattern information is entered into the Access Ms Access ”database. The simplicity of the sheet metal forming is the most important factor in the effectiveness of the study. They used parametric design approach in their study.	S.S. Lee. (2007). An Automated Design System of Press Die Components Using 3-D CAD Library, Professor. School of Mechanical Engineering, Konkuk University, Korea. Part II, p. 961–974
Sheet Metal Forming Design Technology	Using CAD Macros and Expert System Approach	They used the “Knowledge” module, a special extension of the CATIA program. This module allows writing rules for the desired transactions. The study is an example of the development of solid model programs. The technical expert is the system approach.	Skarka, W. (2006). Application of MOKA Methodology in Generative Model Creation Using CATIA. Engineering Applications of Artificial Intelligence, Poland. P. 677-690
Sheet Metal Forming Design Technology	Using CAD Macros and productivity	They wrote macros and obtained parametric intelligent design libraries by using “Knowledge CA module, a special extension of CATIA program.	Bintaş, Mustafa and et all (2011). Development of Computer Aided Die Design Software and Die Design Process Modeling, 6th International Conference and Exhibition on Design and Production of Machines and Dies/Molds, 23-26 June 2011, Atılım University, Ankara
User Experience Research	Using Eye-Tracking for Yacht Design	The parameters of yacht hull design were examined with eye tracking device and visual evaluations were made. As a	Doğan, Kemal Mert. Suzuki, Hiromasa. Günpınar, Erkan. (2018). “Eye Tracking for

		result of these evaluations remarkable opints were determined in order to improve the quality of the relationship between the parameters	Screening Design Parameters in Adjective-based Design of Yacht Hull", The University of Tokyo, Graduate Scholl of Engineering, Bunkyo, Hongo, Chome 3-1, 113-8654, Tokyo Japan,
User Experience Research	Using Eye-Tracking for Hand Gestures Based 3D CAD Modelling	Sketch and 3D drawings used in CAD software fort he use of the şnterface has made a study taking ino account movement. In the study qualitative and quantitative understanding of the various elements that form a gesture based CAD system. These studies mainly focus on just one of the aspect such as algorithm development, ergonomics/ biomechanics or linguistic viewpoint of hand gestures. The number of research studies focusing on the interaction aspect involved in a 3D CAD modeling task performed by the users is less. Some tasks were given to users and record camera. Movements in command were recorded. After that mapping of movements in CAD commands. This was made with 15 users.	Thakur, Aditya, Rai Rahul. (2015).User Study Of Hand Gestures For Gesture Based 3d Cad Modelıng, Proceedings of the ASME 2015 International Design Engineering Technical Conferences&Computers and Information in Engineering Conference IDETC/CIE 2015 August 2-5, Boston, Massachusetts, USA.
User Experience Research	Using Eye-Tracking for interface design for CAD software developers	They had developed an interface that will increase the capabilities of the users and facililate the use of the tools. They developed an argument for reconfigurable CAD interfaces. The spesific purpose of this study is to enable software developers to develop software with the features required by CAD users. Fort his reason a system has been developed and tested	Kelley, Mitchell and Rosen, David W. (2016). Reconfigurable User Interfaces For Cad Applıcatıons Proceedings of the ASME 2016 International Design Engineering Technical Conferences and

		more than the interface. We have only explored rCAD for mouse and keyboard access to a 2D CAD API. We look forward to adapting the concept to more input devices and 3D CAD APIs. Some high dimensional user interface devices such as cameras and motion trackers have not yet established wide-spread usage paradigms. Perhaps rCAD can help enhance usability for novel interface devices.	Computers and Information in Engineering Conference IDETC/CIE 2016 August 21-24, Charlotte, North Carolina.

APPENDIX B

MOLD DESIGN



APPENDIX C

PRE-TEST FORM (TURKISH)

A. <u>KİŞİSEL VE MESLEKİ BİLGİLER</u>

1. **Tasarımcı Adı Soyadı:**
2. **Yaşınız:**

<25	
26-30	
31-35	
36-40	
41-45	
46 >	

3. **Eğitim Durumunuz ve Bölümünüz(Makine Mühendisi, Teknik Öğretmen vb.):**

Eğitim Durumunuz		**Mezun olduğunuz bölüm**
Lisans ve üstü		
Ön Lisans		
Lise		

4. **İş Tecrübeniz:**

<1 Yıl	
1-3 Yıl	
4-6 Yıl	
7-15Yıl	
15	

5. **Kullandığınız 3D Program(lar) ve Tecrübesi:**

Program	**Yıl**
Catia V5	
Solid Works	
NX	
Visi CAD	
Diğer (Belirtiniz)	

6. **Tasarım yaptığınız sektör:**

Sektör	**Yıl**
Otomotiv	
Makine	
Savunma San.	
Havacılık	
Diğer (Belirtiniz)	

B. SAC KALIP TASARIM - VERİMLİLİK, YALINLIK VE FONKSİYONELLİK İLE İLGİLİ SORULAR

1. **Kullandığınız programdaki ürün ağaç yapısından bahseder misiniz? Kalıp tasarımı sürecinde uygulamak istediğiniz bütün işlemleri rahatça gerçekleştirmenize (örneğin; gruplama yapılabilmesi, body mantığına dayalı çalışılabilmesi, yardımcı kontur, yüzey gibi unsurları bulundurabilmesi vb.) olanak sağlıyor mu ? Beğendiğiniz ya da eksiklik olarak gördüğünüz özellikleri varsa belirtiniz.**

2. **Tasarımlarınızda kalıp seti kullanıyor musunuz? Kullanıyorsanız kalıp setini oluşturmada izlediğiniz yöntem nedir? Kullanmıyorsanız bu durumun tasarım sürecine etkisi ne yöndedir?**

		Açıklama
Evet		
Hayır		

3. **Parametrik ve linkli çalışma yapıyor musunuz? Yapıyorsanız veya yapmıyorsanız bu durumun kalıp tasarımı konusunda avantaj ve dezavantajları neler oluyor?**

Evet	
Hayır	

Açıklama	

4. **Standart ve imalat parçaların montajını tasarım içerisinde nasıl gerçekleştiriyorsunuz?**
5.
6. **Kalıp revizyona uğradığında, değişiklikleri tasarıma uygulama konusunda zaman ve uygulanabilirlik açısından sorun yaşıyor musunuz? Evet ise neler olduğunu açıklayınız.**

Evet	
Hayır	

Açıklama	

7. **Kalıp tasarım sürecinde, kullandığınız Cad programında hangi aşama veya işlem sizin için uğraştırıcı oluyor ve tasarım sürecini uzatıyor? Bu konu hakkında bir öneriniz var mı ?**

8. **Montaj ve sketch ortamında unsurları taşıma/hareket ettirme işlemlerini nasıl gerçekleştiriyorsunuz? Bu konuda kullandığınız programı yeterli buluyor musunuz?**

9. **Kullandığınız programdaki komutları kalıp tasarımı alanında yeterli olma ve işlevsellik açılarından değerlendiriniz.**

10. **Tasarım programınızda beğendiğiniz ve sorun olarak gördüğünüz 3'er özellik yazınız.**

Beğenilen Özellikler	**Sorun görülen**

11. **İçe/Dışa dosya aktarımı konusunda tasarım programınızı yeterli görüyor musunuz? (Tasarım içerisine eğri, yüzey vb. unsurları almada bir sorunla karşılaşıyor musunuz?)**

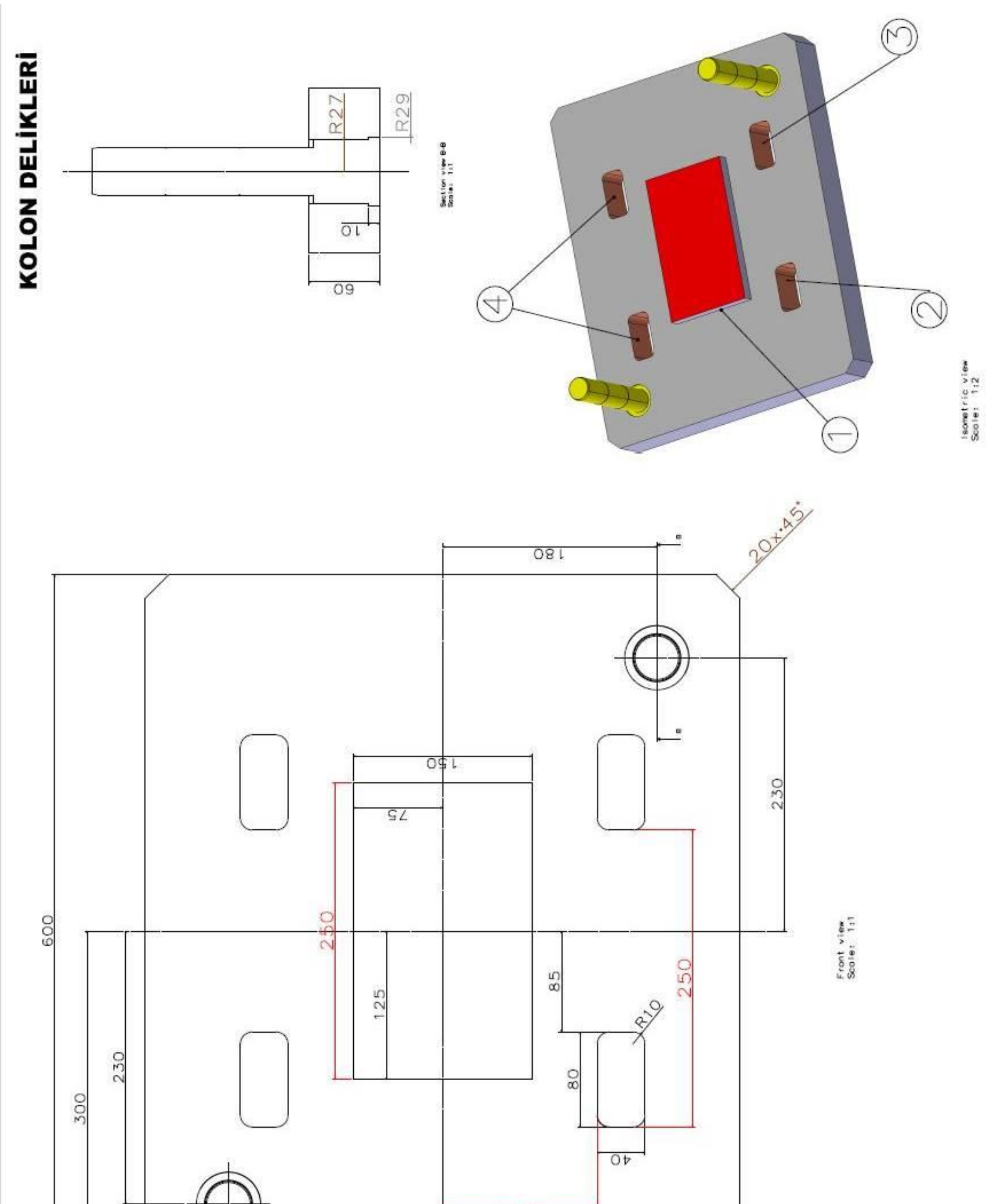
Yeterli	
Yetersiz	

Açıklama	

12. **Kullandığınız tasarım programının sahip olduğu Arayüz tasarımını kalıp tasarımcısı gözüyle değerlendiriniz.**

APPENDIX D

MODEL SAMPLE DESIGNED FOR TASKS



APPENDIX E

TASKS FOR EYE TRACKING ANALYSIS (TURKISH)

☐ *1. GÖREV*

Çizen programını başlatınız. Başlangıç ekranında bulunan 'Parça' komutu ile yeni bir çalışma sayfası açınız. xy düzleminde skeç oluşturunuz. Resimde ölçüleri verilen tablayı oluşturmak için dikdörtgen çiziniz (Eksene göre simetrik olacak şekilde). Dikdörtgeni ölçülendiriniz.

☐ *2. GÖREV*

Ölçülendirdiğiniz en ve boy büyüklüklerinin her biri için 'en' ve 'boy' isimlerinde parametre atayınız. (2 adet parametre atayacaksınız.)

☐ *3. GÖREV*

Dikdörtgen çizimini 60 mm yüksekliğinde +z yönünde katılaştırarak tablayı oluşturunuz. ‘Yakınlaştır-Uzaklaştır’ komutunu kullanarak tablayı birkaç defa mouse ile yakınlaştırıp uzaklaştırınız.

☐ *4. GÖREV*

Teknik resimde 1 numara ile gösterilen dikdörtgen parçayı çizip, 20 mm değerinde +z yönünde katılaştırınız. Bu parçayı ‘Birleştirme Operasyonu’ komutunu kullanarak tabla ile birleştiriniz.

☐ *5. GÖREV*

Resimde görülen 2 numaralı unsurun çizimini oluşturunuz.(2 numaralı unsur: 40x80 mm ölçülerinde dikdörtgen) Çizimi -z yönünde 60 mm değerinde katılaştırınız.

☐ *6. Görev*

Oluşturulan çizime ‘Doğrusal Çoğaltma’ komutunu uygulayarak teknik resimde 3 ve 4 numara ile gösterilen geometrileri oluşturunuz.(2 numaralı geometriyi doğrusal çoğaltarak 3 ve 4 numaralı geometrileri oluşturunuz.)

- *7. GÖREV*

Oluşturulmuş olan 3 adet dikdörtgen şeklindeki katıyı aynalayarak teknik resimde 5 numara ile gösterilen geometrileri oluşturunuz.

- *8. GÖREV*

Oluşturulan 6 adet geometriyi ‘Boşaltma Operasyonu’ komutu ile tabladan çıkarınız.

- *9. GÖREV*

Teknik resimde ölçüleri bulunan 2 adet kolon deliğini deliniz. (‘Kademeli Delik’ komutunu kullanmalısınız.)

- *10. GÖREV*

Resimde görülen R=10 mm değerindeki radyus ve 20x45 mm değerindeki pah operasyonlarını uygulayınız. Çizimi masaüstüne kaydediniz.

- *11. GÖREV*

Çalışmanın ‘step’ formatında Cizen_01 ismiyle dışa aktarımını yapınız. Çalışmayı kapatınız

- *12. GÖREV*

Çizen programı başlangıç ekranından bir montaj dosyası açınız. Az önce kaydettiğimiz Cizen_01 isimli çalışmayı montaj dosyası içinde açınız. Program içerisinde bulunan ‘Standart Parça Ekle’ sihirbazını açınız. Şapkalı Kademeli Kolon grubunda yer alan

Ø40x56_186 ölçülü kolondan montaj dosyası içerisine 2 adet ekleyiniz. Kolonların tablaya montajını yapınız.

- ☐ *13. GÖREV*

Teknik resimde bulunan izometrik görünüşten yardım alarak sırasıyla kırmızı, pembe ve turuncu renkli yüzeylere renk atayınız.

Kırmızı renk kodu: Red: 255, Green: 0, Blue: 0

Açık Gri renk kodu: Red: 192, Green: 192, Blue: 192

Turuncu renk kodu: Red: 234, Green: 132, Blue: 102

- ☐ *14. GÖREV*

Yardım dosyasını açınız, 'Ölçme ve Analiz Araç Çubuğu' ile ilgili bilgi edininiz. Başında üçgen şekli bulunan 3 adet ölçülendirmeyi Çizen'de ölçüm aletleriyle ölçünüz.

- ☐ *15. GÖREV*

Tasarımı 'İzometrik Görünüş' komutunu kullanarak izometrik hale getiriniz , ekran görüntüsünü alınız ve Cizen_01 adıyla kaydediniz.

- ☐ *16. Görev*

Çalışmayı kapatınız. Önünüze gelen mavi arayüz ekranında bulunan geribildirim bağlantısına tıklayarak Çizen ile ilgili görüşlerinizi bildiriniz.

APPENDIX F

POST-TEST (TURKISH)

A. PROGRAM KOMUTLARI VE ARAYÜZÜ HAKKINDA GÖRÜŞLER

1. **Arayüz yönlendirmeleri hakkında ne düşünüyorsunuz? Önerileriniz nelerdir?**

2. **Tasarım anında mouse kullanımını nasıl buldunuz? Önerileriniz nelerdir?**

Kolay	
Zor	
Açıklama	

3. **2D ve 3D ortamındaki komutlar yeterli mi? Önerileriniz nelerdir?**

Yeterli	
Yetersiz	
Açıklama ve öneriler	

4. **Model unsurlarının bulunduğu ürün ağacının içeriği ve oluşumunu nasıl buldunuz?**

5. **Parametrik modelleme yönteminin kullanımını nasıl buldunuz?**

Kolay	
Zor	
Açıklama ve öneriler	

6. **Program hakkında genel görüşleriniz nelerdir**

7. **Programın geliştirilebilir gördüğünüz yönleri nelerdir? Görüşlerinizi belirtiniz.**

B- SYSTEM USABILITY SCALE (SUS)

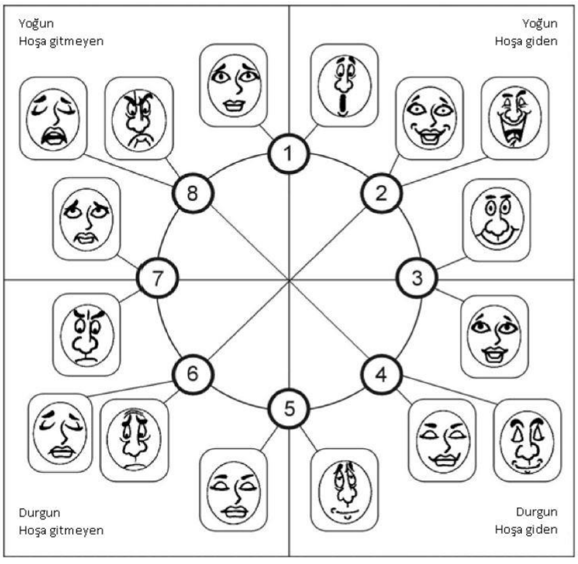
ÇİZEN programı hakkında aşağıdaki görüşleri 1'den 5'e kadar puanlandırınız

1.Strongly Disagree (Kesinlikle katılmıyorum) 5.Strongly Agree (Tamamen Katılıyorum)

		1	2	3	4	5
1	I think that I would like to use this system frequently Bu programı sıklıkla kullanacağımı düşünüyorum					
2	I found the system unnecessarily complex Programı gereksiz bir şekilde karmaşık buldum					
3	I thought the system was easy to use Programın kullanımı çok kolay					
4	I think that I would need the support of technical person to be able to use this system Bu programı kullanabilmek için teknik bir desteğe ihtiyacım olacağını düşünüyorum					
5	I found the varius functions in this system were integrated Programa çeşitli fonksiyonların entegre edildiğini gördüm					
6	I thougt there was too much inconsistency in this system Programda çok fazla tutarsızlığın olduğunu düşünüyorum					
7	I would imagine that most people would learn to use system very quickly Bu programı birçok insanın çok kısa sürede öğreneceğini düşünüyorum					
8	I found the system very awkward to use Programın kullanımını çok garip buldum					
9	I felt very confident using the system Programı kullanırken kendimi güvenli ve rahat hissettim					
10	I needed to learn a lot of things before I could get going with this system Programı kullanmadan önce birçok şeyi öğrenmeye ihtiyacım var.					

C-ÇİZEN ile ilgili duygularınızı, lütfen aşağıda görselleştirilmiş ifadeleri daire içine alarak belirtiniz.

Belirteceğiniz duygular bir veya birden fazla olabilir



APPENDIX G

USABILITY TEST NOTES FORM

Participant:

Date:

Task	Timestamp	Observation	Category (Error, success…)	Severity (1-5)
TASK 1				
TASK 2				
TASK 3				
TASK 4				

APPENDIX H

PERSONA FORM

SHEET METAL FORMING DESIGN

PERSONA FORM

Profile Attributes

Name Surname:

Age:

Job Experience:

Working Company and Capacity:

Profil: *Otomotiv sektöründe çalışan sac kalıp tasarımcısı*

Education: *Lise ve üzeri eğitimli kişiler*

Age: *21+*

Yetenekler: *CAD program kullanabilme*

APPENDIX I

DESIGN OF A MOLD SAMPLE IN SOLIDWORKS, ÇİZEN, VISICAD, CATIA PROGRAMS AND INVESTIGATION OF DESIGN METHODOLOGY

BİR DÖKÜM KALIP ÖRNEĞİNİN SOLIDWORKS, ÇİZEN, VISICAD, CATIA PROGRAMLARINDA TASARLANMASI ve TASARIMLARIN İNCELENMESİ

Gül Çiçek ZENGİN BİNTAŞ
Cengiz ÖZ
Hülya FERİK
Coşkun DEMİR

MUBİTEK TASARIM BİLİŞİM MAKİNA SAN. TİC. LTD. ŞTİ.
Rapor No: 1.1.3
Hazırlanma Tarihi: 24.09.2019

Nilüfer / BURSA
2019

ÖZET

BİR DÖKÜM KALIP ÖRNEĞİNİN SOLIDWORKS, ÇİZEN, VISICAD, CATIA PROGRAMLARINDA TASARLANMASI ve TASARIMLARIN İNCELENMESİ

ZENGİN BİNTAŞ, Gül Çiçek
ÖZ, Cengiz
FERİK, Hülya
DEMİR, Coşkun

MUBİTEK TASARIM BİLİŞİM MAKİNA TİC. LTD.ŞTİ.
Eylül 2019, 87 sayfa

ÇİZEN programının pratik, yalın, kullanışlı nitelikte kalıp tasarımını gerçekleştirebilmesi ve kullanıcı rahatlığını göz önünde bulunduran bir arayüz tasarımının oluşturulmasını sağlamak amacıyla bir çalışma başlatılmıştır. Bu çalışma kapsamında; belirlenen bir döküm kalıp örneği SOLIDWORKS, ÇİZEN, VISICAD ve CATIA programlarında farklı tasarımcılar tarafından tasarlanmış, tasarım aşamaları görüntülü olarak kayıt altına alınmıştır. Bu raporda SOLIDWORKS, ÇİZEN, VISICAD ve CATIA programlarında yapılan tasarımların detaylı bir şekilde incelemesi yapılarak, tasarım sürecinin olumlu ve olumsuz yönleri ortaya konulmuştur. Rapor sonucunda elde edilen veriler ÇİZEN programının kalıp tasarımı alanında geliştirilmesi çalışması için girdi olarak kullanılacaktır.

Anahtar sözcükler: Solidworks, Visicad, Çizen, Catia, Arayüz Tasarımı, Kalıp Tasarımı

İÇİNDEKİLER

ŞEKİLLER DİZİNİ

1. Giriş

Bu raporda, belirlenen bir döküm kalıbın Solidworks, VisiCAD, ÇİZEN ve Catia'da tasarlanması çalışması ile ilgili genel değerlendirmeler ve saptamalar bulunmaktadır. Bu rapor ile birlikte Çizen programının kalıp tasarımı alanında geliştirilmesine yönelik veriler oluşturulmuş olacaktır.

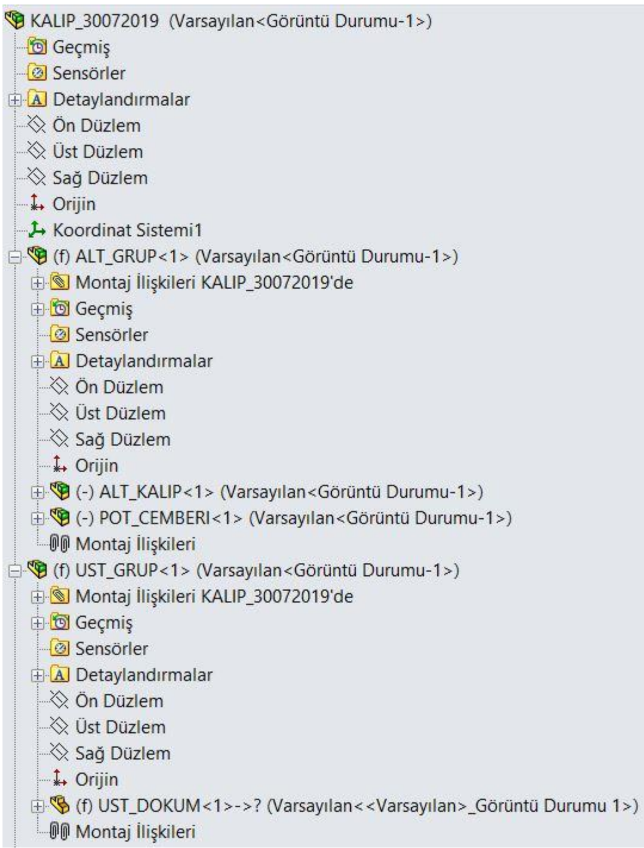
2. SOLIDWORKS

2.1 Tasarım Sürecinde İzlenen Yolun ve Tasarım Aşamalarının İncelenmesi

Tasarım süreci; ürün ağacı yapısının oluşturulması, preslerin konumlandırılması vb. aşamalara bölünerek anlatılmaya çalışılmıştır. Raporun bu bölümünde tasarım aşamaları hakkında detaylı bilgi verilmiştir.

2.1.1 Ürün Ağacı Yapısının Oluşturulması

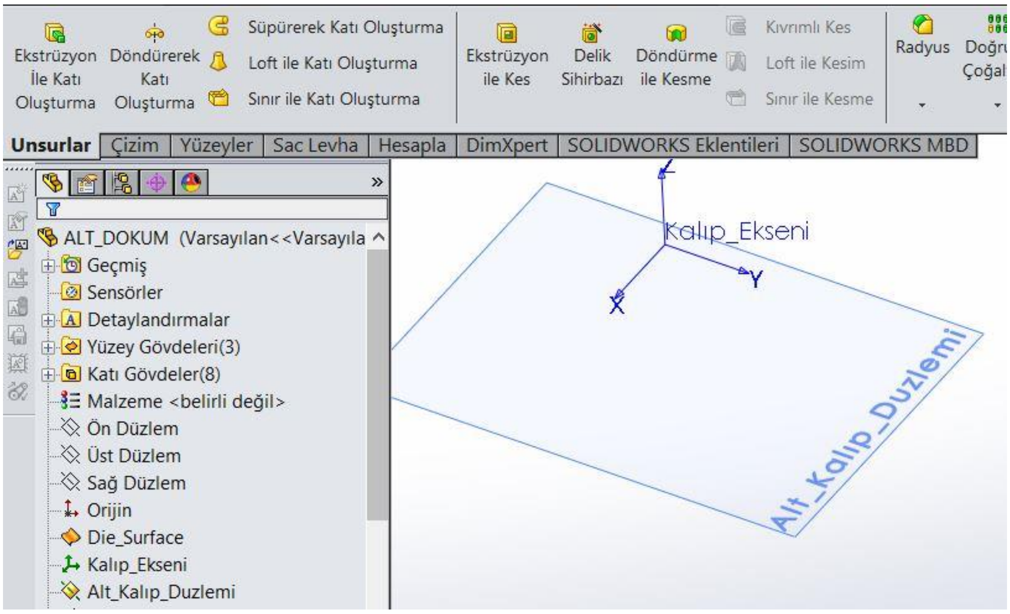
Tasarıma ürün ağacı yapısının oluşturulması ile başlanmıştır. Başlangıçta aşağıdaki resimde görüldüğü gibi döküm gruplarına göre ürün ağacı yapısı oluşturulmuştur.



Şekil 1: Tasarım Ürün Ağaç Yapısı (Solidworks)

Tasarım ilerledikçe ürün ağacına 'Referans Parça' adı verilen bir bileşen eklenmesi gerektiği görülmüştür. Kalıp eksenleri dışında tasarımsal bütün bilgilerin (kalıp ebatları, standart parçalar için oluşturulan çizimler, ortak olarak kullanılacak çizimler, tij noktaları vb.) bu parça altında bulunması ve bu bilgilerin döküm parçaları içerisine eklenmesi mantığıyla çalışılmasının Solidworks programı içerisinde mümkün olduğu görülmüştür. Bu yöntem ile çalışılması durumunda istenilen bilgilere daha kolay ulaşılması sağlanacak ve kalıp içinde çoğu bilgilerin birbirine bağımlı olmasına katkı sağlanacaktır.

Solidworks programında oluşturulan eksenlerin linkli çalışma yapılarak, döküm parçalar içerisine eklenmesi mümkün olmadığı için kalıp eksenlerini oluşturmak için her bir döküm grubu içinde alt döküm düzlemi referans alınarak düzlem oluşturulmuştur. Alt kalıp grubu içerisindeki alt döküm parçasına, pot çemberi grubunun döküm parçasına ve üst grup döküm parçasına bir adet taban düzlemi atanmış ve kalıp bu düzlemler üstünde çizilmiştir. Şekil 2'de örneği mevcuttur.



Şekil 2: Alt Döküm Parçası İçine Atanmış Düzlem

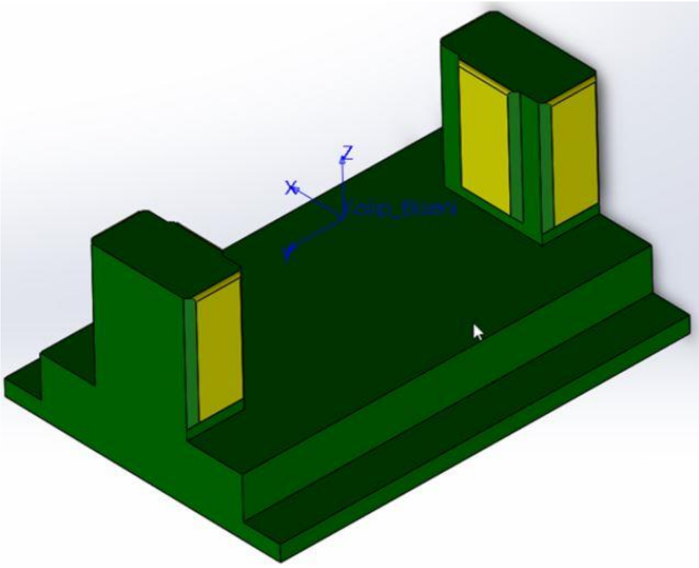
2.1.2 Döküm Gruplarının Tasarlanması

2.1.2.1 Alt Dökümün Tasarlanması

Alt döküm parçasının tasarım işlemleri gruplandırılarak açıklanmaya çalışılmıştır.

2.1.2.1.1 Döküm Tablasının Oluşturulması

Alt döküm parçası ayrı bir pencerede açılmıştır. 0.0.0 noktasına 1 adet kalıp ekseni atanmıştır. Parça içerisinde taban ve alt döküm olmak üzere 2 adet düzlem oluşturulmuştur. Bu düzlem üstüne kalıp tablası çizilmiş, olması gereken ölçülerde kule ve kızak yapıları tasarlanmıştır. Döküm içerisine 'Unsurlar → Alınmış' komutu kullanılarak parçaya ait metot dosyası yerleştirilmiştir.



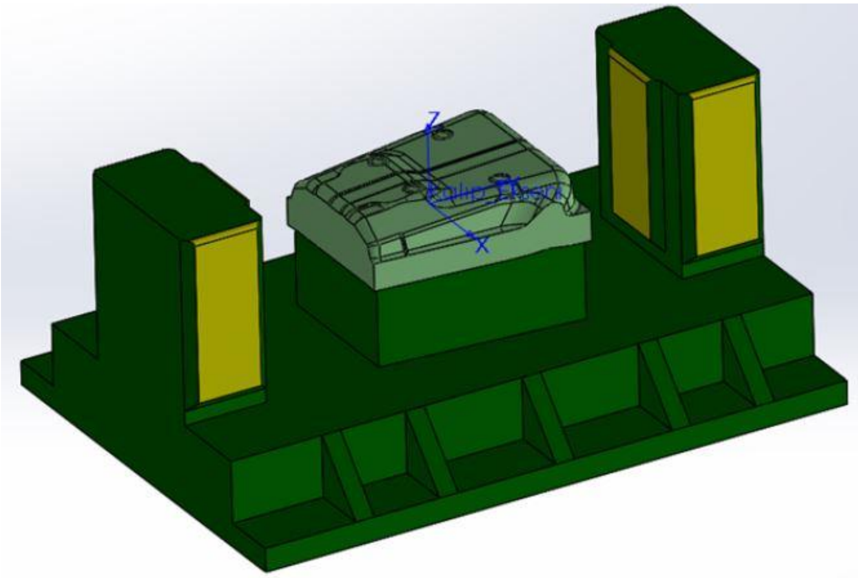
Şekil 3: Alt Döküm Tablası ve Kule Oluşumu

2.1.2.1.2 Erkek Yapısının Oluşturulması

Kalıbın erkeği olarak bildiğimiz, bölünecek yapının oluşturulması işlemine geçilmiştir. Tasarımın başlarında Catia'da ki 'Project' komutunun eşdeğerine Solidworks programında ulaşılamadığı için kalıbın konturu dikdörtgen olarak bir çizim içinde çizilmiştir. Bu yapıyı oluşturmaya altındaki destek bölümünden başlanmıştır.

- Kalıp yüzeyinin -z yönünde 60 mm ötelenmiş hali olan 'Öteleme Yüzeyi' denilen yüzeyi başka bir programda oluşturulmuştur. (Solidworks programı yüzey öteleme konusunda yetersiz kalmış, işlemi gerçekleştirememiştir.) Bu yüzey alt döküm içerisine çağırılmış ve konumlandırılmıştır.
- Bir çizim içine dörtgen geometri şeklinde geçici bir kontur oluşturulmuş, bu kontur 'Ekstrüzyon' komutu ile Öteleme Yüzeyine kadar katılaştırılmıştır. Tekrar bir çizim oluşturulmuştur. Oluşturulan konturun 400 mm içeriye doğru Öteleme Yüzeyine kadar 'Ekstrüzyon ile Kes' komutu ile boşaltılmasıyla erkek bölümünün destek kısmı oluşturulmuştur.
- Destek üstündeki erkek kısmı oluşturulmaya başlanmıştır. Bu bölümü oluştururken her iki yönde de bölme işlemi uygulanması gerektiği için gövdeler halinde çalışmak gerekti. Tasarım iki ayrı gövdeye bölündü. (Gövde mantığı; tasarım gövdelere bölünerek komutların sadece seçtiğimiz gövdeye uygulanmasına imkan tanımaktadır.)

Bir çizim içine kontur alınmış ve bu çizim katılaştırılmıştır. Oluşan katı kalıp yüzeyi ve öteleme yüzeyi ile bölünmüştür. Böylece erkek bölümü oluşturulmuştur. Kenar federlerini de oluşturduktan sonra kalıp aşağıdaki hali almıştır;

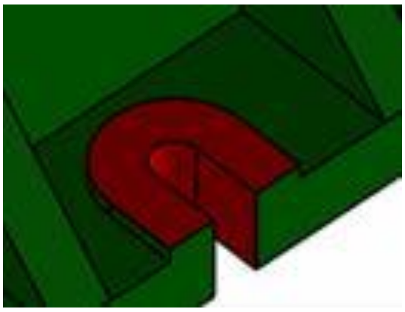


Şekil 4: Erkek Grubunun Tasarlanması

2.1.2.1.3 Pres Bağlantılarının Tasarlanması

Pres bağlantılarını oluşturma mantığı şu şekildedir; pres bağlantı noktaları adında bir çizim oluşturulmuştur. Bağlantıların eklemesi çizilmiştir, bu ekleme pres bağlantı noktaları çizimi ile çoğaltılmıştır. Aynı şekilde bağlantıların boşaltması çizilmiştir, bu boşaltı pres bağlantı noktaları çizimi ile çoğaltılmıştır. Böylece bağlantılar oluşturulmuş oldu.

Bu aşamada pres bağlantıları rastgele konumlarda oluşturulmuştur. Tasarım içine pres bilgisi alındıktan sonra konumları düzenlenecektir.

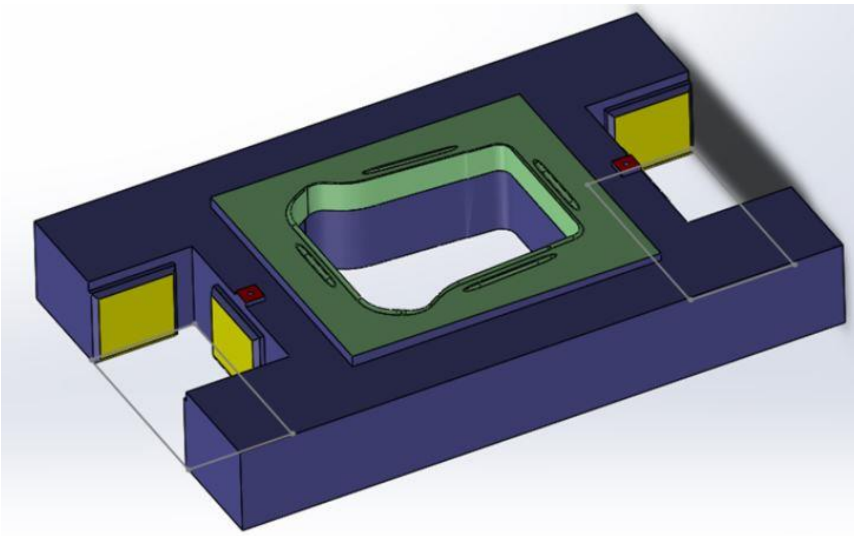


Şekil 5: Pres Bağlantısı

2.1.2.2 Pot Çemberinin Tasarlanması

- Pot Çemberi parçasına, alt döküm parçası içinde oluşturulan düzlem referans alınarak bir taban düzlemi atanmıştır.

- Alt döküm tablasının çizimi ile ilişki kurularak pot çemberi tablası oluşturulmuştur. Öteleme yüzeylerinin oluşturulması ve konumlandırılması zaman kaybına sebep olacağı için pot çemberi tablasının üstü yüzey ile bölünmemiştir.
- Alt dökümün kızak çizimleri referans alınarak pot çemberinin kızak boşaltıları oluşturulmuştur.
- Metot dosyası döküm parça içine alınmıştır. Parçanın açınımı 30 mm dışarı yöne ötelenerek oluşturulan çizim ekstrüzyon ile katılaştırılmış ve yüzey ile bölme işlemi uygulanmıştır.
- Döküm tablanın alt boşaltması için çizim oluşturulmuş ve Öteleme Yüzeyine kadar boşaltılmıştır. Metot dosyası içindeki kontur ile bir çizim oluşturarak tabla boşaltılmıştır. Kontur 7 mm dış yöne ötelenerek bir çizim oluşturulmuş, bu çizim 40 mm kalınlığında ekstrüzyon ile öteleme yüzeyine kadar katılaştırılmıştır. Böylece pot çemberinin balkon kısmı oluşturulmuştur. (Konturun 10 mm ötelenmesi gerekirdi fakat program bu işlemi gerçekleştiremedi.)
- Referans elemanları ve sürtünme plakalarının da tasarımı yapıldıktan sonra pot çemberi genel hatlarıyla oluşturulmuştur.



Şekil 6: Genel Hatlarıyla Pot Çemberi

2.1.2.3 Üst Dökümün Tasarlanması:

Pot Çemberinin oluşturulmasında olduğu gibi alt düzlem referans alınarak 2 adet düzlem oluşturularak tasarıma başlanmıştır. Alt döküm tablasının çizimi ile ilişki kurularak tabla oluşturulmuştur. Sırasıyla kızak boşaltıları, sürtünme plakaları, referanslama elemanları, mapalar için yuvalar, döküm federleri, pres bağlantılarının bulunacağı kısım tasarlanmıştır.

2.1.2.3.1 Pres Bağlantılarının Tasarlanması

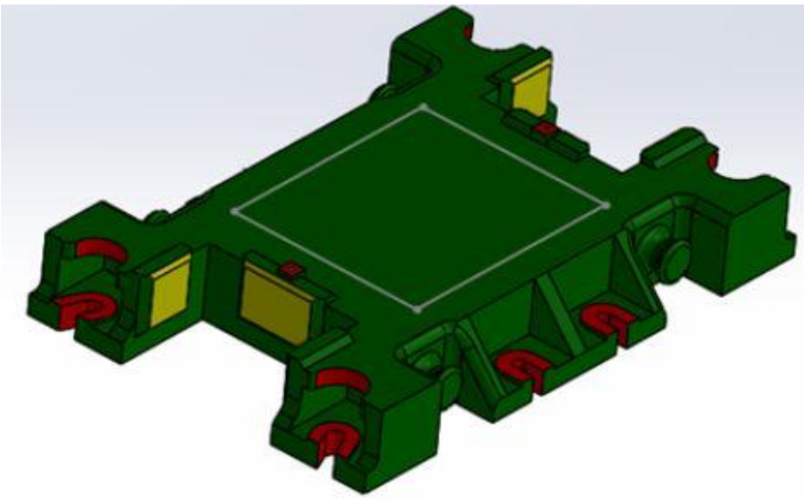
Üst dökümde 2 farklı pres bağlantısı bulunmaktadır. Bağlantılardan biri açılı diğeri düzdür. Kalıbın sol ve sağında bulunan açılı pres bağlantısı için bütün ekleme ve çıkarmalar tek tek çizilmiştir. Alt dökümdeki bağlantı noktalarında olduğu gibi çizim ile çoğaltma yöntemi uygulanmamıştır.

Kalıp önü ve arkasında bulunan düz bağlantılar alt döküm bağlantılarında uygulanan yöntemde olduğu gibi çoğaltılarak tasarlanmıştır.

2.1.2.3.2 Mapa Eklenmesi

Mapaların tasarlanması için 2 adet aynalama ekseni ve 1 adet mapa ekseni çizilmiş, mapa tasarımı yapıldıktan sonra aynalayarak çoğaltılmıştır.

Mapalar eklendikten sonra tablanın alt boşaltısı oluşturulmuştur.



Şekil 7: Genel Hatlarıyla Üst Döküm

2.1.3 Döküm Gruplarında Eksik Kalan Kısımların Tamamlanması

Döküm grupları genel hatlarıyla oluşturulduktan sonra tasarımda eksik kalan kısımlar tamamlanmaya çalışılmıştır. Uygulanan işlemler şu şekildedir;:

- Alt döküm içerisinde; kalıp kulelerinin üst boşaltıları, referanslama elemanları, üst dökümde olduğu gibi taşıyıcı eleman olan mapalar ve mapaların boşaltıları oluşturulmuştur.
- Ürün ağacına ‘Reference_Part’ isimli yeni bir bileşen eklenmiştir. (Tasarımın başında böyle bir bileşen oluşturup tasarımsal bütün bilgiler bu bileşenin içinde bulunmalıydı.) Bu bileşen içinde kalıp ebatları adıyla bir çizim oluşturulmuş, bu çizimle ilişkilendirerek alt dökümde ve pot çemberinde ortak olarak kullanılacak oturma eklemelerinin çizimi oluşturulmuştur. Bu çizim alt döküm içine alınıp katılaştırılmış ve oturma elemanları oluşturulmuştur.

- Pot Çemberi döküm parçası ayrı bir pencerede açılmıştır. Alt döküm parçasında olduğu gibi oturma elemanları, mapalar, pot çemberi üstündeki boşaltmalar oluşturulmuştur.

2.1.4 Standart Parçaların Tasarıma Eklenmesi

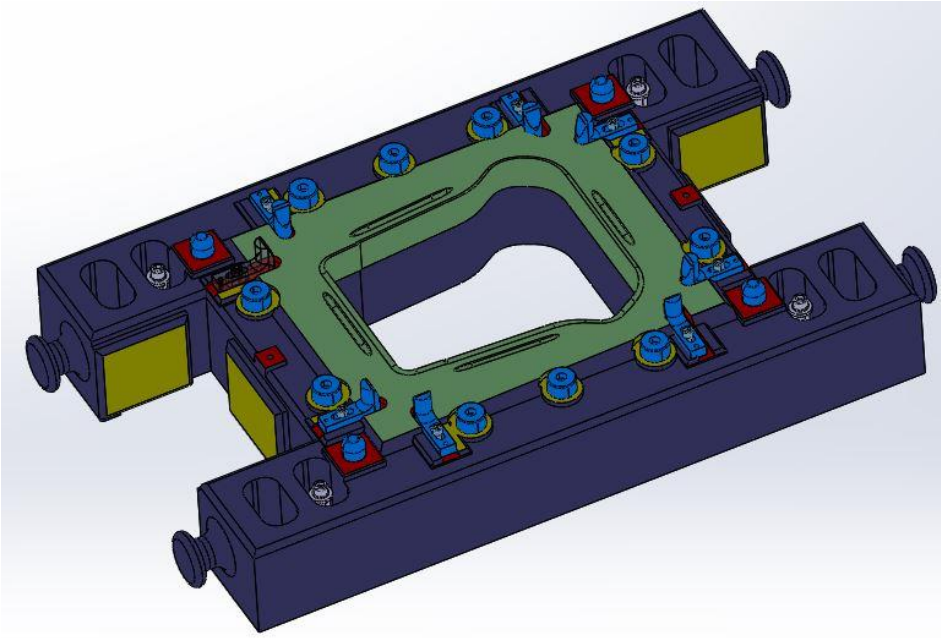
Döküm grupları genel olarak oluştuktan sonra standart parçalar Aç komutu ile program içinde açılmış ve tasarım içerisine kopyalanmıştır.

Tasarımda olması gereken; her bir standart parçanın ilgili döküm montajlarının içerisinde bulunmasıdır. Örneğin pot çemberinde yer alan standart parça ürün ağacında pot çemberi montajının içinde yer almalıdır. Bu tasarım çalışmasında bütün standart parçalar kalıbın montaj dosyasının içerisinde bulunmaktadır.

Tasarımda ilgili parçanın konumlarının yer aldığı çizimler Reference_Part bileşeni içinde çizilmiştir. Her bir alınan standart parça içerisine koordinat sistemi eklenerek Reference Part içindeki çizimi ile parça arasında montaj ilişkisi eklenerek konumlandırma yolu izlenmiştir. Her bir standart parça ilişkilendirildikten sonra eklemeleri, boşaltmaları, delikleri manuel olarak, çizimler referans alınarak oluşturulmuştur.

Tasarım içine alınan parçalar aşağıdaki gibidir;

- Sac dayama parçası
- Stoper
- Merkezleme Pimi
- Askı Cıvatası
- Lazer Parçası



Şekil 8: Standart Parçaların Döküme Eklenmesi

2.1.5 Tasarımsal Eksiklerin Tamamlanması

Standart parçalar tasarıma eklendikten sonra tasarımsal eksikliklerin olduğu görülmüştür ve bunlar tamamlanmaya çalışılmıştır;

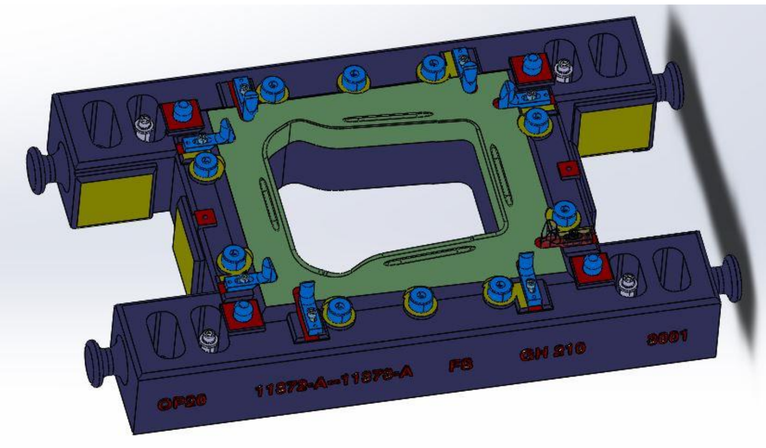
- Pres dataları bu aşamada tasarım içerisine alınmadığı için alt dökümdeki tij delikleri ve pot çemberinde ki tij destek eklemeleri için rastgele ölçülerde bir çizim oluşturularak tasarlanmıştır. (Presler konumlandıktan sonra çizim içinde gerekli düzenleme yapılacaktır.)
- Stoper konumları belirlendikten sonra alt döküm yüzeyinde bulunacak stoper destek eklemeleri eklenmiştir. Bu eklemelerin pot çemberindeki karşılıkları pot çemberine eklenmiştir.

2.1.6 Kalıp Yazılarının ve Oklarının Eklenmesi

Tasarım içerisine yazıların ve okların eklenmesi işleminde izlenen yol aşağıdaki gibidir;

- Kalıp tasarımında kullanılan Dxf dosyası Solidworks programı içerisinde açılmıştır.
- İlgili yazılar kopyalanarak döküm parçası içerisinde açılmış çizim içine yapıştırılmıştır. Yazı tam konumunda gelmediği için 'Taşı-Döndür' komutu yardımıyla konumlama yapılıp fazla çizgiler temizlendikten ve ölçülendirme yapıldıktan sonra oluşturulan çizim katılaştırılmıştır.

- Bu şekilde işlem tamamlandıktan sonra Solidworks programının içerisinde bir ‘Çizim Metni’ komutu olduğu görülmüştür. Bu komut ile kalıp yazılarının eklenmesi daha pratik bir şekilde yapılmıştır.



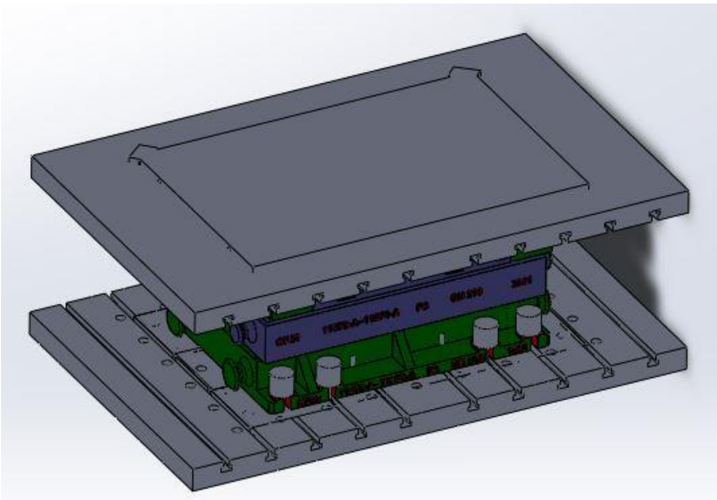
Şekil 9: Kalıp Yazılarının ve Okların Tasarıma Eklenmesi

2.1.7 Preslerin Yerleştirilmesi

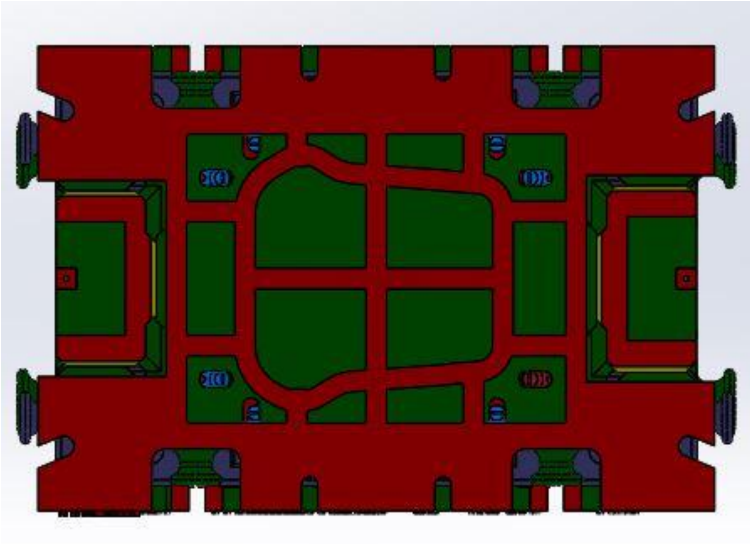
Pres dosyaları Catia dosyası olarak kalıp içerisine alınmış ve konumlandırılmıştır. Pres bilgilerine göre bağlantı noktaları düzenlenmiştir.

Pres bilgilerine göre tij noktaları Reference_Part bileşeni içinde oluşturulmuş, bu çizim referans alınarak alt dökümdeki ve pot çemberinde ki tij milleri boşaltıları ve eklemeleri düzenlenmiştir.

Standart parçalar kalıp içerisine yerleştirildikten sonra döküm tablalarının alt boşaltılarında bulunacak federlerin düzenlemesi yapılmıştır.



Şekil 10: Preslerin Konumlanması



Şekil 11: Üst Döküm Parçası Feder Yapısı

2.2 Tasarımda Kullanılan Komutların Saptanması Ve Kullanım Sıklığı Fazla Olan Komutların Belirlenmesi

Belirlenen bir döküm kalıbın Solidworks programında tasarımının yapılması çalışmasında kaydedilen videolar incelenmiştir. Tasarım esnasında kullanılan bütün komutlar saptanmıştır. Bu komut listesi aşağıdaki gibidir;

- Bileşen Ekle (Parça, Montaj) (Montaj)
- Montajı Yeniden Adlandır
- Parçayı Yeniden Adlandır
- Montajı Düzenle – Montaj Üstünde Üst Komut Grupları Sekmesi
- Parçayı Düzenle - Parça Üstünde Üst Komut Grupları Sekmesi
- Çizimi Düzenle – Çizim Üstünde Üst Komut Sekmesi
- Farklı Kaydet (Dosya)
- Aç
- Eksen
- Unsurlar → Alınmış
- Çizim
- Dikdörtgen
- Akıllı Ölçülendirme
- Ekstrüzyon İle Katı Oluşturma
- Görünüm Komutu (Renklendirme)
- Çizgi

- Görünümü Kopyala
- Görünümü Yapıştır
- Radyus
- Pah
- Yüzey ile kes
- Yüzeyi buda
- Objeleri Ötele
- Yüzey Öteleme
- Gövde Taşı/Kopyala
- Ayır (Ekle → Unsurlar → Ayır)
- Ekstrüzyon ile Kes
- Aynalama (Çizim)
- Merkez Çizgisi
- Nokta (Çizim)
- Yuva (Çizim)
- Doğrusal Çoğaltma → Çizim İle Çoğaltma
- Çakışık (İlişki)
- Delik Sihirbazı (Unsurlar)
- Unsuru Düzenle
- Buna Dik
- Gösterir
- Gizler
- Simetri (İlişki)
- Yatay (İlişki)
- Dikey (İlişki)
- Tümünü Kaydet (Dosya)
- Çizim Düzlemini Değiştir
- Daire
- Düzlem
- Unsur Özellikleri (parça adının değiştirilebileceği bölüm)
- Radyus Çiz (Çizim)
- Döndürerek Katı Oluşturma

- Aynalama (Unsurlar)
- Ölçüm
- Ekle → Parça
- Montaj İlişkisi
- Blok Oluştur (Çizim)
- Taşı (Çizim)
- Döndür (Çizim)
- Çizim Metni (Çizim)
- Şeffaflığı Değiştir

Videolar incelenip değerlendirildiğinde sıklıkla kullanılan 5 komutun aşağıdakiler olduğu görülmektedir:

1. Ekstrüzyon ile Katı Oluşturma
2. Akıllı Ölçülendirme
3. Ekstrüzyon İle Kes
4. Dikdörtgen Geometrisi Oluşturma Komutu
5. Alınmış Komutu (Dışarıdan Kontur, Yüzey alma komutu)

2.3 Tasarım Sürecinde Kullanılan Modüller

Tasarım sürecinde Solidworks programı içindeki Çizim , Montaj, Hesapla, Yüzeyler, Unsurlar modülleri kullanılmıştır.

Şekil 12: Solidworks Modüller

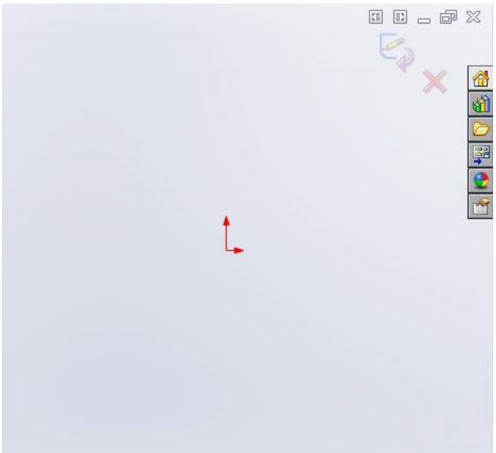
- Çizim modülü; tasarlamak istenilen elemanın 2 boyutlu geometrisinin çizilmesinde,
- Montaj modülü; ürün ağacı yapısını oluştururken bileşen eklenmesinde, bileşenin taşınmasında, döndürülmesinde, parçaların çoğaltılmasında, standart elemanların konumlanması için ilişki tanımlanmasında, referans geometrilerin oluşturulmasında (çizgi, nokta, eksen takımı, düzlem),
- Hesapla modülü; parça üstünden ölçü alınmasında,
- Yüzey modülü; kalıbın erkeği ve dişisini oluştururken bölme işleminde, parça yüzeylerini budama işlemlerinde,

- Unsurlar modülü; 2 boyutlu çizimin 3 boyutlu hale getirilmesinde, radyus, pah kırma, çoğaltma işlemlerinde, referans geometrisi eklenmesinde kullanılmıştır.

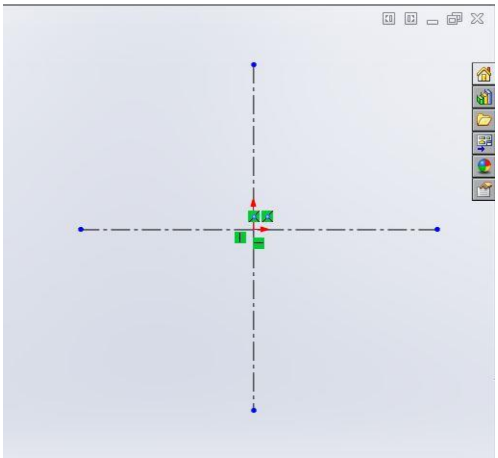
2.4 Tasarım Sürecinde Zorlanılan Noktaların Saptanması Ve Öneriler

Solidworks programı ile yapılan tasarım çalışmasında bazı noktalarda zorlanılmış ya da tasarım esnasında bir takım öneriler getirilmiştir. Tasarım videolarının incelenmesi sonucunda ortaya konulan saptamalar aşağıda açıklanmaya çalışılmıştır. Bu saptamalar, kullanılan Solidworks programının değerlendirilmesi, tasarımcı gözüyle eleştriler, komutların kullanım kolaylığının değerlendirilmesi, kalıp tasarımında gerekli özelliklerin saptanması vb. durumların değerlendirilmesiyle ortaya konulmuştur.

1. Tasarımın yapıldığı Solidworks programında Çizim ortamındayken otomatik olarak atanan çizim okları program içerisinde ölçülendirme, aynalama vb. işlemler için kullanılamamıştır. Her bir çizimde ayrı olarak eksen çizgileri oluşturulmak durumunda kalmıştır.

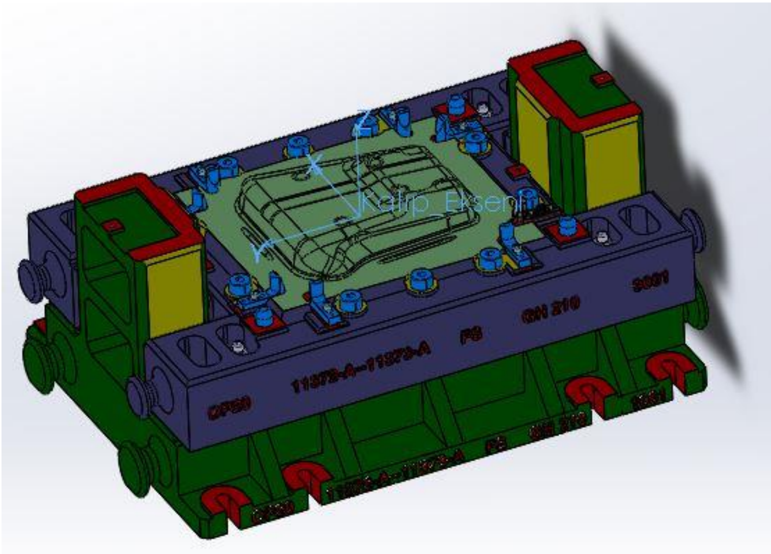


Şekil 13: Çizim Modülü İçindeki Eksen

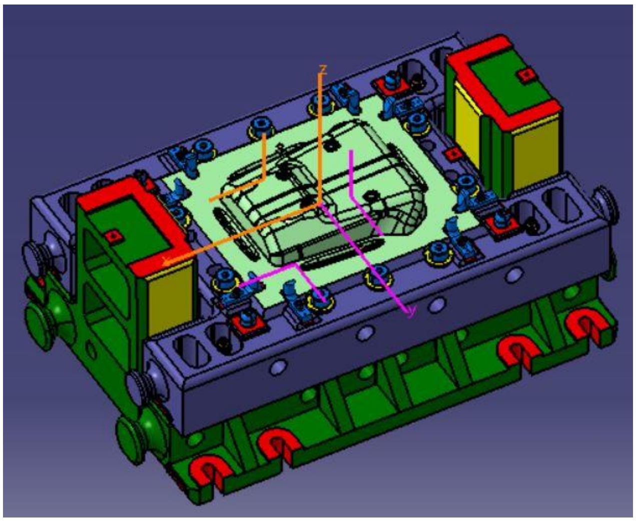


Şekil 14: Çizim Modülü İçindeki Eksenin Merkezine Çizgi Atanması

2. Yukarıdaki saptama ile benzer olan bir diğer durum ise şudur; kalıp tasarımı gereği parçalar için koordinat sistemleri oluşturulması gerekmektedir. Solidworks programı bu eksenlerin kullanılmasını sınırlandırmıştır. Ölçü alma, aynalama vb. işlemler bu koordinat sistemleri üstünden yapılamamaktadır. Bu durum kalıp tasarımında hata yapılmasını kolaylaştırmakta, ölçü alınmasını zorlaştırmaktadır. Dolayısıyla tasarım işlemlerini zorlaştırmaktadır.

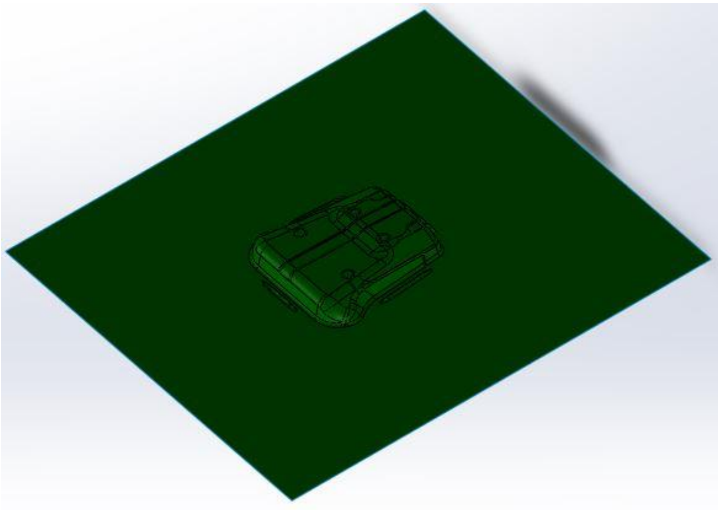


Şekil 15: Solidworks Programı İçindeki Kalıp Koordinatı

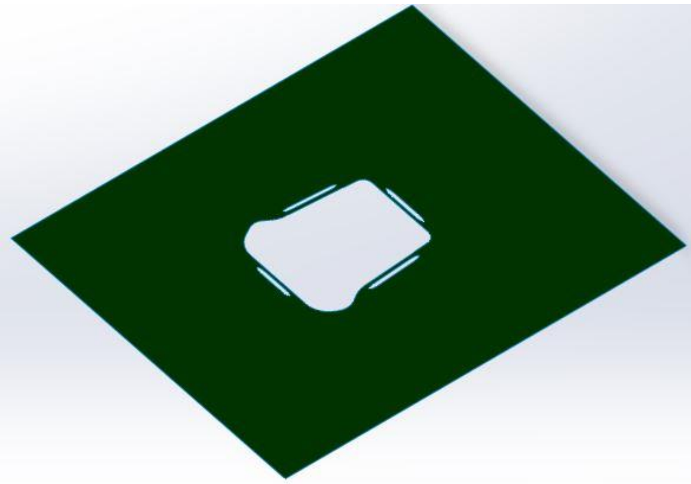


Şekil 16: Catia Programı İçindeki Kalıp Koordinatı

3. Solidworks programında kalıp tasarımı için gerekli olan öteleme yüzeylerinin oluşturulması için kullanılan ‘Yüzey Öteleme’ komutu yetersiz kalmıştır. Tasarım sırasında öteleme yüzeyleri başka bir programda oluşturulup kalıp içine alınmıştır. Kalıp tasarımını gerçekleştirme amacına odaklanmış bir programın yüzey öteleme komutunun yeterli düzeyde çalışır olması gerekmektedir.



Şekil 17: Ötelenmesi Gereken Kalıp Yüzeyi

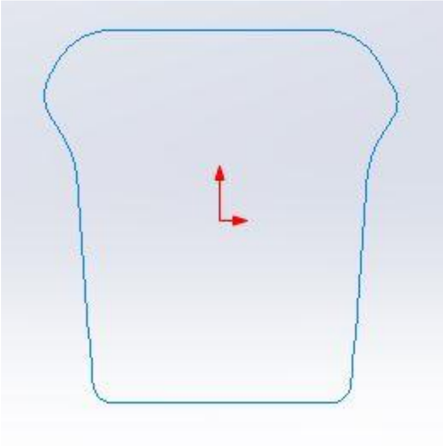


Şekil 18: Solidworks Programının Ötelediği Bozuk Yüzey

4. Farklı tasarım programlarının sağladığı bir özellik olan tasarımla ilgili referans çizimler, noktalar, eksenler, düzlemler, yüzeyler vb. elemanların yer aldığı bir bileşen tipine Solidworks programında ulaşılamamıştır. Bu sebeple bu bilgiler ürün ağacında dağınık olarak bulunmaktadır. Bu durum ileride tasarıma müdahale edilmesi gerektiğinde işlem yapılmasını zorlaştıracaktır. (Catia programında Geometrical Set adı verilen bir bileşen tipi bulunmaktadır.)

Şekil 19: Ürün Ağacı İçinde Dağınık Konumlarda Bulunan Tasarım Elemanları

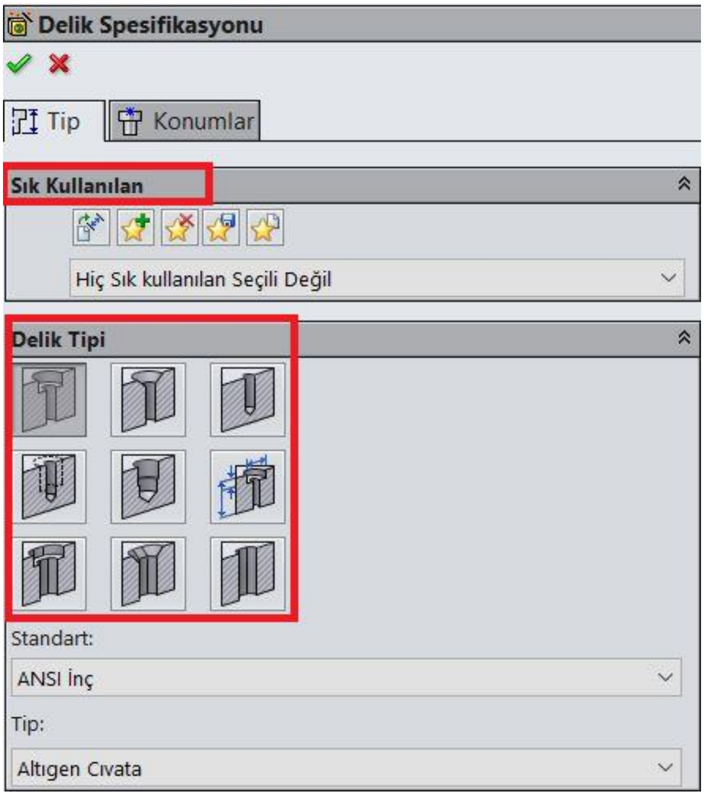
5. Solidworks programında linkli çalışma sağlanabilecek bir komuta ulaşılamamıştır. Bu sebeple kalıp revizyona uğradığında değişikliklerin uygulanması uğraştırıcı olacaktır.
6. Tasarım esnasında çizim modülü içerisinde çalışırken kalıp tasarımının oluşturulması için bütün dökümlerde parça konturunun belirli değerlerde ‘Objeleri Ötele’ komutu ile ötelenmesi gerekmektedir. Program, konturun istenen değerde ötelenmesini sağlamamıştır. Öteleme değerinin büyük olduğu uyarısını vermiştir.



Şekil 20: Ötelenecek Kontur

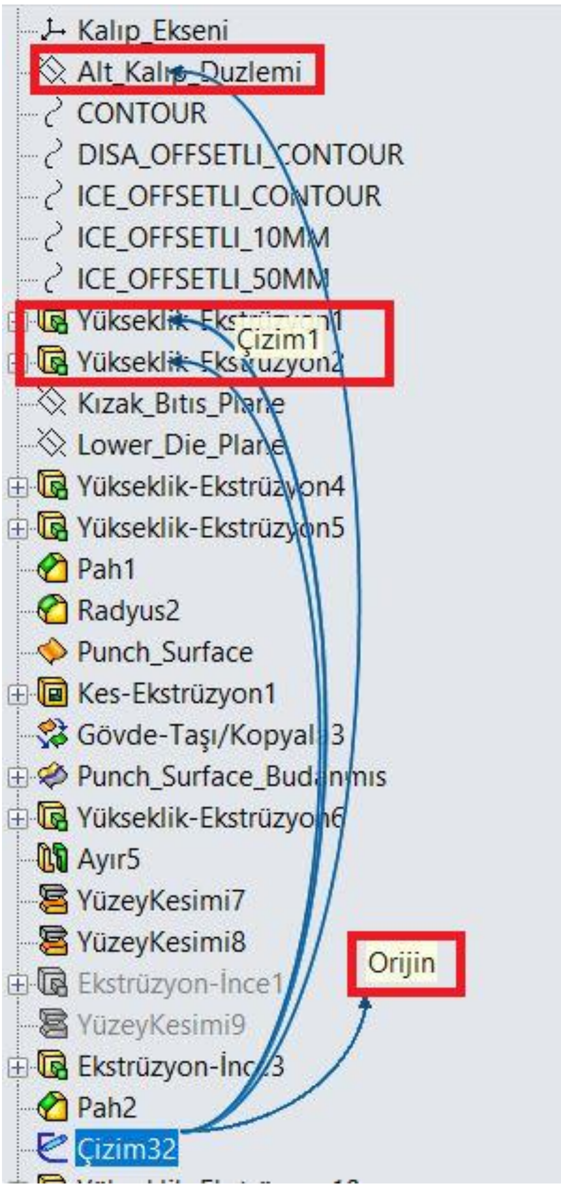
Şekil 21: Öteleme Hata Ekranı

7. Solidworks programının ‘Delik Sihirbazı’ komutu ile, sunulan görsellik açısından ve sık kullanılan sekmesi özelliği ile birlikte delik delme işleminin mevcut tasarım programlarına göre daha işlevsel olduğu kanısına varılmıştır.



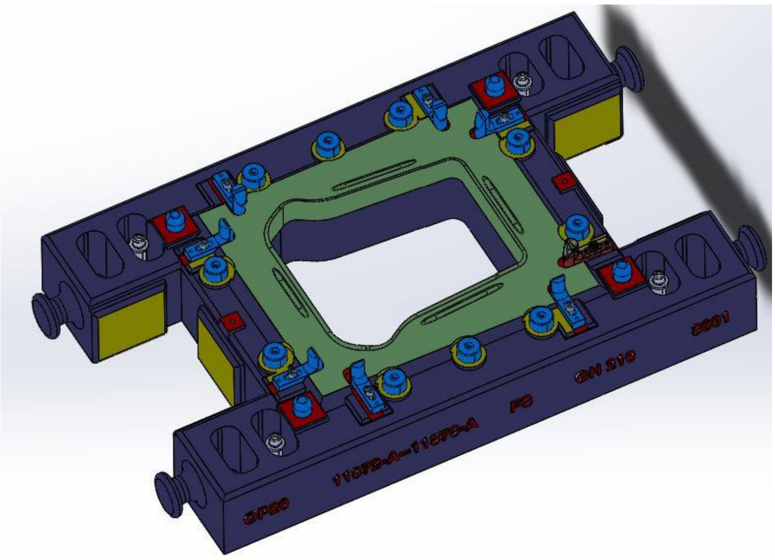
Şekil 22: Solidworks Delik Delme Sihirbazı

8. Program ürün ağacında üstüne fare imleci ile tıkladığımız unsurun geçmişini gösteren (hangi bileşenler referans alınarak oluşturulduğu bilgisi) oklar belirmektedir. Bu durumun tasarımda kolaylık sağladığı görülmüştür fakat başka bir bileşen içinden referans alındığında bu referansa ulaşmak kolay olmamaktadır.



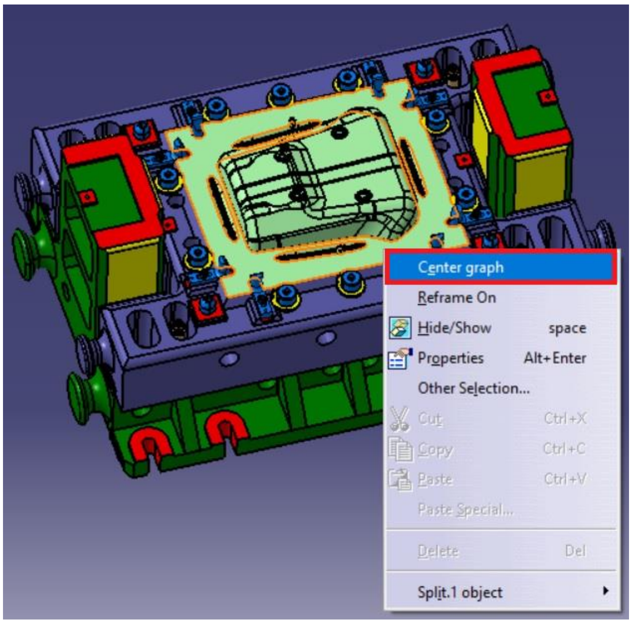
Şekil 23: Unsur Geçmişinin Ürün Ağacında Gösterilmesi

9. Tasarım içerisine standart parçaların eklenmesi aşamasında programda linkli çalışma sağlanamadığı için vakit kaybına sebep olmuştur. Her bir standart parçanın ekleme, frezeleme, delik unsurları döküm gruplarının içinde tek tek çizilmiştir. Tasarım revizyona uğradığında bu durumun kontrolü zor olacaktır.

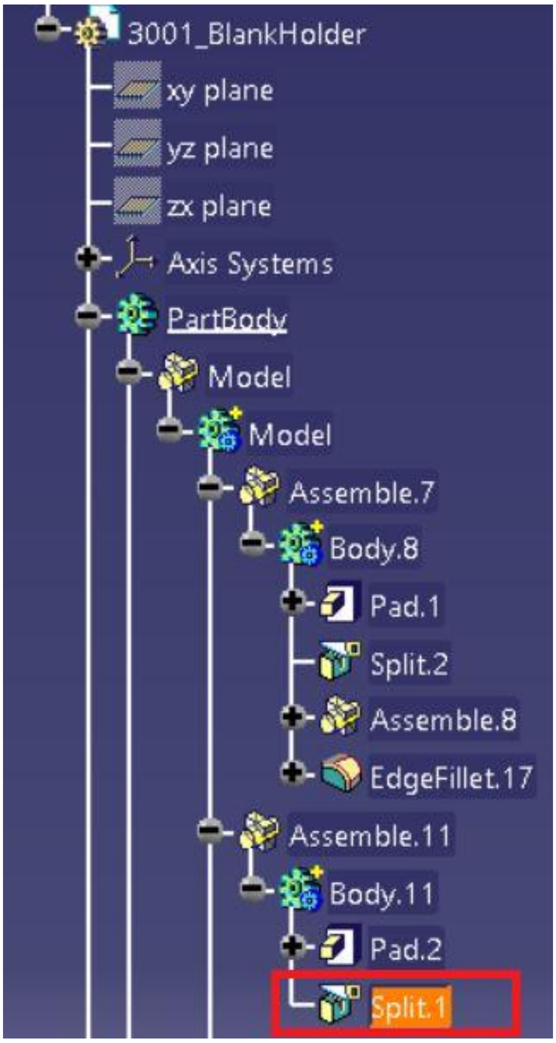


Şekil 24: Pot Çemberi Parçası İçindeki Standart Parçaların Ekleme, Frezeleme Unsurları

10. Katı model üstünde herhangi bir unsura tıklandığında yapılan yönlendirme ile ürün ağacında ilgili operasyonun yerini bulan komutun bulunmadığı görülmüştür. Bu durum tasarım sürecini negatif yönde etkilemiştir.



Şekil 25: Catia Programı İçindeki Center Graph Komutu



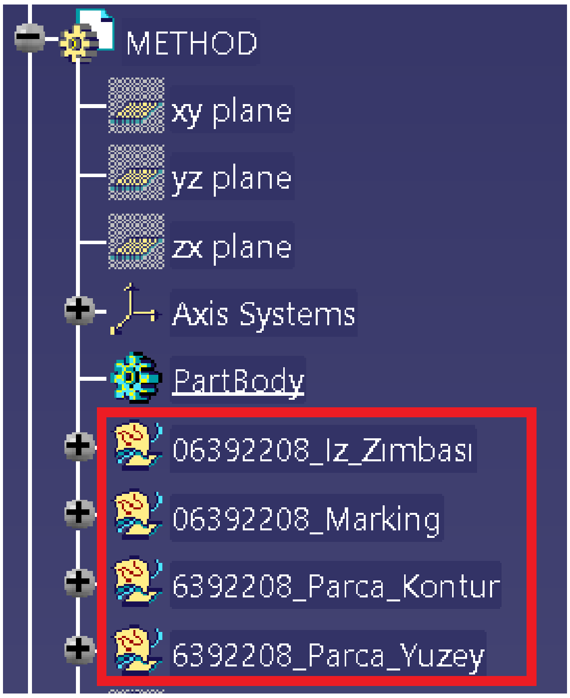
Şekil 26: Catia Programında 'Center Graph' Komutu İle Bulunan İlgili Operasyon

2.5 Tasarım Çalışmasına Revizyon İşlemlerinin Uygulanması

Raporun bu bölümünde kalıp tasarımına revizyon uygulanmak istenmesi ya da değişiklik yapılması ihtiyacı hissedildiği durumlarda tasarımcı tarafından değişikliklerin uygulanma kolaylığı değerlendirilmiştir.

Kalıp tasarımında kullanılan standart parçaların kalıba montajını ve konumlaması sağlayan bir çok ekleme, frezeleme, bağlantı delik bileşenleri bulunmaktadır. Çalışmada linkli çalışma esas alınmadığı için herhangi bir değişiklik durumunda tek tek işlem yapmak ve değişiklikleri uygulamak zaman kaybına sebep olacaktır.

Değişikliklerin uygulanmasını zorlaştıracak diğer bir konu ise tasarım içerisinde dağınık çalışılması olmuştur. Tasarımda yardımcı elemanların (çizimler, düzlemler, noktalar ,yüzeyler vb.) bir arada bulunmasını sağlayacak bir bileşen bulunmaması ve yardımcı elemanların kalıp içerisinde dağınık olarak yer alması değişikliklerin uygulanmasını zorlaştıracaktır. Örnek olarak başka bir programda bu elemanların düzenlenme şekli aşağıdaki gibidir;



Şekil 27: Catia Programı İçinde Geometrical Set Bileşeni Altında Tasarımsal Elemanların Gruplandırılması

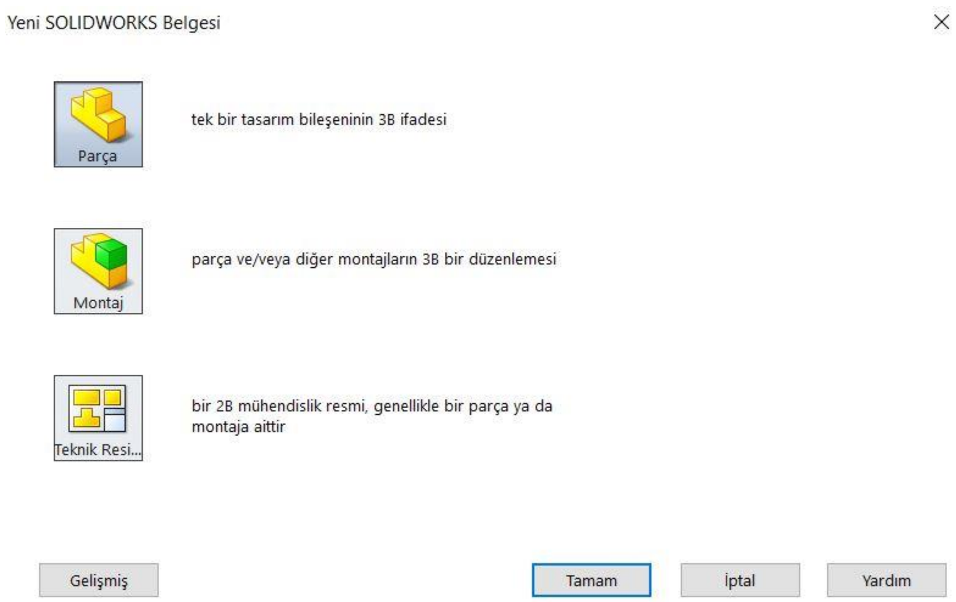
Örnekte olduğu gibi bir düzenleme yapıldığında değişikliklerin uygulanması kolaylaşacaktır.

2.6 Solıdworks Programının Görsel Ve Kullanım Kolaylığı Yönleri Ele Alınarak Değerlendirilmesi

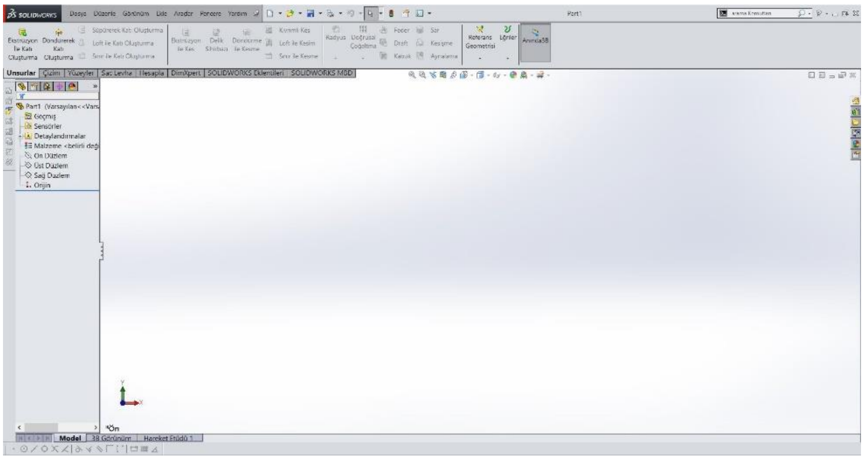
Solidworks programının kullanıcı gözüyle değerlendirildikten sonra varılan düşünceler aşağıda maddeler halinde sunulmuştur;

- Programın; pratik, anlaşılabilir, kullanımı kolay bir arayüze sahip olduğu görülmüştür. Çizim ortamına giriş, çıkış, komutların onaylanması, komut pencerelerinin yönlendirmeleri yeterli ve anlaşılır bulunmuştur. Komut ikonları görsel açıdan yeterli ve işlevleriyle uyumludur.

Şekil 28: Solidworks Programı Giriş Ekranı



Şekil 29: Solidworks Programı Bileşen Açma Ekranı (Parça, Montaj, Teknik Resim)



Şekil 30: Solidworks Programı Parça Ortamı

- Modüller arası kolay geçiş sağlanabilmesinin artı bir özellik olduğu görülmüştür.

Şekil 31: Solidworks Programı Modül Geçişi

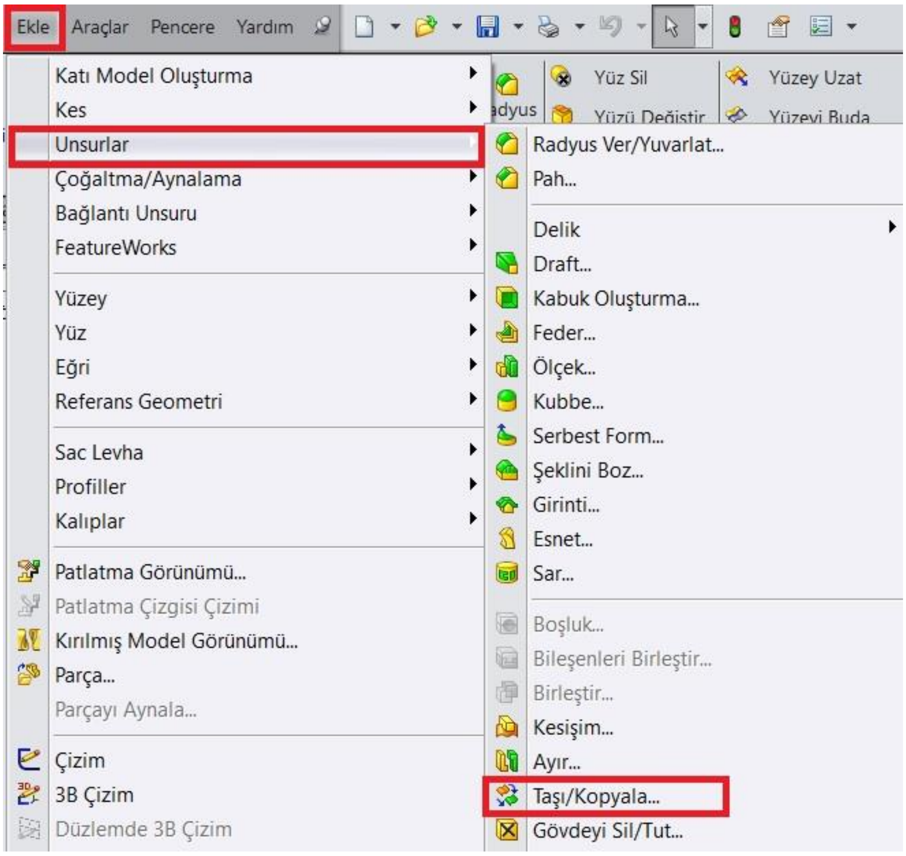
- Ürün ağacı yapısı ve görselliği kullanışlıdır.

Şekil 32: Solidworks Programı Part Bileşeninin Ürün Ağacı Yapısı

- Ürün ağacında komutlar arasında gruplandırma yapılamadığı görülmektedir. Bu durum dağınık bir ürün ağacının ortaya çıkmasına sebep olmuştur.

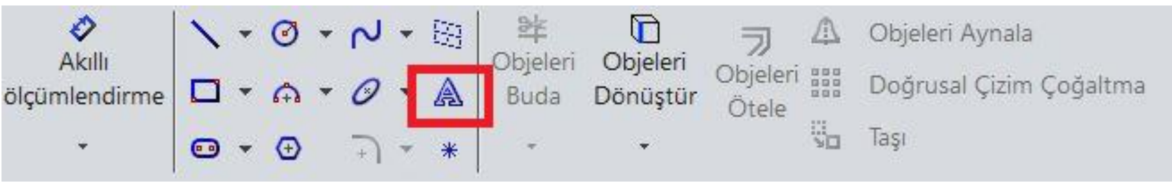
Şekil 33: Üst Döküm Parçasının Ürün Ağacı

- Programda eklenen parçaların konumlandırılması için herhangi taşıma elemanı bulunmamaktadır ve montaj içinde taşıma döndürme komutlarının kullanımı işlevsel değildir.



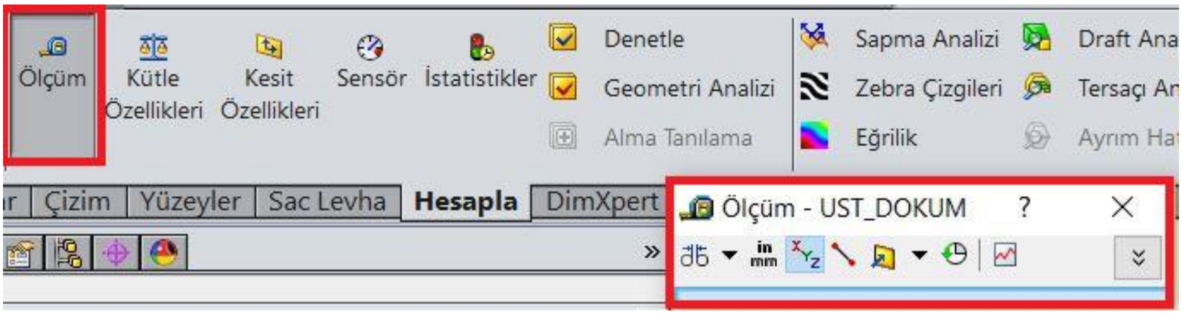
Şekil 34: Taşı/Kopyala Komutu

- üzey komutlarının ve linkli çalışmanın sağlanması için gereken komutlar konusunda programın yetersiz kaldığı kanısına varılmıştır.
- Programda kalıp yazılarının oluşturulması için ayrı bir komut vardır. Böylelikle tasarım aşamasında yardımcı dosyalar kullanılması ihtiyacı ortadan kalkmıştır.



Şekil 35: Kalıp Yazılarının Oluşturulduğu Çizim Metni Komutu

- Programda kalıp üstünde ölçü alınmasını sağlayan komut sadece 'Ölçüm' komutudur. Bu komut kullanışlı bulunmamıştır.



Şekil 36: Ölçüm Komutu

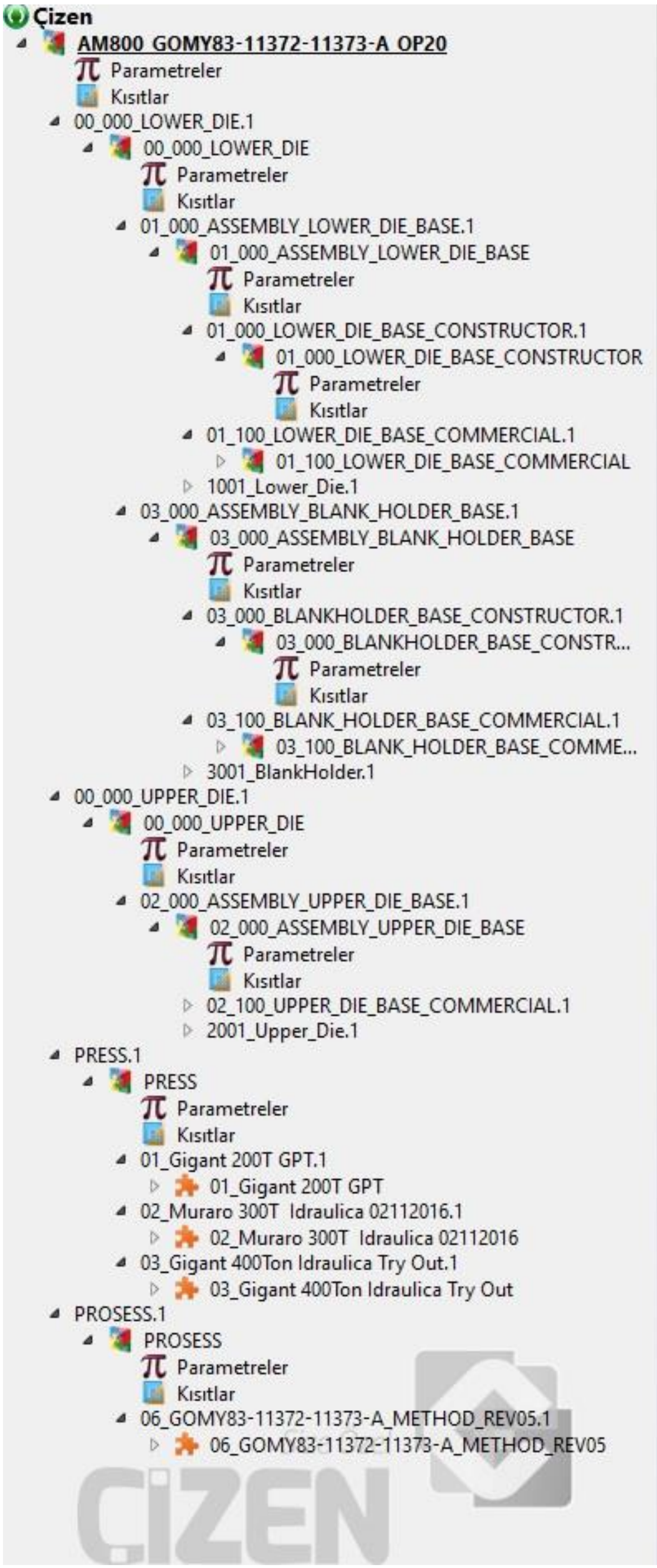
3. ÇİZEN

3.1 Tasarım Sürecinde İzlenen Yolun Ve Tasarım Aşamalarının İncelenmesi

Tasarım süreci; ürün ağacı yapısının oluşturulmasından başlayıp, preslerin konumlandırılması, döküm grupların tasarlanması vb. aşamalara bölünerek gerçekleştirilmeye ve anlatılmaya çalışılmıştır. Raporun bu bölümünde tasarım aşamaları hakkında detaylı bilgi verilmiştir.

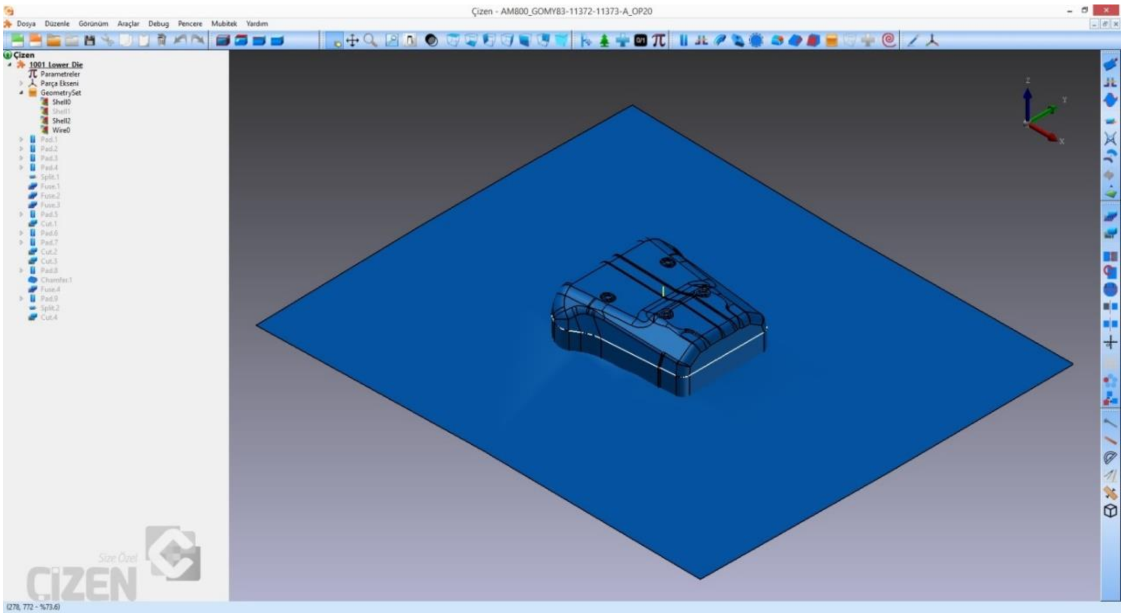
3.1.1 Ürün Ağacı Yapısının Oluşturulması

Tasarıma, tasarım ürün ağaç yapısının oluşturulması ile başlanmıştır. Başlangıçta aşağıdaki resimde görüldüğü gibi döküm gruplarına göre tasarım ürün ağaç yapısı oluşturulurken, aynı zamanda müşteri normlarına uygun şekilde isimlendirilmiştir.



Şekil 37: Tasarım Ürün Ağaç Yapısı (Çizen)

Kalıp tasarımına başlarken metot dosyasında bulunan eğri ve yüzey datalarının tasarıma girdi olması gerekmektedir. Bu dataları ÇİZEN içerisine aktarım (import) yapabilmek mümkündür. İçeri aktarılan eğri ve yüzeyler için Çizen yeni bir parça dosyası açmakta ve step dosyasından alınan bilgileri bu dosyada kullanıcıya sunmaktadır. Daha önce açılan bir parça dosyasına eğri ve yüzeyleri içeri aktarmak mümkün değildir. Bu yüzden öncelikle eğri ve yüzeyler içeri aktarılmalı, sonra geometrik çizimler yapılmalıdır. Birden fazla eğri ve yüzey aynı anda içeri aktarılmalıdır. İçeri aktarılan yüzeyler bir "GeometrySet" içerisinde toplanmaktadır. "GeometrySet" içerisine toplanan eğri ve yüzeyler aşağıda ki resimde görülmektedir.



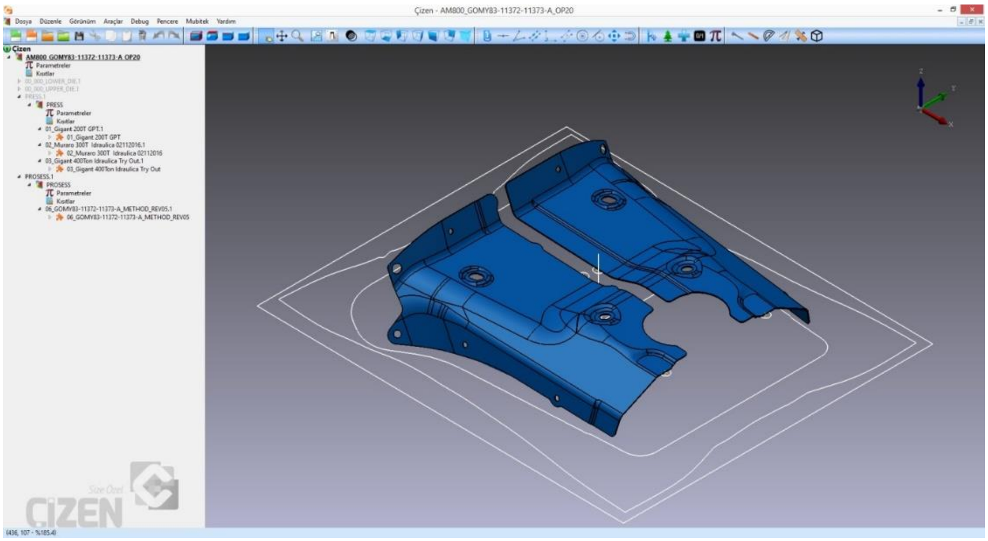
Şekil 38: GeometrySet İçerisine Toplanan Eğri ve Yüzeyler

Önce geometrik çizimleri yapıp sonra gerektikçe yüzeyleri içeri aktarmayı düşünmek çalışmalarınızı kaybetmenize neden olabilmektedir. Tasarıma başlarken hangi yüzeylerin gerekli olacağı düşünülmeli ve ona göre bir yol izlenmelidir.

Bunun yanı sıra tasarımın belli bir aşamasına geldiğinizde mevcut eğri ve yüzey girdilerinin güncellenmesi de söz konusu olmadığı için tasarımda fazladan işler yapmanıza sebep olabilmektedir.

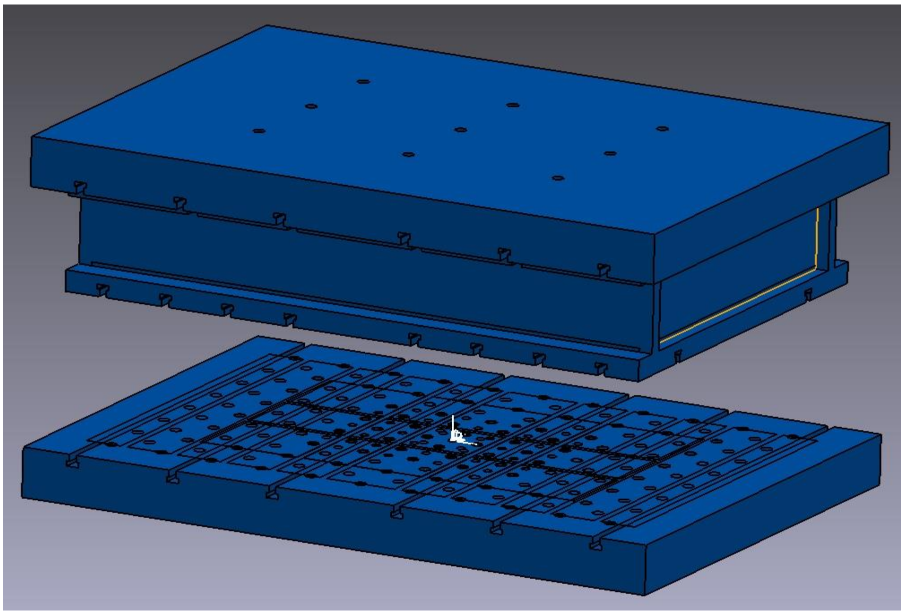
Tasarım öncesinde hangi eğri ve yüzeylerin gerekli olacağı düşünülmüş ve metot dosyasından gerekli step dönüşümleri gerçekleştirilmiştir. Metot dosyasına ait stepler ÇİZEN'de içeri aktarılarak ana döküm parça dosyaları oluşturulmuştur. Oluşturulan bu döküm parça dosyaları montaj ortamında "Parça Aç" ikonu kullanılarak tasarım ürün ağacında bulunması gereken yerlere yerleştirilmiştir.

Ardından, prosese ait step data içeri aktarılarak proses dosyası oluşturulmuş ve aşağıdaki resimde görüldüğü gibi tasarım ürün ağacına yerleştirilmiştir. Proses dosyası kalıplanacak parçaya ait eğri, yüzey ve 3D parça datalarını içerisinde barındırmaktadır.



Şekil 39: Prosesin Tasarım Ürün Ağacına Yerleştirilmesi

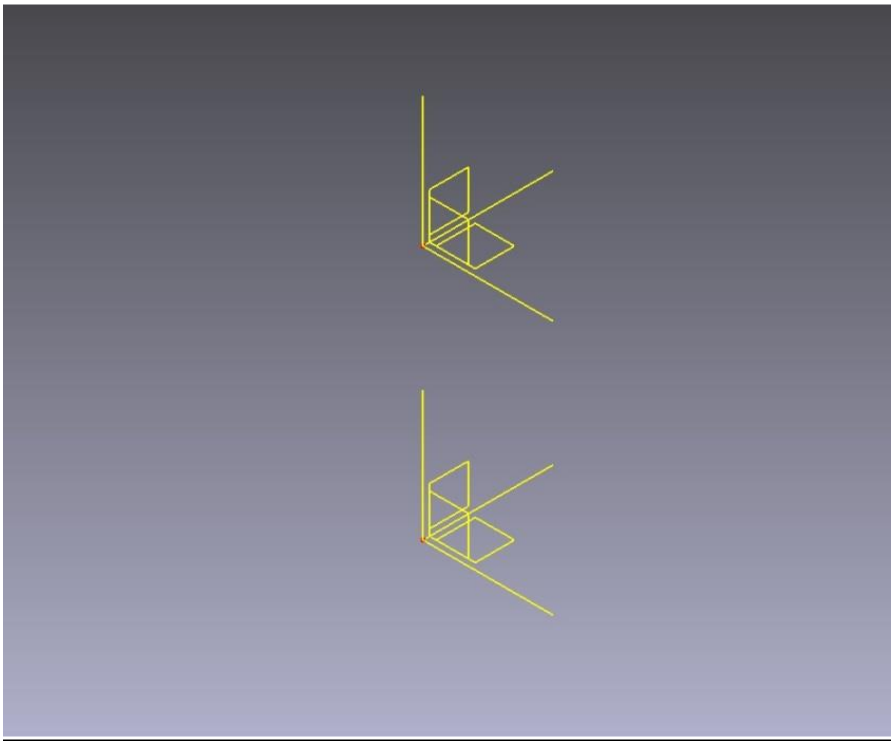
Pres step dataları ÇİZEN'de "İçe Dosya Aktar" ile içeri aktarılarak pres dataları oluşturulmuş ve aşağıdaki resimde görüldüğü gibi tasarım ürün ağacına yerleştirilmiştir.



Şekil 40: Pres Datalarının Tasarım Ürün Ağacına Yerleştirilmesi

CATIA ile kalıp tasarımı yapılırken, kalıp içerisinde birden fazla yerde kullanılacak bileşenleri bir referans parçadan yayınlamak tasarım sürelerini büyük oranda kısaltmayı mümkün kılmaktadır. ÇİZEN'de bu özellik olmadığı için tasarım ürün ağacına ilave bir referans dosya açılmasına gerek yoktur.

ÇİZEN'de yayınlama (publication) ve linkli çalışma mantığı bulunmadığı için döküm parçaların her birinin içerisinde parça eksenleri tanımlanmıştır. Aşağıdaki görselde örneği mevcuttur.



Şekil 41: Döküm Parçaların İçerisine Tanımlanan Parça Eksenleri

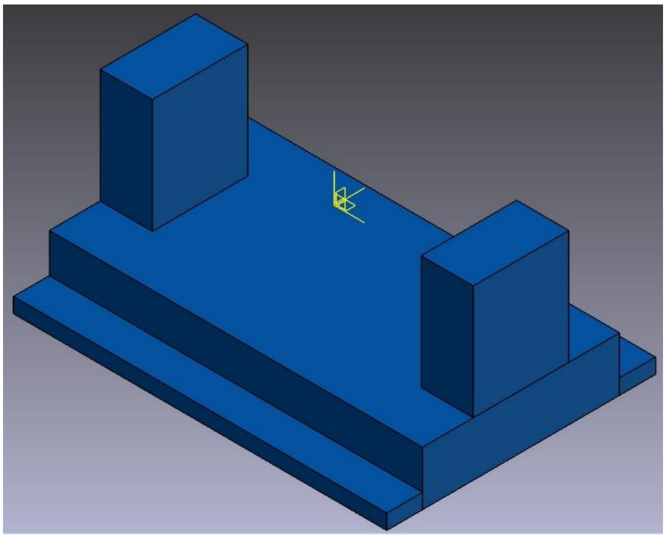
3.1.2 Döküm Gruplarının Tasarlanması

3.1.2.1 Alt Dökümün Tasarlanması

Alt döküm parçasının tasarım işlemleri gruplandırılarak açıklanmaya çalışılmıştır.

3.1.2.1.1 Döküm Tablasının Oluşturulması

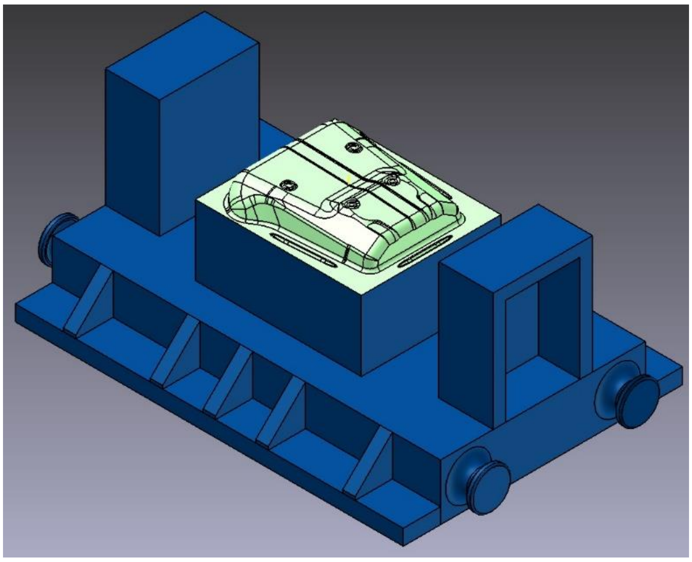
Tasarım öncesinde hangi eğri ve yüzeylerin gerekli olacağı düşünülmüş ve metot dosyasından gerekli step dönüşümleri gerçekleştirilmişti. Metot dosyasına ait stepler ÇİZEN'de içeri aktarılarak alt döküm parça dosyası oluşturulmuştu. Alt döküm parçası ayrı bir pencerede açılmıştır. Alt döküm ekseni oluşturulmuştur. Bu eksen üstüne kalıp tablası çizilmiş, olması gereken ölçülerde kule ve kızak yapıları tasarlanmıştır.



Şekil 42: Alt Döküm Tablası ve Kule Oluşumu

3.1.2.1.2 Erkek Yapısının Oluşturulması

Kalıbın erkeği olarak bildiğimiz bölünecek yapının oluşturulması işlemine geçilmiştir. Catia'da ki 'Project' komutunun eşdeğeri "İzdüşüm" operasyonu ÇİZEN'de düzgün çalışmadığı için kalıbın konturu dikdörtgen olarak bir skeç içinde çizilmiştir. Ardından çizilen tüm parçalar birleştirilerek aşağıda resimde görülen çizim oluşturulmuştur.

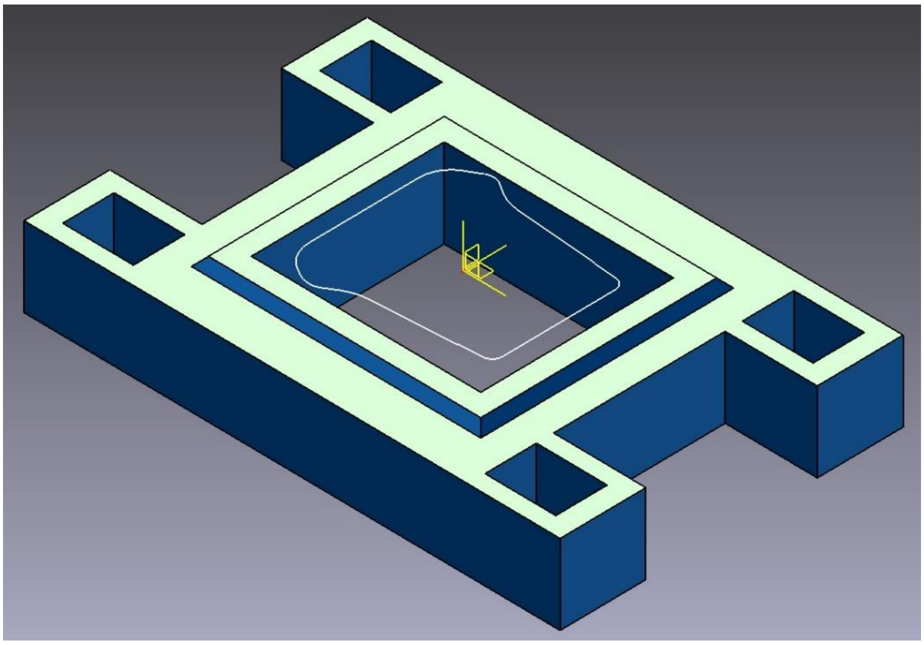


Şekil 43: Erkek Grubunun Tasarlanması

3.1.2.2 Pot Çemberinin Tasarlanması

Tasarım öncesinde hangi eğri ve yüzeylerin gerekli olacağı düşünülmüş ve metot dosyasından gerekli step dönüşümleri gerçekleştirilmişti. Metot dosyasına ait stepler ÇİZEN'de içeri aktarılarak pot çemberi döküm parça dosyası oluşturulmuştu. Pot çemberi döküm parçası ayrı bir pencerede açılmıştır.

Pot çemberi ekseni oluşturulmuştur. Bu eksen üzerine skece düşülerek pot çemberi çizilmiştir. Pot çemberi boşaltısı için metot dosyasında ki eğriler kullanılmaktadır fakat Catia'da ki Project komutunun eşdeğeri "İzdüşüm" operasyonu ÇİZEN'de düzgün çalışmadığı ve stepten alınan eğri bilgileri ÇİZEN'de kullanılamadığı için kalıbın konturu dikdörtgen olarak bir skeç içinde çizilmiştir. Pot çemberi genel hatlarıyla aşağıda resimde görülmektedir.

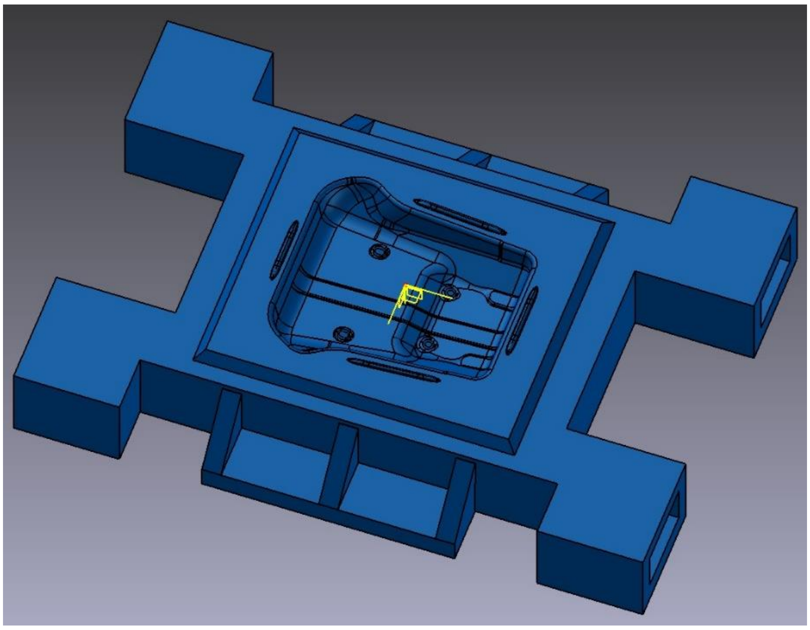


Şekil 44: Genel Hatlarıyla Pot Çemberi

3.1.2.3 Üst Dökümün Tasarlanması

Tasarım öncesinde hangi eğri ve yüzeylerin gerekli olacağı düşünülmüş ve metot dosyasından gerekli step dönüşümleri gerçekleştirilmişti. Metot dosyasına ait stepler ÇİZEN'de içeri aktarılarak üst döküm parça dosyası oluşturulmuştu. Üst döküm parçası ayrı bir pencerede açılmıştır.

Üst döküm ekseni oluşturulmuştur. Bu eksen üzerine skece düşülerek üst döküm çizilmiştir. Üst döküm genel hatlarıyla aşağıda ki resimde görülmektedir.



Şekil 45: Genel Hatlarıyla Üst Döküm

3.1.3 Döküm Gruplarında Eksik Kalan Kısımların Tamamlanması

Tasarıma devam edilirken ÇİZEN'in daha önce ki projelerinde karşılaştığımız ve çözmeye çalıştığımız index (geometrik isimlendirme) problemiyle tekrar karşılaşılmıştır. Daha önce ki projelerde karşılaşılan index (geometrik isimlendirme) problemi tüm Case'leri ile çözülmediği için tasarımda ilerlememizi engellemiştir.

Döküm grupları genel hatlarıyla oluşturulduktan sonra tasarımda eksik kalan kısımlar tamamlanmaya çalışılmıştır.

3.1.4 Standart Parçaların Tasarıma Eklenmesi

Döküm grupları genel olarak oluştuktan sonra standart parçalar "Parça Aç" komutu ile program içinde açılmış ve tasarım içerisine alınmıştır.

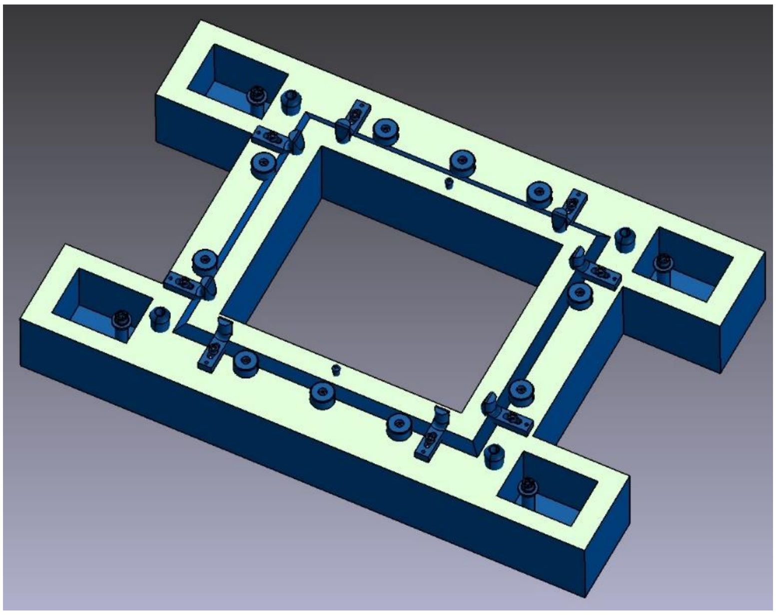
Tasarımda olması gereken; her bir standart parçanın ilgili döküm montajlarının içerisinde bulunmasıdır. Örneğin Pot çemberinde yer alan standart parça, ürün ağacında Pot çemberi montajının içinde yer almalıdır. Bu tasarım çalışmasında bütün standart parçalar kalıbın montaj dosyasının içerisinde bulunmaktadır.

Tasarım içerisine alınan parçalar aşağıdaki gibidir;

- Sac dayama
- Stoper
- Merkezleme Pimi

- Askı Cıvatası
- Lazer Referans Zımbası

ÇİZEN montaj ortamında standart parçalar ilişkilendirme operasyonları ile gerekli konumlara taşınabilmektedir. Taşınma koordinatlarını bilmemiz gerektiği için 3B pozisyonlama ve 3B Oryantasyon kullanılarak parçalar kalıp içerisinde ki konumlarına taşınmıştır.



Şekil 46: Standart Parçaların Kalıp Tasarımına Eklenmesi

3.2 Tasarımda Kullanılan Komutların Saptanması Ve Kullanım Sıklığı Fazla Olan Komutların Belirlenmesi

Belirlenen bir döküm kalıbın ÇİZEN programında tasarımının yapılması çalışmasında kaydedilen videolar incelenmiştir. Tasarım esnasında kullanılan bütün komutlar saptanmıştır. Bu komut listesi aşağıdaki gibidir:

- Bileşen Ekle (Parça, Montaj) - Montaj
- Montajı Yeniden Adlandır – Sağ Tık – Sol Çift Tık
- Parçayı Yeniden Adlandır – Sağ Tık – Sol Çift Tık
- İçe Dosya Aktar (Import)
- Kaydet – File
- Aç
- Parça Aç
- Skeç
- 2B Çizgi çizme

- 2B Dikdörtgen çizme
- 2B Daire çizme
- Uzaklık Ölçülendirme
- Çakışıklık
- Paralellik
- Diklik
- Yarıçap Ölçüsü
- Eş Merkezlilik
- Tanjantlık
- Simetriklik
- 2B Aynalama
- 23B Radyus
- 2B Pah
- Uzat İle Katı Oluşturma
- Döndürme İle Katı Oluşturma
- Yeni Eksen Takımı
- 3B Radyus
- 3B Pah
- Renk Atama
- 3B AynalTasarım Ürün Ağaç Yapısı (Çizen)
- 3B Simetriklik
- Çoğaltma
- Çakışık (İlişki)
- Basit Delik
- Kademeli Delik
- Birleştirme Operasyonu
- Boşaltma Operasyonu
- Gizle/Göster
- Yüzey Ayırma
- Yüzey Birleştirme
- Zoom/Pan fonksiyonları
- Ekrana Sığdır

- Tüm Eksenleri Gizle
- Katı Seçim Filtresi
- Yüzey Seçim Filtresi
- Kenar Seçim Filtresi
- Uzaklık Ölçüsü
- Uzunluk/Yarıçap Ölçüsü
- Montajda Koordinat ile 3B Pozisyonlama
- Montajda Koordinat ile 3B Oryantasyon

Videolar incelenip değerlendirildiğinde sıklıkla kullanılan 5 komutun aşağıdakiler olduğu görülmektedir;

- Uzat İle Katı Oluşturma
- Uzaklık Ölçüsü
- Yüzey Ayırma
- Dikdörtgen Geometrisi Oluşturma Komutu
- İçe Dosya Aktar (Import)

3.3 Tasarım Sürecinde Kullanılan Modüller

Tasarım sürecinde ÇİZEN programı içindeki 2B Skeç, 3B Katı, 3B Yüzey, 3B Montaj modülleri ve hesaplama araçları kullanılmıştır.

Şekil 47: ÇİZEN'de "Araçlar Çubuğu"

- 2B Skeç; tasarlamak istenilen elemanın 2 boyutlu geometrisinin çizilmesinde,
- 3B Katı; 2B tasarlanan geometrilerin/ çizimlerin 3B hale getirilmesinde,
- 3B Yüzey; kalıbın erkeği ve dişisini oluştururken yüzey ayırma ve birleştirme işlemlerinde,
- 3B Montaj; ürün ağaç yapısını oluştururken bileşen eklenmesinde, bileşenin taşınmasında, döndürülmesinde, parçaların çoğaltılmasında, standart elemenların konumlanması için ilişki tanımlanmasında, referans geometrilerin oluşturulmasında,

- Hesapla araçları; parça üstünden ölçü alınmasında kullanılmıştır.

3.4 Tasarım Sürecinde Zorlanılan Noktaların Saptanması Ve Öneriler

ÇİZEN programı ile yapılan tasarım çalışmasında bazı noktalarda zorlanılmış ya da tasarım esnasında bir takım öneriler getirilmiştir. Tasarım videolarının incelenmesi sonucunda ortaya konan saptamalar aşağıda açıklanmaya çalışılmıştır. Bu saptamalar, kullanılan ÇİZEN programının değerlendirilmesi, tasarımcı gözüyle eleştiriler, komutların kullanım kolaylığının değerlendirilmesi, kalıp tasarımında gerekli özelliklerin saptanması vb. durumların değerlendirilmesiyle ortaya konulmuştur.

1. Kalıp tasarımına başlarken metot dosyasında bulunan eğri ve yüzey datalarının tasarıma girdi olması gerekmektedir. Bu dataları ÇİZEN içerisine aktarım (import) yapabilmek mümkündür. İçeri aktarılan eğri ve yüzeyler için Çizen yeni bir parça dosyası açmakta ve stepten alınan bilgileri bu dosyada kullanıcıya sunmaktadır. Daha önce açılan bir parça dosyası içerisine eğri ve yüzeyleri içeri aktarmak mümkün değildir. Bu yüzden öncelikle eğri ve yüzeyler içeri aktarılmalı, sonra geometrik çizimler yapılmalıdır. Birden fazla eğri ve yüzey aynı anda içeri aktarılmalıdır. İçeri aktarılan yüzeyler bir "GeometrySet" içerisinde toplanmaktadır. Önce geometrik çizimleri yapıp sonra gerektikçe yüzeyleri içeri aktarmayı düşünmek çalışmalarınızı kaybetmenize neden olmaktadır. Tasarıma başlarken hangi yüzeylerin gerekli olacağı düşünülmeli ve ona göre bir yol izlenmelidir. Bunun yanı sıra tasarımın belli bir aşamasına geldiğinizde mevcut eğri ve yüzey girdilerinin güncellenmesi de söz konusu olmadığı için tasarımda fazladan işler yapmanıza sebep olabilmektedir. Tasarımın belli bir aşamasında veya revizyon esnasında bazı girdileri çizim dosyasına ilave edememek kalıp tasarımı için büyük bir sorundur.
2. ÇİZEN'de eğriler ve yüzeyler "İçe Dosya Aktar" ile alınabilmektedir fakat Catia'da ki Project komutunun eşdeğeri "İzdüşüm" operasyonu ÇİZEN'de düzgün çalışmadığı için eğriler 2B Skeç'te kullanılamamaktadır.
3. ÇİZEN'de tasarımla ilgili referans çizimler, noktalar, eksenler, eğriler, yüzeyler vb. elemanlar part içerisinde ki "GeometrySet" içerisinde bulunmaktadır. Çizim karmaşıklaştıkça yeni set ekleme ve bu bileşen içerisinde gruplama ihtiyacı doğmaktadır. (Catia programında "Geometrical Set" adı verilen bir bileşen tipi bulunmaktadır. Yeni bileşen oluşturulabilmekte ve bu bileşen içerisinde gruplamalar yapılabilmektedir.)

4. Kalıpta birden fazla yerde kullanılacak bileşenlerin bir referans dosya içerisinden tüm kalıba yayınlanması (linkli çalışma-publication) durumu kalıp tasarımını kolaylaştırmaktadır. ÇİZEN programında linkli çalışma (publication) bulunmamaktadır. Bu sebeple kalıp tasarım süreci yavaş ilerlemektedir ve kalıp revizyona uğradığında değişikliklerin uygulanması uğraştırıcı olacaktır.
5. Tasarıma devam edilirken ÇİZEN'in daha önce ki projelerinde karşılaştığımız ve çözmeye çalıştığımız index (geometrik isimlendirme) problemiyle tekrar karşılaşılmıştır. Daha önce ki projelerde karşılaşılan index (geometrik isimlendirme) problemi tüm case'leriyle çözülmediği için tasarımda ilerlememizi engellemiştir.

6. ÇİZEN montaj ortamında standart parçalar ilişkilendirme operasyonları ile gerekli konumlara taşınabilmektedir. 3B Montaj İlişkilendirme operasyonlarında seçimler ve seçimlerin görünürlüğü anlamında bazı geliştirmeler yapılmalıdır. Mevcut ÇİZEN'de ölçülü taşıma gerekli olduğunda koordinatlarını bildiğimiz konumlara 3B pozisyonlama ve 3B Oryantasyon kullanılarak parçalanr kalıp içerisinde ki konumlarına taşınabilmektedir. CATIA'da ki gibi standart parçaların eksen veya noktalara bağlanabilmesi konusu değerlendirilebilir.

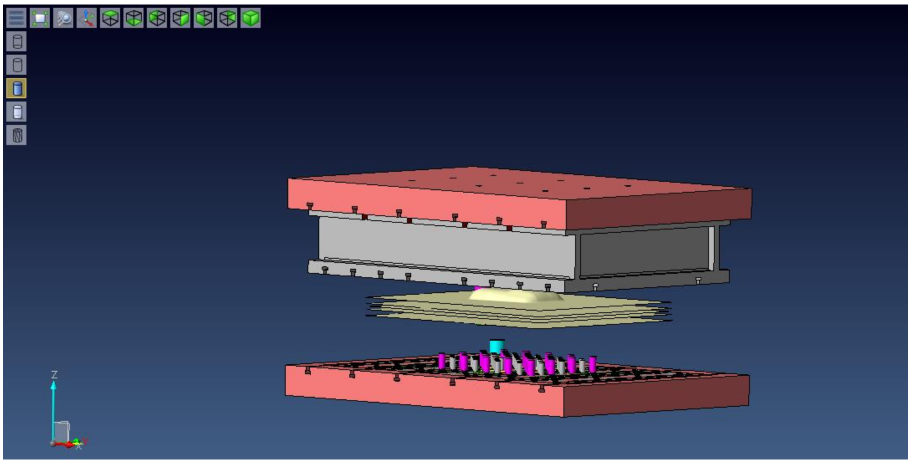
4. VISICAD

4.1 Tasarım Sürecinde İzlenen Yolun Ve Tasarım Aşamalarının İncelenmesi

Tasarım süreci; preslerin ve metot yüzeylerinin konumlandırılmasından başlayıp tüm tasarım aşamaları bölünerek gerçekleştirilmeye ve anlatılmaya çalışılmıştır. Raporun bu bölümünde tasarım aşamaları hakkında detaylı bilgi verilmiştir.

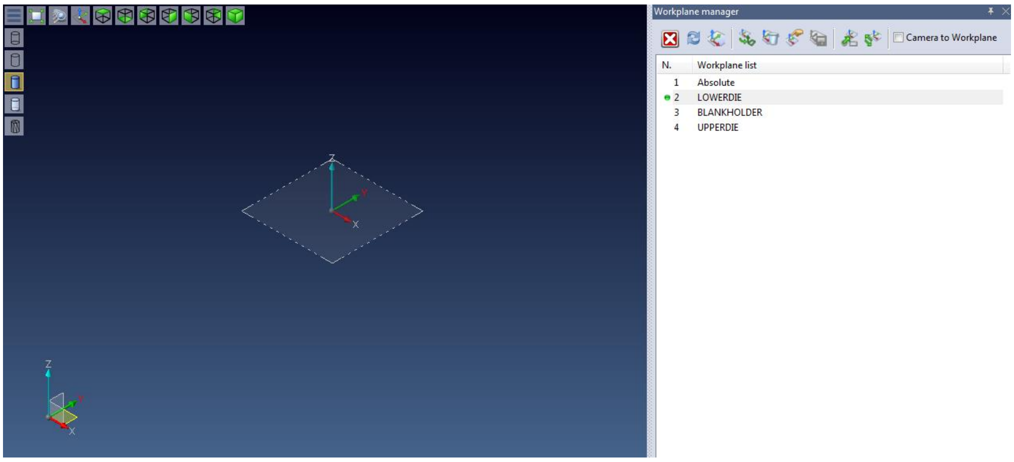
4.1.1 Preslerin Ve Metot Yüzeylerin Konumlandırılması

Tasarıma preslerin ve metot yüzeylerin Visicad programına kopyalanması ile başlanmıştır. Başlangıçta aşağıdaki resimde görüldüğü gibi presler ve metot yüzeyleri konumlandırılmıştır.



Şekil 48: Preslerin ve Metot Yüzeylerin Konumlandırılması

Visicad programında oluşturulan eksenlerin linkli çalışma yapılarak, döküm parçalar içerisine eklenmesi mümkün olmadığı için kalıp eksenlerini oluşturmak için her bir döküm grubu içinde alt döküm düzlemi referans alınarak düzlem oluşturulmuştur. Alt kalıp grubu içerisindeki alt döküm parçasına, pot çemberi grubunun döküm parçasına ve üst grup döküm parçasına bir adet taban düzlemi atanmış ve kalıp bu düzlemler üstünde çizilmiştir. Aşağıdaki görselde örneği mevcuttur;



Şekil 49: Alt Döküm Parçası İçine Atanmış Düzlem

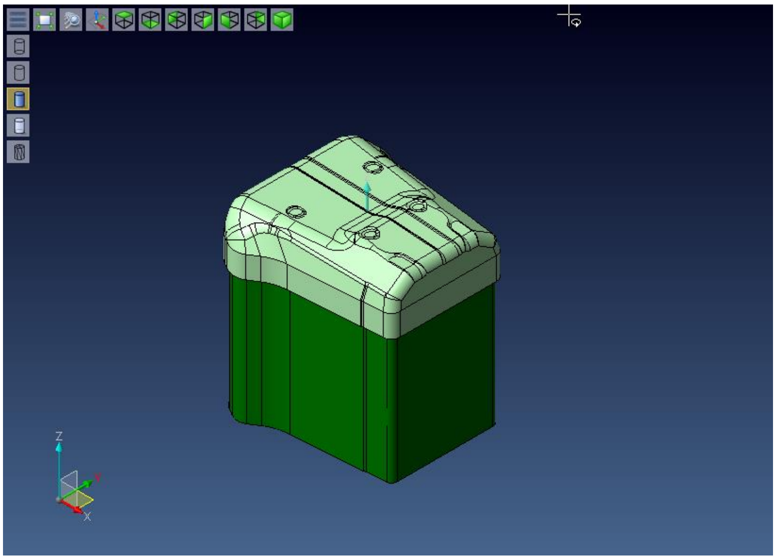
4.1.2 Döküm Gruplarının Tasarlanması

4.1.2.1 Alt Dökümün Tasarlanması

Alt döküm parçasının tasarım işlemleri, gruplandırılarak açıklanmaya çalışılmıştır.

4.1.2.1.1 Erkek Yapısının Oluşturulması

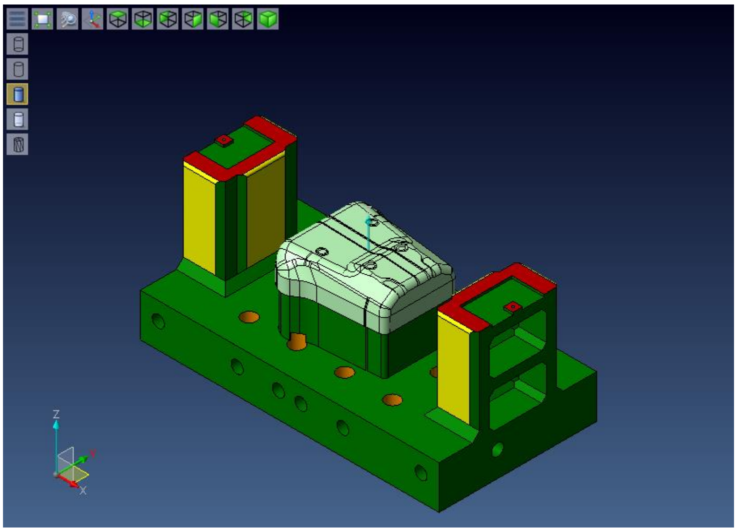
Kalıba erkek yapının oluşturulması ile başlanmıştır. Parça konturunun ‘Edit → Project on Workplane’ komutu ile çalışma eksenine izdüşümü alınmıştır. Oluşturulan izdüşüm; profil, parçayı geçecek şekilde ‘Extrude Elements’ komutu ile katı hale getirilmiştir. Oluşturulan katıdan metot yüzeyi ‘Operation → Cavity’ komutu ile ayrılmış, fazlalıklar silinmiştir. Katı tekrar öteleme yüzeyiyle ayrılmıştır. Profil 10mm ve 40mm içeriye doğru ötelendi. Katı bu ötelenmiş profillerden ‘Cut Bodies’ komutu ile kesildi. Gereksiz yerler silinip ‘Unite’ komutuyla katılar birleştirildikten sonra ‘Change Attribute → Create New Layer’ ile layer ismi verilmiştir. Erkek oluşturulmuş oldu.



Şekil 50: Erkek Yapısının Tasarlanması

4.1.2.1.2 Alt Tablanın Oluşturulması

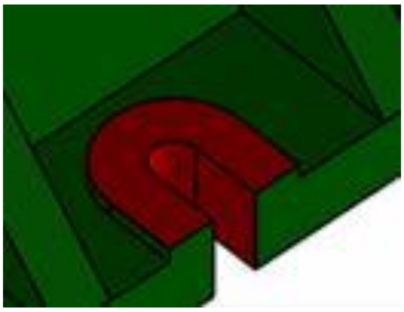
Alt döküm ‘Cuboid’ komutuyla merkez noktasından çizilmeye başlanmıştır. Alt tabla , kuleler ve kızak yapıları olması gereken ölçülere göre tasarlanmıştır. Tij millerinin boşaltıları yapılmıştır. Alt dökümde ki boşaltılar yapılmıştır.



Şekil 51: Alt Döküm Tablası ve Kule Oluşumu

4.1.2.1.3 Pres Bağlantılarının Tasarlanması

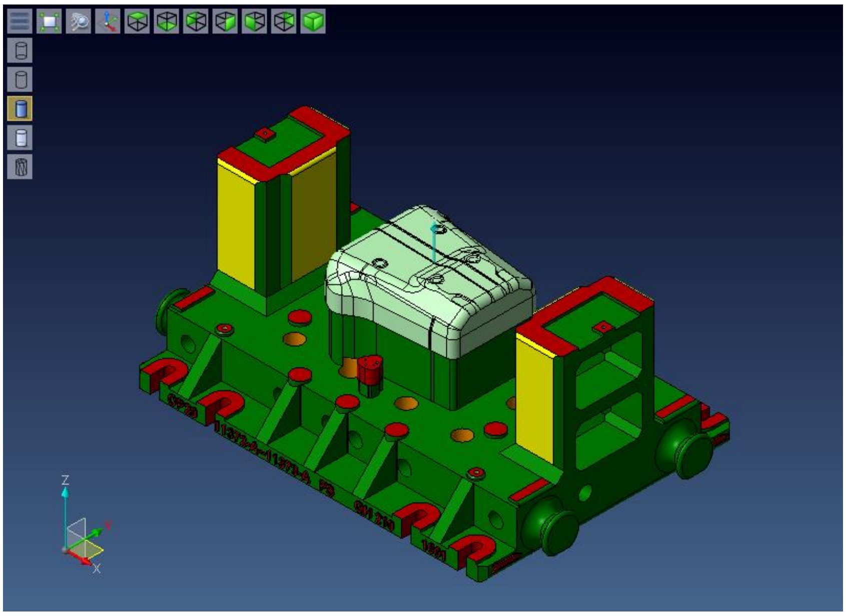
Presler ilk aşamada eklendiği için pres bağlantı yerleri ve tij millerinin yerleri bellidir. Pres bağlantı yerleri için iki adet slot profil çizilmiş ve bağlantı olacak yerlere ötelenmiştir. Bir taraftaki feder ve bağlantı kanalları oluşturulduktan sonra merkeze göre 'Mirror' komutuyla karşı tarafa da kopyalanmıştır.



Şekil 52: Pres Bağlantısı

4.1.2.1.4 Mapa eklenmesi

Mapa tasarlanması için bir adet yardımcı eksen atılmıştır. Mapa tasarımı yapıldıktan sonra merkeze göre dört köşeye aynalanarak mapalar oluşturulmuştur.

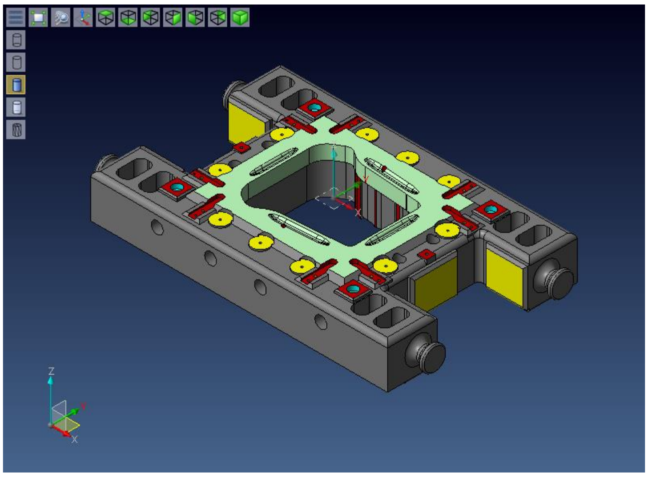


Şekil 53: Alt Döküm

4.1.2.2 Pot Çemberinin Tasarlanması

- Pot çemberi düzlemi olarak alt döküm parçası içinde oluşturulan düzlem referans alınarak bir taban düzlemi atanmıştır.

- Atılan taban düzleminin merkezine göre pot çemberinin katısı oluşturulmuştur. Metotla alınan öteleme yüzeylerini kullanarak pot çemberinin üst yüzeyi oluşturulmuştur.
- Alt dökümün kızakları referans alınarak pot çemberinin kızak boşaltıları oluşturulmuştur.
- Parçanın açınımı 30 mm dışarı yöne ötelenerek oluşturulan profil ekstrüzyon ile katılaştırılmış ve yüzey ile bölme işlemi uygulanmıştır.
- Taşıma mapasının olduğu yüzeye yardımcı eksen atılıp mapa çizildikten sonra aynalama yapılarak her köşeye dağıtılmıştır.
- Döküm tablanın alt boşaltması ve balkon için profiller çizilip katı hale getirilip ötelenmiş metotla kesilmiştir. Daha sonra o katılar pot çemberi dökümünden çıkarılmıştır. Pot çemberinin balkon ve alt boşaltmaları tamamlanmıştır.
- Pot çemberinin üst kısmındaki boşaltılar , L dayamaların ve stoperlerin oturduğu yerler ve merkezleme pimlerinin yerleri yapılmıştır.
- Referans elemanları ve sürtünme plakalarının da tasarımı yapıldıktan sonra pot çemberi oluşturulmuştur.



Şekil 54: Pot Çemberi

4.1.2.3 Üst Dökümün Tasarlanması

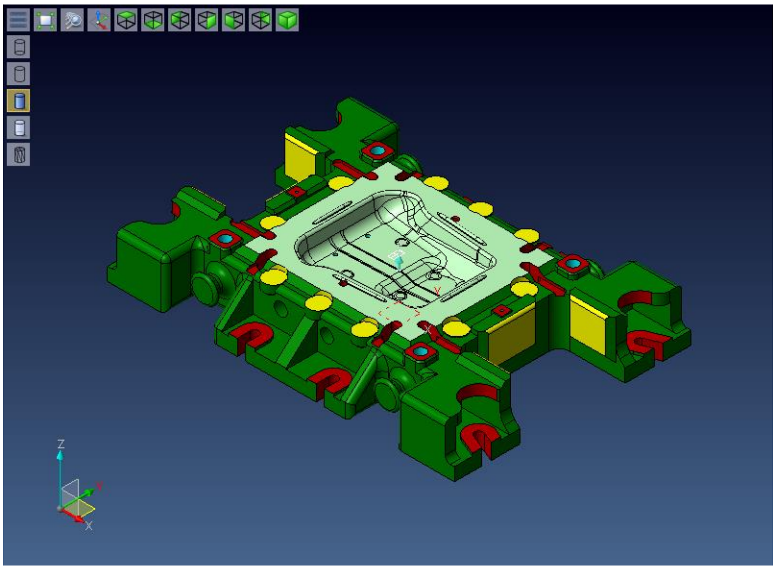
Pot çemberinin oluşturulmasında olduğu gibi alt düzlem referans alınarak üst döküm düzlemi oluşturulmuş ve tasarıma başlanmıştır. Alt döküm tablasının ebatlarında üst döküm tablası oluşturulmuştur. Yüzey boşaltmaları, tablanın alt boşaltıları, kızak boşaltıları, sürtünme plakaları, referanslama elemanları, mapalar için yuvalar, döküm federleri, pres bağlantılarının bulunacağı kısım tasarlanmıştır.

4.1.2.3.1 Pres Bağlantılarının Tasarlanması

Üst dökümde 2 farklı pres bağlantısı bulunmaktadır. Bağlantılardan biri açılı diğeri düzdür. Her iki pres bağlantısı içinde alt döküm bağlantılarında uygulanan yöntemde olduğu gibi çoğaltılarak tasarlanmıştır.

4.1.2.3.2 Mapa Eklenmesi

Mapaların bulunduğu yüzeye yardımcı bir eksen atılmıştır. Mapa tasarlandıktan sonra aynalama komutu ile çoğaltılmıştır.



Şekil 55: Üst Döküm

4.1.3 Standart Parçaların Tasarıma Eklenmesi

Döküm grupları oluştuktan sonra standart parçalar farklı bir Visi programı içinde açılmış ve tasarım içerisine kopyalanmıştır.

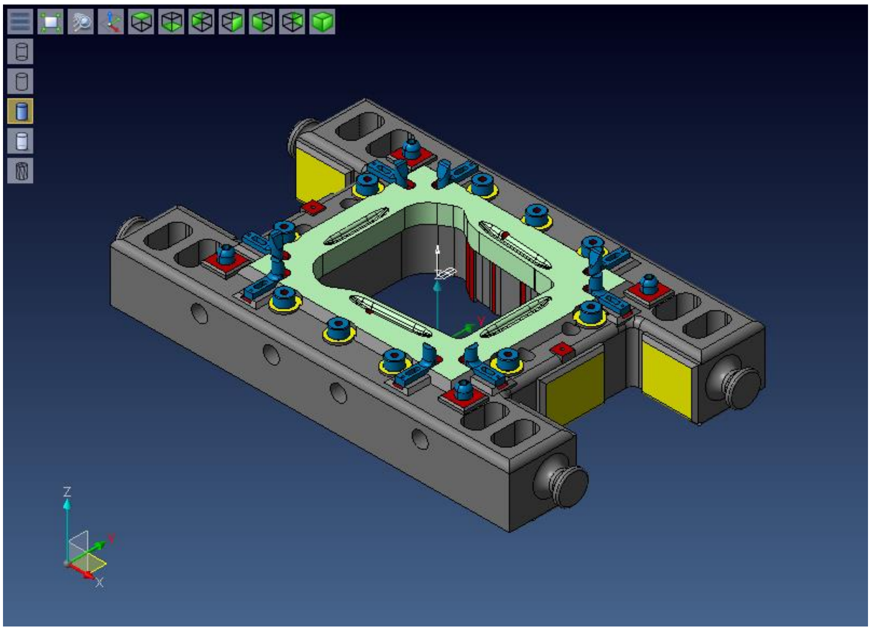
Tasarımda olması gereken; her bir standart parçanın ilgili gruplar içerisinde bulunmasıdır. Örneğin pot çemberinde yer alan standart parça pot çemberi grubunun içinde yer almalıdır.

Tasarımda her standardın oturacağı yüzey dökümler çizilirken yapıldığı için standartlar da o yüzeylere göre konumlandırılmıştır.

Tasarım içine alınan parçalar aşağıdaki gibidir;

- L dayama
- Stoper
- Merkezleme Pimi
- Askı Cıvatası
- Zımba
- Darbe plakası

- İz zımbası
- Zımba Tutucu

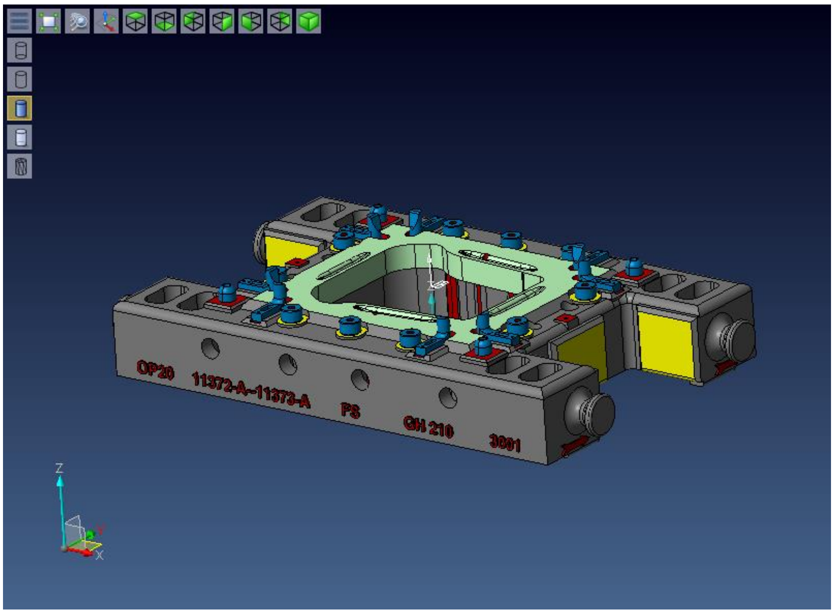


Şekil 56: Standart Parçaların Döküme Eklenmesi

4.1.4 Kalıp Yazılarının Ve Oklarının Eklenmesi

Tasarım içerisine yazıların ve okların eklenmesi işleminde izlenen yol aşağıdaki gibidir;

- ‘Sketch lines’ komutu ile ok işareti çizildi. ‘Close Profil’ ile sketch profil haline getirilip katılaştırıldı. Oklar gerekli yerlere koyulup dökümler ile birleştirildi.
- Kalıp için gerekli yazılar ise ‘Text’ komutu ile yazılmış, katı hale getirildikten sonra dökümlere birleştirilmiştir.



Şekil 57: Kalıp Yazılarının ve Okların Tasarıma Eklenmesi

4.2 Tasarımda Kullanılan Komutların Saptanması Ve Kullanım Sıklığı Fazla Olan Komutların Belirlenmesi

Belirlenen bir döküm kalıbın Visicad programında tasarımının yapılması çalışmasında kaydedilen videolar incelenmiştir. Tasarım esnasında kullanılan bütün komutlar saptanmıştır. Bu komut listesi aşağıdaki gibidir;

- **Project on workplane :** çalışma eksenine izdüşüm komutu
- **Extrude elements :** katı model yapma
- **Profile offset :** profili ofsetleme
- **Cut bodies :** kesme komutu
- **Cavity :** ayırma
- **Unite :** birleştirme
- **Cuboid :** profilsiz katı oluşturma
- **Delete :** silme
- **Parallel :** paralel çizgi atma
- **Translation :** taşıma
- **Chamfer :** pah kırma
- **Mirror :** aynalama
- **Move a face :** yüzey öteleme
- **Change attribute :** katıları layer e kaydetme ve renklendirme komutu

- **Blends :** Radius
- **Centre Radius :** daire çizme
- **Cylinder :** profilsiz katı silindir oluşturma
- **Feature manager :** delik delme / diş açma komutu
- **Progress insert elements :** cıvata pim atma komutu
- **Create workplane :** plane oluşturma
- **Close profile :** kapalı sketch i profil yapma
- **Subtract :** bir katıdan bir katıyı silerek çıkarma
- **Rectangle profile :** dikdörtgen profil oluşturma
- **Edit cylindirical face :** çap yüzeyleri büyütüp küçültme
- **Slot profile :** slot profil çizme
- **Replace face with surface :** ilk seçilen yüzeyi ikinci seçilen yüzeye indirme
- **Create profile :** katının yüzeyinde ya da çizgilerinde profil oluşturma
- **Sketch lines :** çizgi
- **Text :** yazı

Videolar incelenip değerlendirildiğinde sıklıkla kullanılan 5 komutun aşağıdakiler olduğu görülmektedir:

1. **Cavity :** ayırma
2. **Unite :** birleştirme
3. **Cuboid :** profilsiz katı oluşturma
4. **Mirror :** aynalama
5. **Delete :** slime

4.3 Tasarım Sürecinde Kullanılan Modüller

Tasarım sürecinde Visicad programı içindeki Standard, Edit, Wireframe, Modeling, Dimensions modülleri kullanılmıştır.

Şekil 58: Visicad modülleri

- Standard modülü; parçaların gizlenmesinde ya da tek bırakılmasında, katı ya da şeffaf görüntülenmesinde, layer isimlerinin verilmesinde,
- Edit modülü; bileşenin taşınmasında, döndürülmesinde, parçaların çoğaltılmasında, silinmesinde, trimlenmesinde,

- Wirefrime modülü; referans geometrilerin oluşturulmasında (çizgi, nokta, paralel çizgi atma, daire vb.), sketchlerden profil oluşturma, profil ofsetleme,
- Modeling modülü; katı çizimlerinde, katıları birleştirmede, pah ve radyus kırmada, split etme işleminde,
- Dimensions modülü; ölçülendirme ve yazı yazmada kullanılmıştır.

4.4 Tasarım Sürecinde Zorlanılan Noktaların Saptanması Ve Öneriler

Visicad programı ile yapılan tasarım çalışmasında bazı noktalarda zorlanılmış ya da tasarım esnasında bir takım öneriler getirilmiştir. Tasarım videolarının incelenmesi sonucunda ortaya konan saptamalar aşağıda açıklanmaya çalışılmıştır. Bu saptamalar, kullanılan Visicad programının değerlendirilmesi, tasarımcı gözüyle eleştiriler, komutların kullanım kolaylığının değerlendirilmesi, kalıp tasarımında gerekli özelliklerin saptanması vb. durumların değerlendirilmesiyle ortaya konulmuştur.

1. Visicad programında kalıp tasarımı için gerekli olan öteleme yüzeylerinin oluşturulması için kullanılan 'Yüzey Öteleme' komutu yetersiz kalmıştır. Bazı yüzeyleri öteleyememiş bazılarını ötelemesine rağmen çok uzun sürmüştür. Tasarım sırasında öteleme yüzeyleri başka bir programda oluşturulup kalıp içine alınmıştır. Kalıp tasarımını gerçekleştirme amacında odaklanmış bir programın yüzey öteleme komutunun yeterli düzeyde çalışır olması gerekmektedir.

2. Visicad programında layer sistemi ile çalışılmaktadır. Bu sistem linkli tasarım mantığı ile benzerlik göstermese de yapılacak olan program herkese hitap etmelidir. Her iki tasarım mantığının da artıları, eksileri ortaya konulmalıdır.

4.5 Tasarım Çalışmasına Revizyon İşlemlerinin Uygulanması

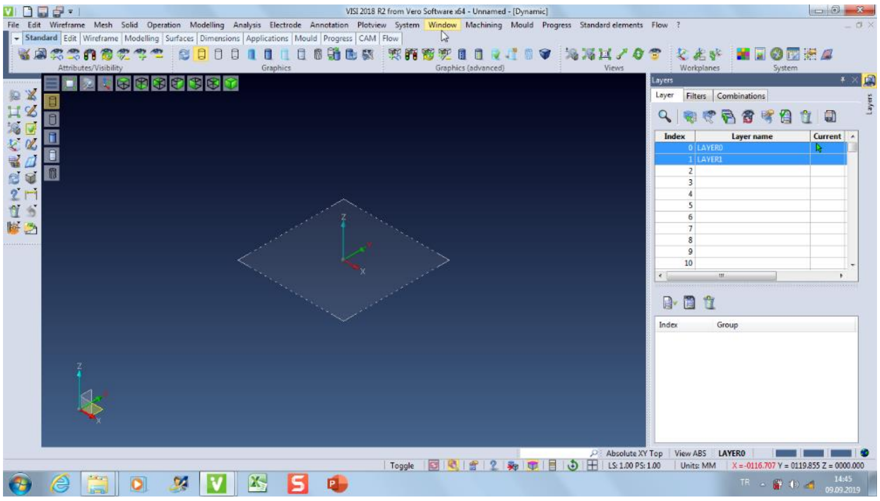
Raporun bu bölümünde kalıp tasarımına revizyon uygulanmak istenmesi ya da değişiklik yapılması ihtiyacı hissedildiği durumlarda tasarımcı tarafından değişikliklerin uygulanma kolaylığı değerlendirilmiştir.

Layer sistemi ile çalışan programlarda bir değişiklik geldiğinde her şey için tek tek uğraşıldığı düşünülmektedir. Aslında her durumda öyle değildir. Örneğin; bir kolon burç kaydırılacaksa o kolon burç ile alakalı bütün yüzeyler seçilip tek seferde kaydırılabilir. Linkli çalışmada ki gibi herhangi bir hata vermez çünkü unsurların hiçbiri birbiri ile bağlantılı değildir.

4.6 Vısıcad Programının Görsel Ve Kullanım Kolaylığı Yönleri Ele Alınarak Değerlendirilmesi

Visicad programı kullanıcı gözüyle değerlendirildikten sonra varılan düşünceler aşağıda maddeler halinde sunulmuştur;

- Programın; pratik, anlaşılabilir, kullanımı kolay bir arayüze sahip olduğu görülmüştür. Komutların onaylanması, komut pencerelerinin yönlendirmeleri yeterli ve anlaşılır bulunmuştur. Komut ikonları görsel açıdan yeterli ve işlevleriyle uyumludur.



Şekil 59: Visicad Programı Ana Ekranı

- Modüller arası kolay geçiş sağlanabilmesinin artı bir özellik olduğu görülmüştür.

Şekil 60: Visicad Programı Modül Geçişi

- Yüzey komutlarında programın yetersiz kaldığı kanısına varılmıştır ve linkli çalışma yapılamaz.

5. CATIA

5.1 Tasarım Sürecinde İzlenen Yolun Ve Tasarım Aşamalarının İncelenmesi

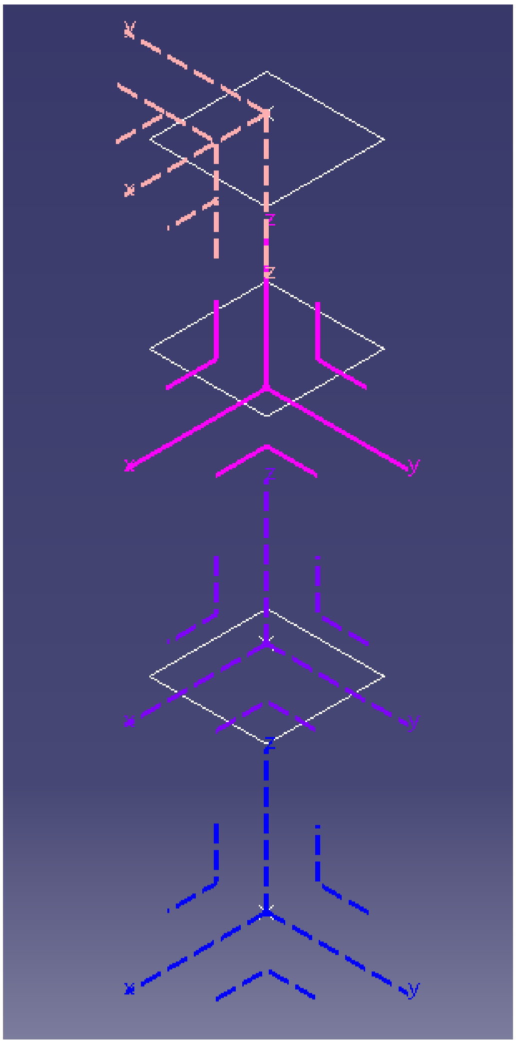
Tasarım süreci; ürün ağacı yapısının oluşturulması, preslerin konumlandırılması vb. aşamalara bölünerek anlatılmaya çalışılmıştır. Raporun bu bölümünde tasarım aşamaları hakkında detaylı bilgi verilmiştir.

5.1.1 Ürün Ağacı Yapısının Oluşturulması

Tasarıma ürün ağacı yapısının oluşturulması ile başlanmıştır. Başlangıçta aşağıdaki resimde görüldüğü gibi döküm gruplarına göre ürün ağacı yapısı oluşturulmuştur.

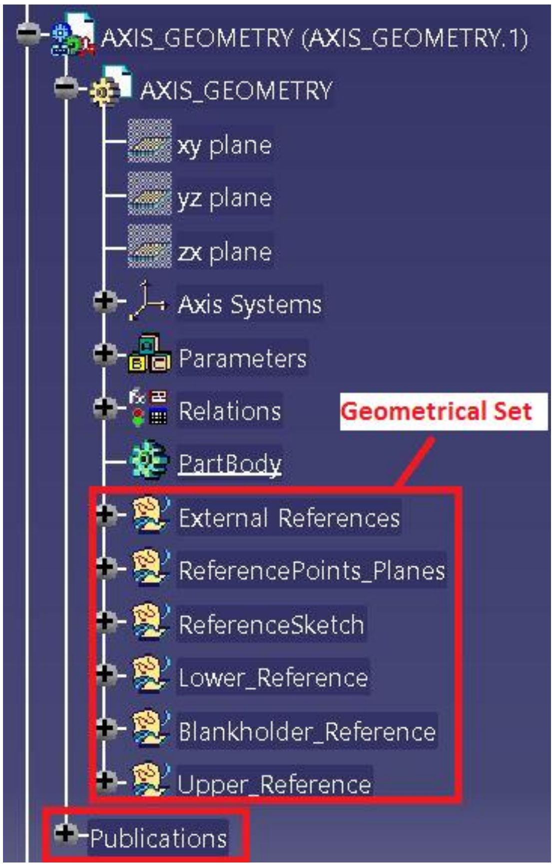
Şekil 61: Tasarım Ürün Ağaç Yapısı (Catia)

Catia'da bulunan Part, Product bileşenleri kullanılarak ağaç yapısı oluşturulmuştur. 'Axis Geometry' adı verilen part bileşeninde; Alt Döküm Noktası, Üst Döküm Noktası, Pot Çemberi Noktası adlarında noktalar atanmış bu noktalara da eksenler tanımlanmıştır. Her bir eksen arası mesafe için bu part bileşeni içinde parametreler tanımlanmıştır. Kalıp bu eksenler kullanılarak tasarlanmıştır. Eksenler dışında Alt Döküm Düzlemi, Üst Döküm Düzlemi ve Ana Merkezleme Yükseklik olmak üzere 3 adet düzlem tanımlanmıştır. Eksen ve düzlemler Şekil 62'de ki gibidir;



Şekil 62: Kalıp eksenleri ve düzlemleri

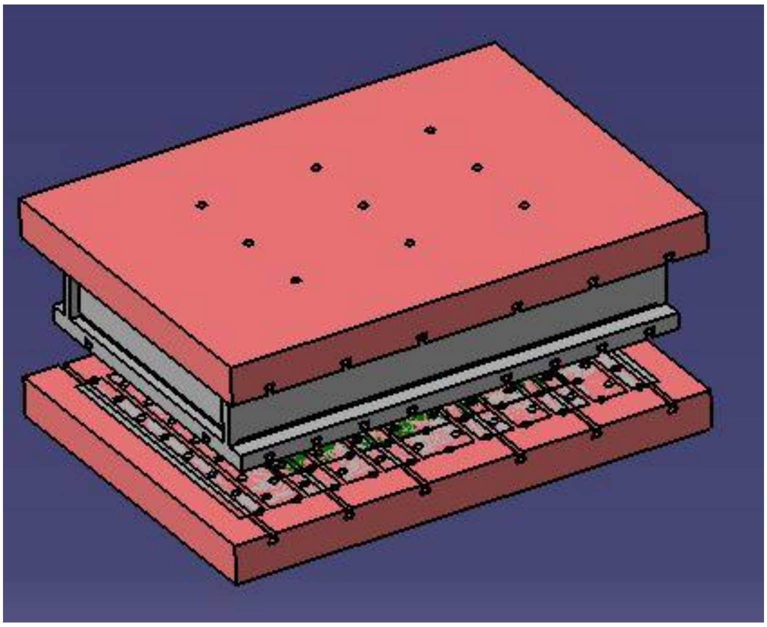
’Axis Geometry’ bileşeninin içinde her bir döküm grubu için bir ‘Geometrical Set’ ve referans ‘Geometrical Set’ ler oluşturulmuştur. Bu bileşenler içinde döküm gruplarına va kalıba ait referans çizimler, noktalar ve düzlemler oluşturulacaktır. Oluşturulan çizimler, noktalar, düzlemler Catia pogramında bulunan ’Publication’ komutu kullanılarak diğer bileşenlerde bağımlı olarak kullanılma özelliği verilecek (yayınlanma) böylece linkli çalışma sağlanmış olacaktır. Publication edilen referanslar kalıp içerisinde ilgili yerlere linkli olarak yapıştırılacaktır, böylece kalıbın bütün bölümlerinde aynı bilgi kullanılmış ve değişikliklere müdahale edilmesi kolaylaşmış olacaktır.



Şekil 63: Publication ve Geometrical Set Unsurlarının Ağaçtaki Görünümü

5.1.2 Preslerin Konumlandırılması

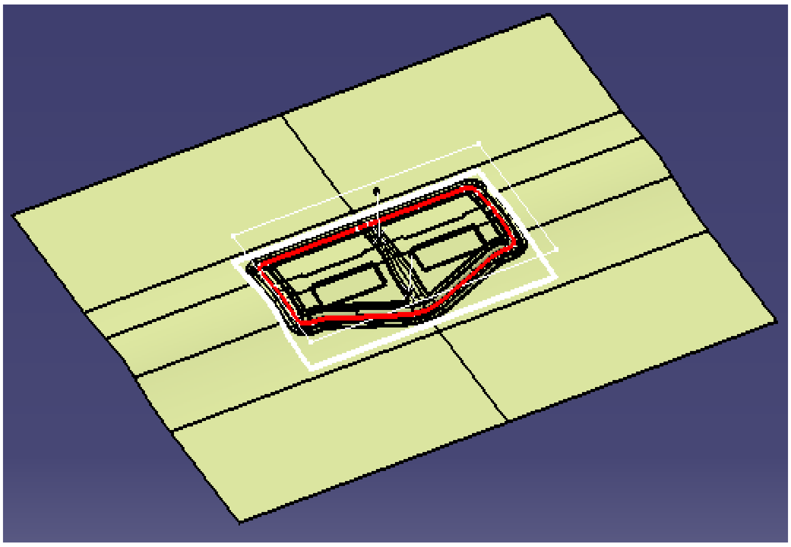
Pres dataları, Catia içerisine sürüklenerek açılmıştır, 'Copy-Paste' komutları kullanılarak ürün ağacında 'Press' bileşeni içine alınmıştır. Presler kalıp merkezinde (0.0.0 noktası) olacak şekilde konumlanmıştır. Konumlama için 'Offset Constraint' montaj taşıma komutu kullanılmıştır. Preslerin program içerisinde konumlanma görüntüsü Şekil 64'de ki gibidir;



Şekil 64: Catia Programında Preslerin Konumlandırılması

5.1.3 Proses Bileşeninin İçine Metod Dosyasının Yerleştirilmesi

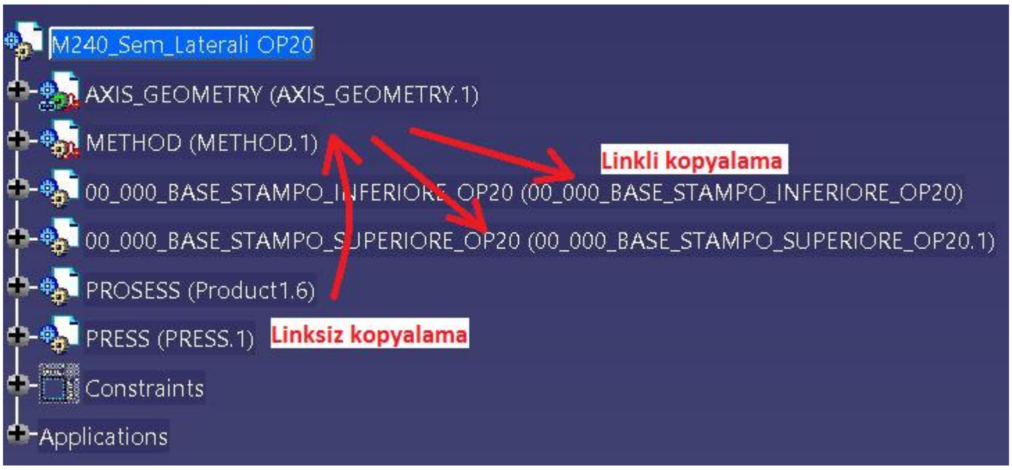
Simülasyon çalışması ile elde edilen, içerisinde parçayla ilgili tasarım işlemlerinde kullanacağımız kontur, açınım, çekilmiş sac konturu, yüzeyler vb. bilgilerin bulunduğu metod dosyası, kalıp içerisinde konumlandırılmıştır. Bu dosya tam kalıp merkezine (0.0.0 noktasında) konumlandırılmalıdır. Eğer parça açınımı simetrik değilse x ya da y doğrultusunda kaydırma yapılabilir. Konumlanan 'Metod' dosyası 'Fix Component' komutu ile sabitlenmiştir.



Şekil 65: Parçaya Ait Konumlandırılmış Metod Dosyası

Metod dosyası konumlandırıldıktan sonra metod dosyası içindeki yüzeylerde bir hata olup olmadığı kontrol edilmiştir. Bu noktada yüzeyler test edilirken yüzey modülü ve komutları sıkça kullanılmıştır. Yüzeyde boşluk olup olmadığını test etmek için Boundary komutu kullanılmıştır.(Create Datum komutu açık bırakılmıştır.) Bir boşluk saptandığında ‘Join’, ‘Fill’, ‘Disassemble’ komutları kullanılarak giderilmeye çalışılmıştır. Yüzeyı uzatmak gerektiği durumlarda ise ‘Extrapolate’, ‘Boundary’ komutları kullanılmıştır.

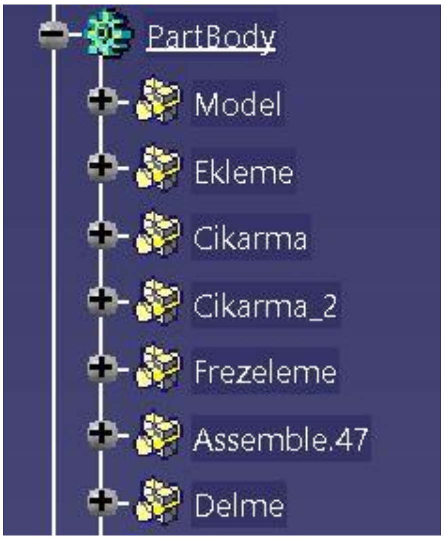
Metod dosyası içindeki bilgiler ürün ağacında bulunan ‘Metod’ isimli bileşen içerisine linksiz bir şekilde ‘Paste Special → As Result’ seçeneği ile yapıştırılmıştır. Bu bilgiler; kontur, açınım, çekilmiş sac konturu ve yüzeylerdir. ‘Metod’ bileşeni içine alınan yüzeyler gerekli öteleme değerlerinde ötelenmiştir. Bu yüzeyler kalıp tasarımında kullanılmıştır.



Şekil 66: Catia Programında Veri Alışverişi

5.1.4 Döküm Gruplarının Tasarlanması

Döküm gruplarının oluşturulmasına başlamadan önce her bir döküm parçası içinde Model, Ekleme, Çıkarma, Frezeleme, Delme isimleriyle body yapısı açılmış, bu bodyler döküm bileşeninin içerisine ‘Assemble’ komutu ile eklenmiştir. Açılan part dosyasının içinde ‘Insert → New Body’ komutu ile body oluşturulmuştur. Body içerisinde çizimimizi tamamladıktan sonra Partbody yapımızın içine ‘Assemble’ komutu ile eklenmiştir. Bizim çizimimiz kalıpla ilgili ana bir çizim ise Model grubu içine, bir delik ise delme grubuna, frezeleme işlemiyse frezeleme grubu içine eklememiz gerekmektedir. Part bileşeni içerisinde ki bu yapı aşağıdaki gibidir;



Şekil 67: Part Bileşeni İçinde ki Assembly Yapısı

5.1.4.1 Alt Dökümün Tasarlanması

Alt döküm parçasının tasarım işlemleri gruplandırılarak açıklanmaya çalışılmıştır.

5.1.4.1.1 Alt Döküm Tablasının Oluşturulması

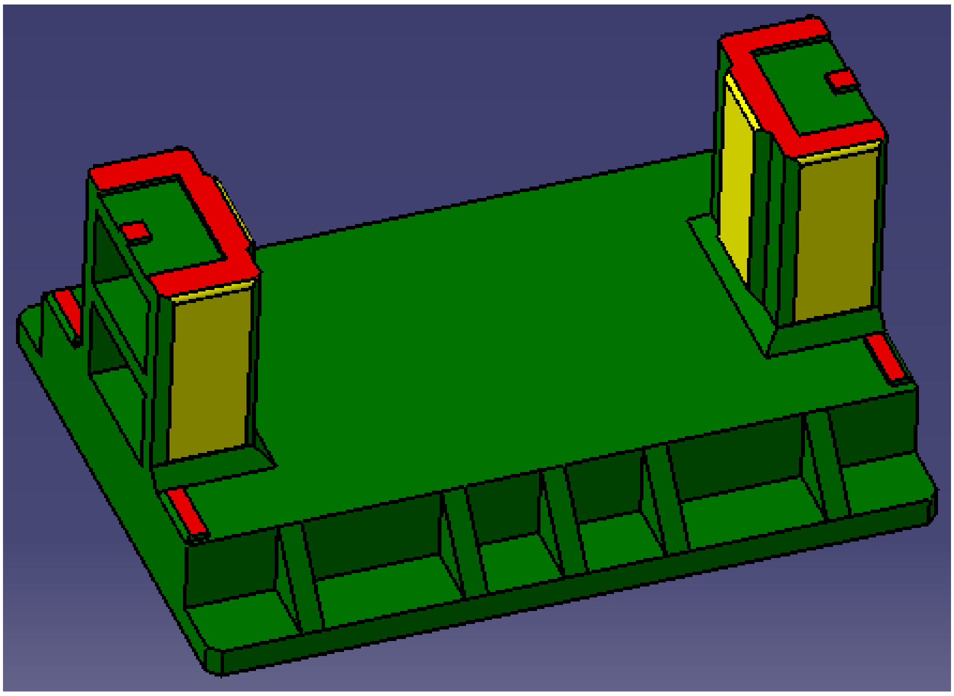
'Axis Geometry' bileşeni içinde ki kalıp ekseni, alt kalıp ekseni, ilgili düzlemler alt döküm içine alınmıştır.

Axis Geometry bileşeni içinde 'Kalıp Ebatları' adında bir çizim oluşturulmuştur. Bu çizim kalıbın en boy bilgisinden oluşmuştur. Bu değerlere, bileşen içinde 'Formula' komutu ile parametre bilgisi atanmıştır. Bu atamadan sonra kalıbın en ve boy bilgisi ilgili parametrelerden değiştirilebilecektir. Kalıp Ebatları çizimi 'Publication' komutu ile linkli çalışılabilir hale getirilmiş ve alt gruba ait döküm parçasının içine linkli şekilde kopyalanmıştır. Bu çizim referans alınarak alt döküm parçasının tablası, bu tabla üstüne ikinci bir tabla tasarlanmıştır. Bu çizimler 'Assemble' komutu ile Model grubu içerisine eklenmiştir.

Tabla çiziminden sonra kule ve kızakların oluşturulmasına geçilmiştir. Kule ve kızaklar için 'Axis Geometry' bileşeni içinde kule çizimi ve kule yükseklik düzlemi oluşturulup 'Publication' komutu uygulandıktan sonra alt döküm parçası içine linkli olarak yapıştırılmıştır. Bu referans çizim ve düzlem kullanılarak kuleler uygun ölçülerde tasarlanmıştır. Kulelerin dışarısında bulunan kızaklar, kule boşaltıları oluşturulmuştur. Kulelerin üstünde ki merkezleme elemanı oluşturulmuştur. Çizimler, Model grubu içerisine 'Assemble' komutu ile eklenmiştir.

Alt döküm ve pot çemberi dökümü arasında bulunacak oturma yükseltileri için; kulelerin oluşturulmasında olduğu gibi 'Axis Geometry' içinde çizimi oluşturulmuştur. Bu çizim publication yapıldıktan sonra alt döküm içerisine linkli olarak kopyalanmıştır. Alt döküm içerisinde linkli yapştırılan çizim referans alınarak oturma yükseltileri tasarlanmıştır. Tablanın altındaki boşaltı, kalıp ebatları referans alınarak oluşturulmuştur. Alt döküm tablası üstündeki federler oluşturulmuştur.(Federlerin konumları daha sonra değiştirilecektir.)

Alt tabla, kule ve kızakların tasarım görseli Şekil 68'de ki gibidir:

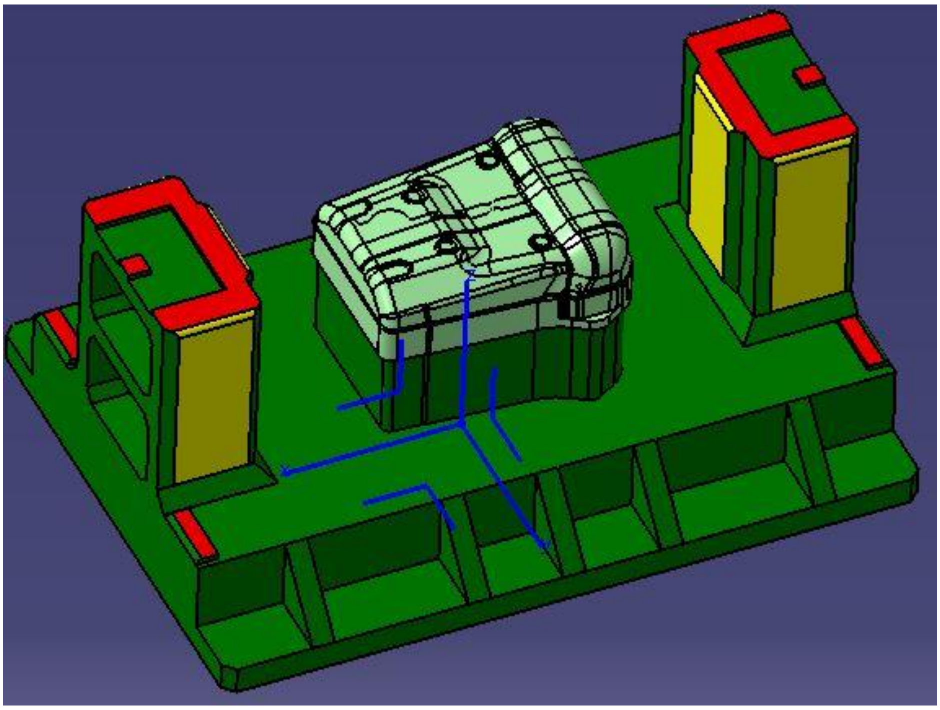


Şekil 68: Alt Tabla Tasarımı

5.1.4.1.2 Erkek Yapısının Oluşturulması

'Metod' bileşeni içerisine linksiz yapıştırılan kontur unsuruna 'Publication' komutunu uyguladıktan sonra linkli olarak alt döküm parçasına yapıştırılmıştır. Alt döküm parçası içinde kontur unsuru kullanılarak çizim oluşturulmuş ve 'Pad' komutu ile katılaştırılmıştır. Daha sonra 'Split' komutu ile kesilmiştir. Başka bir çizimde kontur ve dışına çizilen bir dikdörten ile boşaltma yapılmış ve bu çalışma kabuk yüzey ile kesilmiştir. Kontur 50 mm içeri yönde ötelenmiş, 'Pocket' komutu ile boşaltma yapılmış ve öteleme yüzeyiyle kesilmiştir. Böylelikle erkek yapısı oluşturulmuştur.

Erkek grubunun tasarım görseli Şekil 69'da ki gibidir:

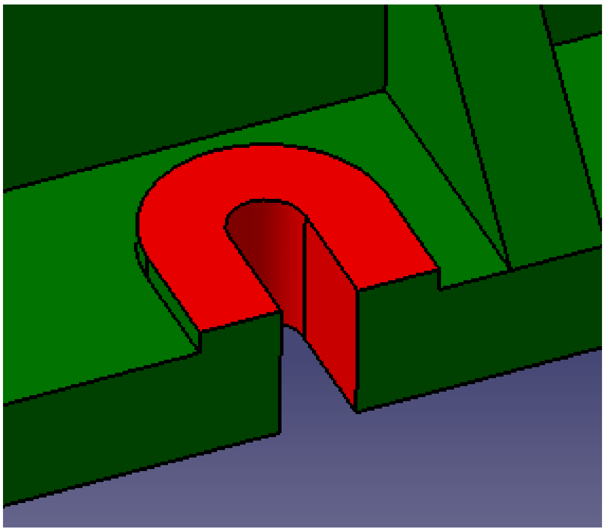


Şekil 69: Erkek Grubunun Tasarlanması

5.1.4.1.3 Pres Bağlantılarının Tasarlanması

Pres bağlantıları için ‘Axis Geometry’ bileşeni içinde alt ve üst kalıp grubu için bağlantı noktalarının bulunduğu iki adet çizim oluşturulmuştur. Alt döküm çizimi linkli olarak alt döküm part bileşeni içine alınmıştır. Pres bağlantısı için bir adet ekleme kalıbın dışında çizilmiş, katılaştırılmıştır. Daha sonra bu unsur ‘User Pattern’ komutu ile çoğaltılmıştır. Aynı yöntem izlenerek pres bağlantısının iç kısımdaki boşaltması ve üst frezeleme kısmı tasarlanıp çoğaltılmıştır.

Bu aşamada pres bağlantılarının tasarımı ve noktalarının bulunduğu çizim oluşturulmuştur. Tasarım ilerleyip presler tasarım içerisine alındıktan sonra bağlantı noktaları pres bilgileri dikkate alınarak ‘Axis Geometry’ içerisinde oluşturduğumuz ilgili çizimden değiştirilecektir. Çizim içinde yaptığımız bu değişiklik linkli çalışma yapıldığı için kalıba yansıyacaktır.



Şekil 70: Pres Bağlantısı

Mapaların oluşturulmasında 'Axis Geometry' içinde her bir döküm grubu için ayrı olarak mapaya ait çizim ve düzlem oluşurulmuştur. Bu düzlem ve çizim alt döküm içine alınmıştır. 'Shaft' komutu ile katılaştırma yapıldıktan sonra oluşturulan unsur 'Mirror' komutu ile aynalanmıştır.

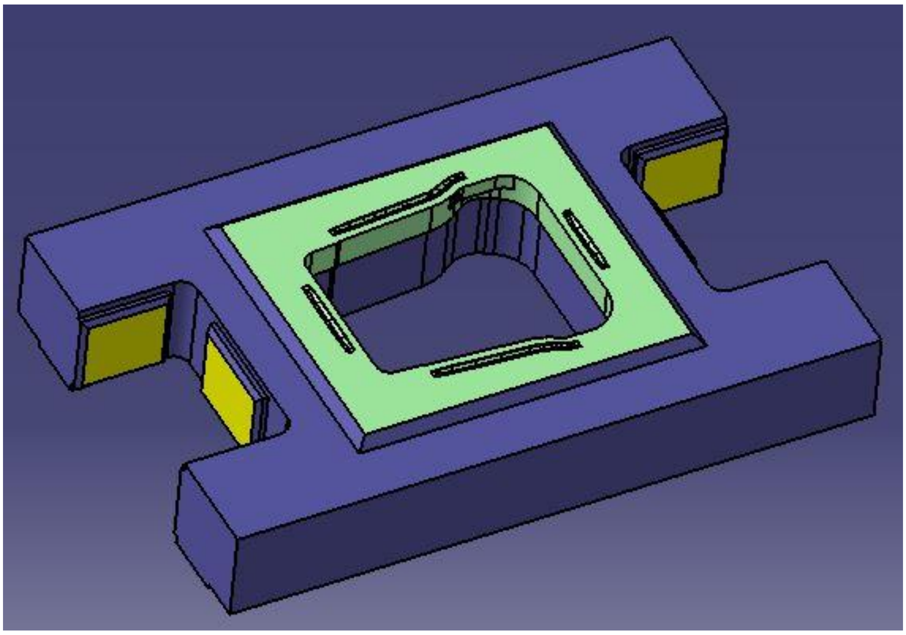
5.1.4.2 Pot Çemberinin Tasarlanması

'Axis Geometry' bileşeni içinden Kalıp Ebatları çizimi, kalıp ekseni, pot çemberi ekseni, kule çizimleri, pot çemberi boşaltma yüzeyi linkli olarak alınmıştır. Kalıp ebatları çizimi referans alınarak tabla oluşturulmuştur, yüzey ile kesilmiştir. Kule çizimleri referans alınarak kule boşaltıları oluşturulmuştur.

Açınım 'Axis Geometry' içerisinden alınıp 30 mm dışa doğru ötelenmiş, katılaştırılmış ve pot çemberi yüzeyi ile kesilmiştir. Kontur aynı yöntemle alınmış, dışa doğru 10 mm ötelenmiş, boşaltma operasyonu uygulanmış ve 'Metod' bileşeni içinden alınan kabuk yüzeyi ile kesilmiştir. Kontur 2 mm dış yöne ötelenmiş ve rastgele bir ölçüde boşaltma yapılmıştır.

Alt dökümde olduğu gibi mapalar ve sürtünme plakaları, taşıma yükseltileri oluşturulmuştur.

Pot çemberinin alt boşaltısı oluşturulmuştur.

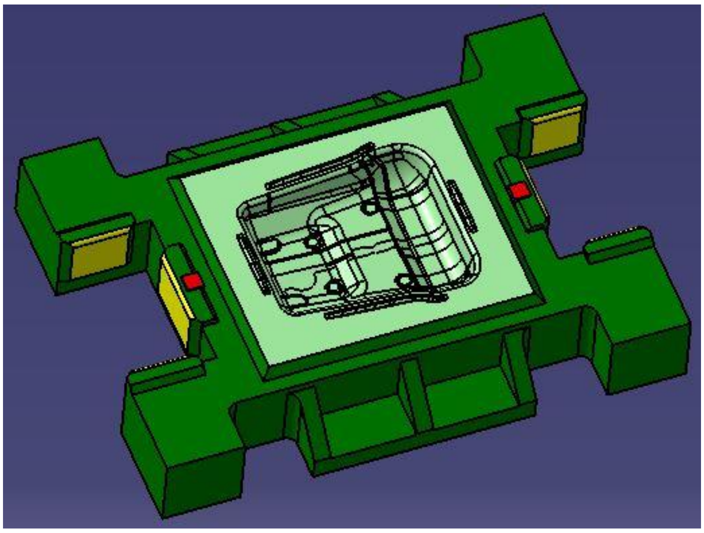


Şekil 71: Genel Hatlarıyla Pot Çemberi

5.1.4.3 Üst Dökümün Tasarlanması:

'Axis Geometry' bileşeni içinden kalıp ebatları çizimi, üst kalıp ekseni, yüzeyler üst döküm içine alınmıştır. Bu bileşenler kullanılarak tabla ve kızak boşaltıları, tabla federleri, sürtünme boşlukları ve plakaları, merkezleme elemanları oluşturulmuştur.

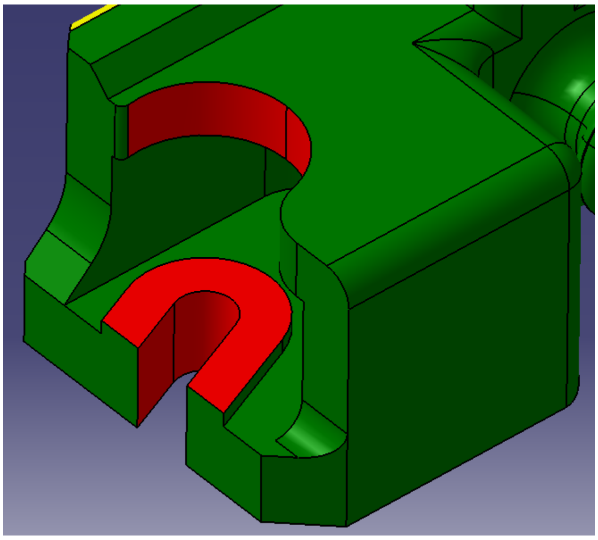
Açınım pot çemberinde olduğu gibi 30 mm dışa ötelenmiş ve kalıp yüzeyi ile kesilmiştir. Tabla altındaki boşaltı oluşturulmuştur.



Şekil 72: Genel Hatlarıyla Üst Döküm

5.1.4.3.1 Pres Bağlantılarının Tasarlanması

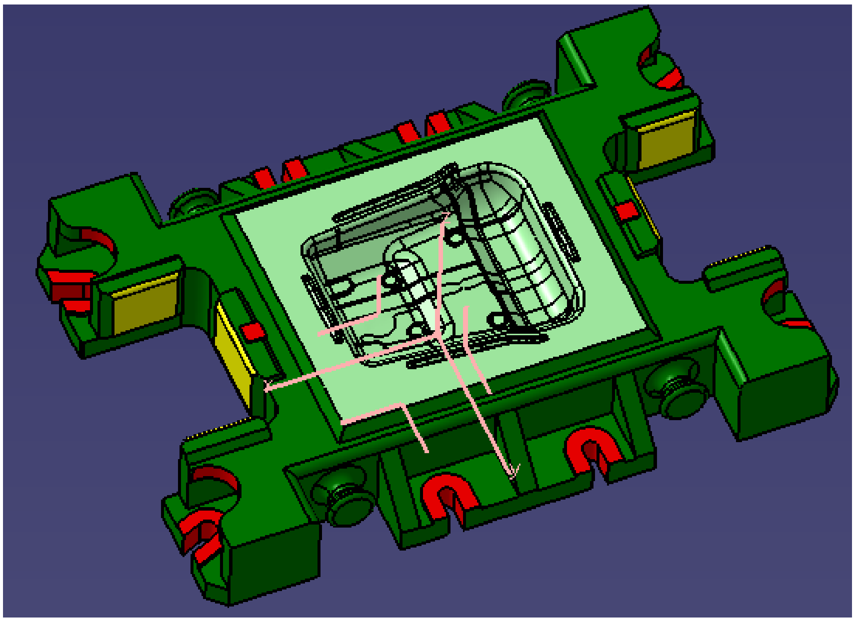
Üst dökümde 2 farklı pres bağlantısı bulunmaktadır. Bağlantılardan biri açılı diğeri düzdür. İki pres bağlantısında da alt döküm bağlantılarında izlenen tasarım yöntemi izlenmiştir.



Şekil 73: Üst Dökümde ki Açılı Pres Bağlantısı

5.1.4.3.2 Mapa Eklenmesi

Mapalar alt döküm ve pot çemberi dökümünde olduğu gibi tasarlanmıştır. Mapalar eklendikten sonra tablanın alt boşaltısı oluşturulmuştur.

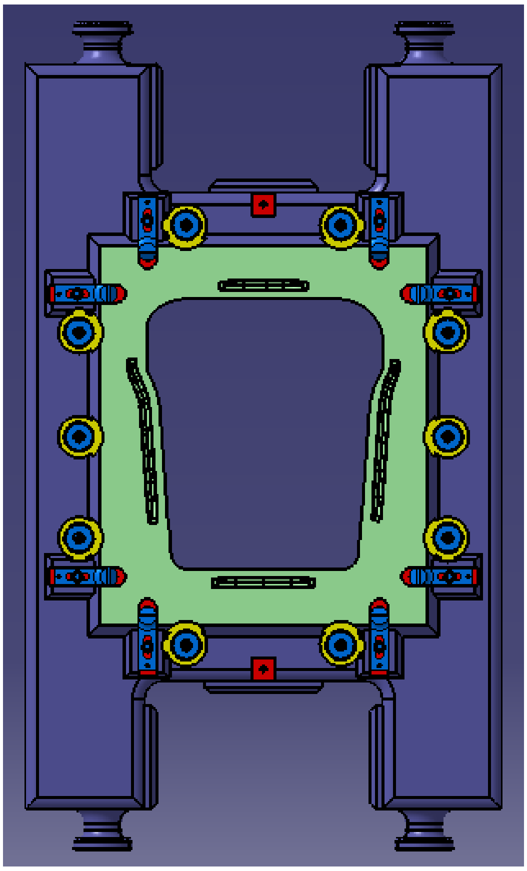


Şekil 74: Genel Hatlarıyla Üst Döküm

5.1.5 Stoper Ve Dayamaların Konumlandırılması

Kalıp döküm grupları genel hatlarıyla oluşturulduktan sonra kalıp ebatlarını belirlemek amacıyla stoper ve dayama standart kalıp elemanlarının konumlandırılması gerekmektedir. Elemanlar kalıp tasarımı kurallarına göre yerleştirilmiştir. Standart parçaların kalıba yerleştirilmesi için gerekli yükselti, boşaltı, delik vb. unsurları bulunmaktadır. Kullanılan standart parçaların içerisinde bu unsurlar çizilmiş ve publication edilmiş halde bulunmaktadır. Ekleme yapmak istediğimiz döküm parçası içerisine publication edilmiş unsur (boşaltı, ekleme,delik vb.) 'Assemble' komutu ile alınarak direk döküm içerisine eklenebilir. Bu aşamada 'Standart Kalıp Dias' programından destek alınmıştır. Catia ile birlikte çalışan bu program ile standart kalıp elemanlarının parametrik tasarımlarına ulaşmak mümkündür.

Stoper ve L dayamalar kalıp içinde konumlandıktan sonra kalıp ebatları kalıp tasarımı kuralları dikkate alınarak belirlenmiştir.

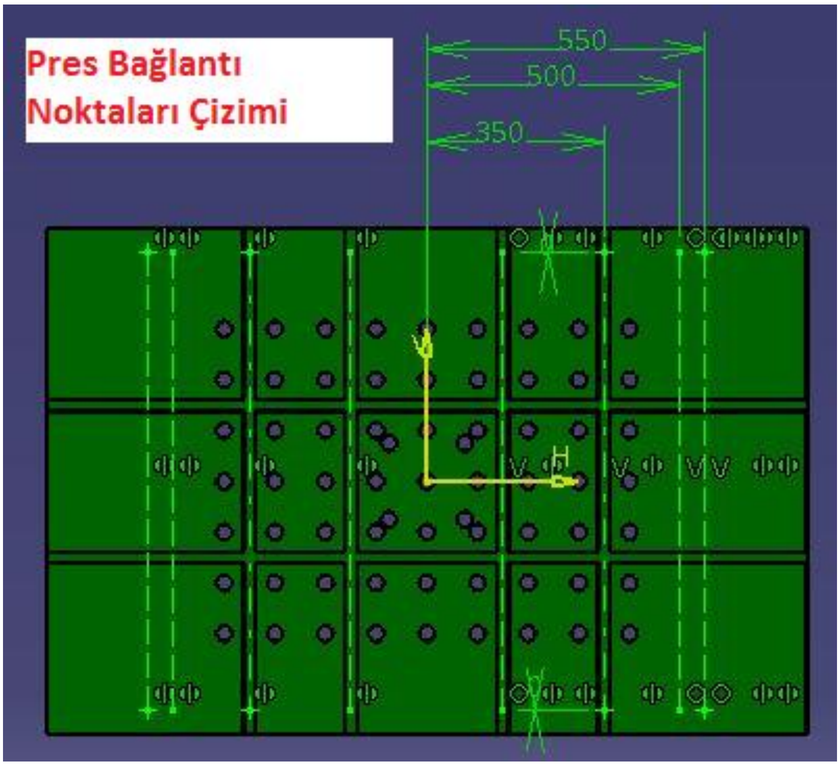


Şekil 75: Stoper ve Dayamaların Konumlandırılması

5.1.6 Pres Bağlantılarının Belirlenmesi

Kalıp ebatları belirlendikten sonra preslerin bağlanabilmesi için noktalar düzenlenmiştir. ‘Axis Geometry’ bileşeni içinden pres noktalarının bulunduğu çizimlerde nokta ölçüleri değiştirilmiş, kalıp üstüne değişiklik aktarımı gerçekleştirilmiş olmuştur.

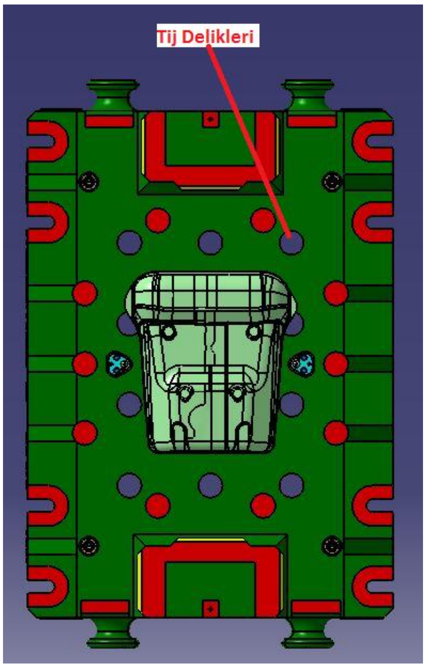
Pres noktaları belirlendikten sonra kalıp üstünde düzenlemeler yapılmıştır. (Federlerde çarpışan kısımlar yeni pres noktalarına göre düzenlenmiştir.)



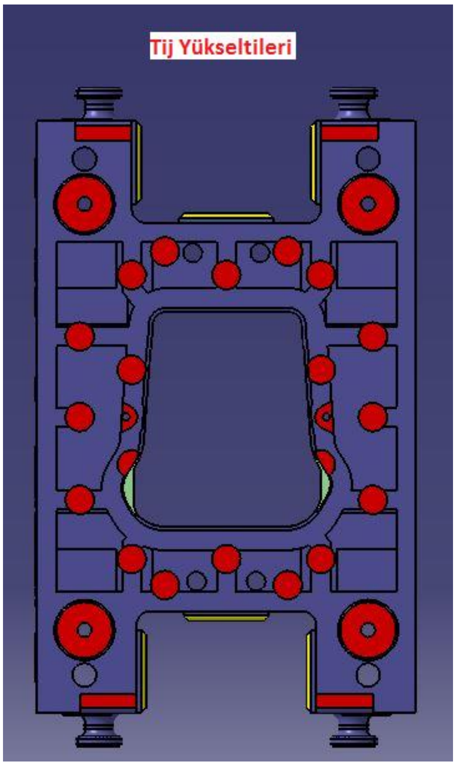
Şekil 76: Pres Bağlantı Noktalarının Çizimi

5.1.7 Tij Noktalarının Tayini

Bütün presler için kalıp tasarım kurallarına uygun olarak seçilen noktaları içeren çizimler oluşturulmuş, bu çizimler ilgili dökümlerin içerisine linkli olarak aktarılmıştır. Alt grup için bu çizimler kullanılarak tij delikleri oluşturulmuş, Pot çemberi için bu noktalar kullanılarak yükseltiler oluşturulmuştur.



Şekil 77: Alt Dökümde ki Tij Delikleri



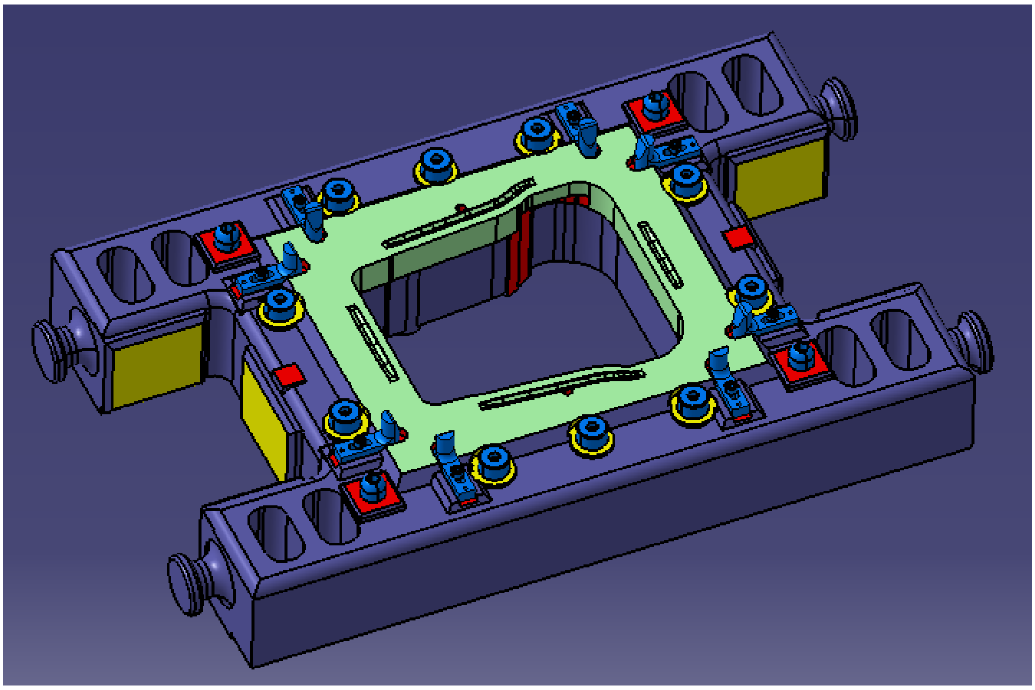
Şekil 78: Pot Çemberindeki Tij Eklemeleri

5.1.8 Diğer Standart Parçaların Tasarıma Eklenmesi

Geriye kalan bütün standart parçalar tasarım kurallarına göre kalıp içerisine alınmış, konumlandırılmış, stoper parçasında olduğu gibi eklemeleri, boşaltmaları, delikleri ilgili dökümlere atılmıştır.

Tasarım içine alınan standart parçalar aşağıdaki gibidir;

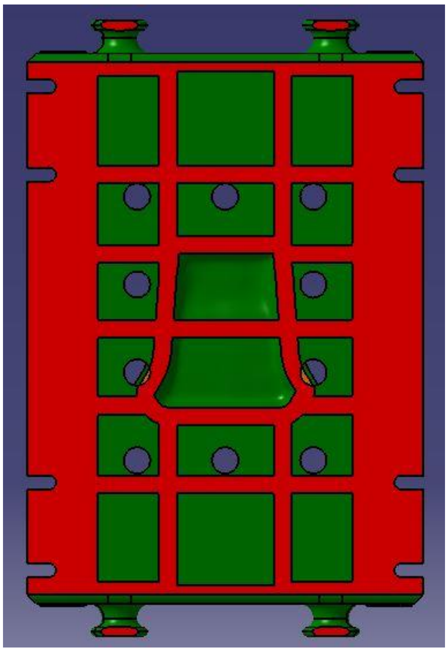
- Merkezleme Pimi
- İz Zımbası
- Askı Cıvatası
- Lazer Parçası



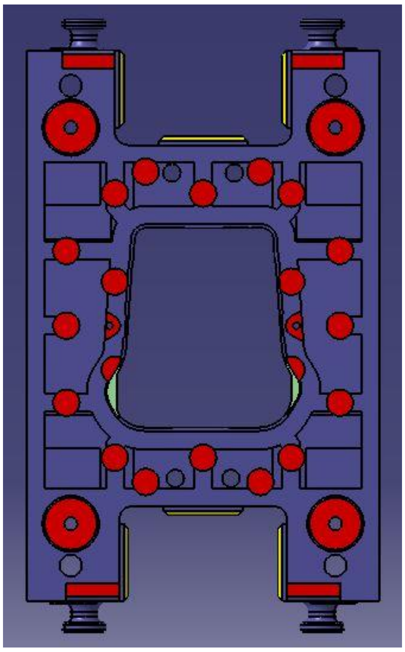
Şekil 79: Standart Parçaların Döküme Eklenmesi

5.1.9 Döküm Feder Yapılarının Oluşturulması

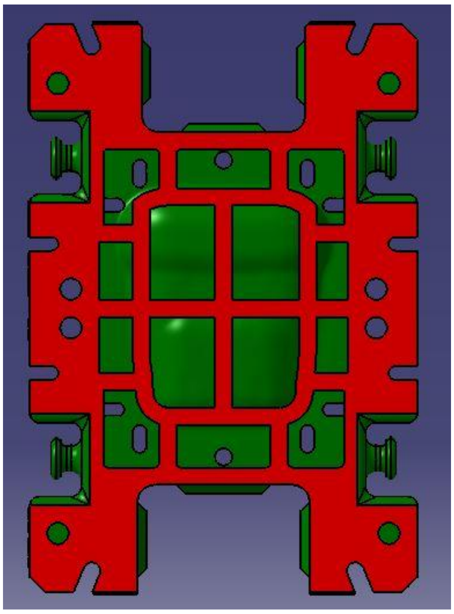
Tij noktaları , stoper konumları da belirlendikten sonra döküm gruplarının tablalarının alt kısmındaki federlerin oluşturulması işlemine geçilmiştir. Stoper elemanı için alt gruba destek eklemeleri tasarlanmış, pot çemberinin tabla boşaltısına ekleme yapılmıştır.



Şekil 80: Alt Döküm Feder Yapısı



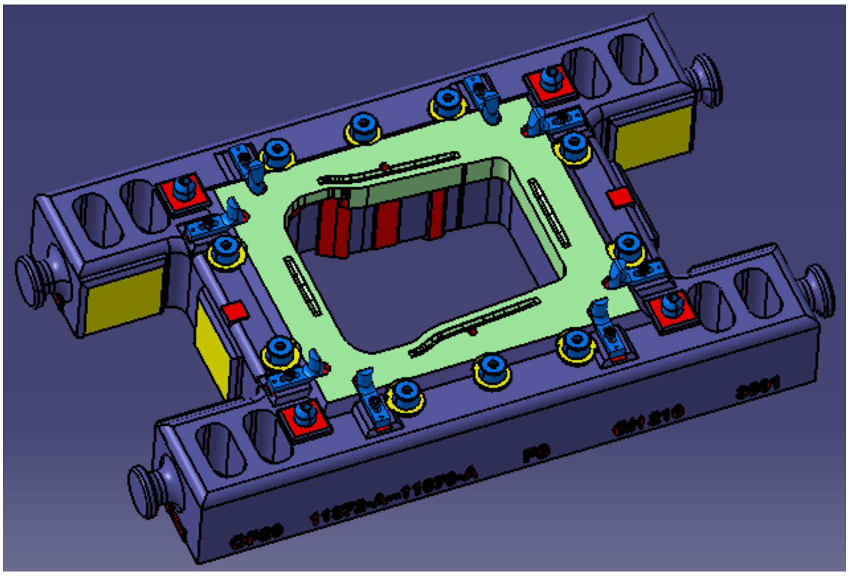
Şekil 81: Pot Çemberi Feder Yapısı



Şekil 82: Üst Döküm Feder Yapısı

5.1.10 Kalıp Yazılarının Ve Oklarının Eklenmesi

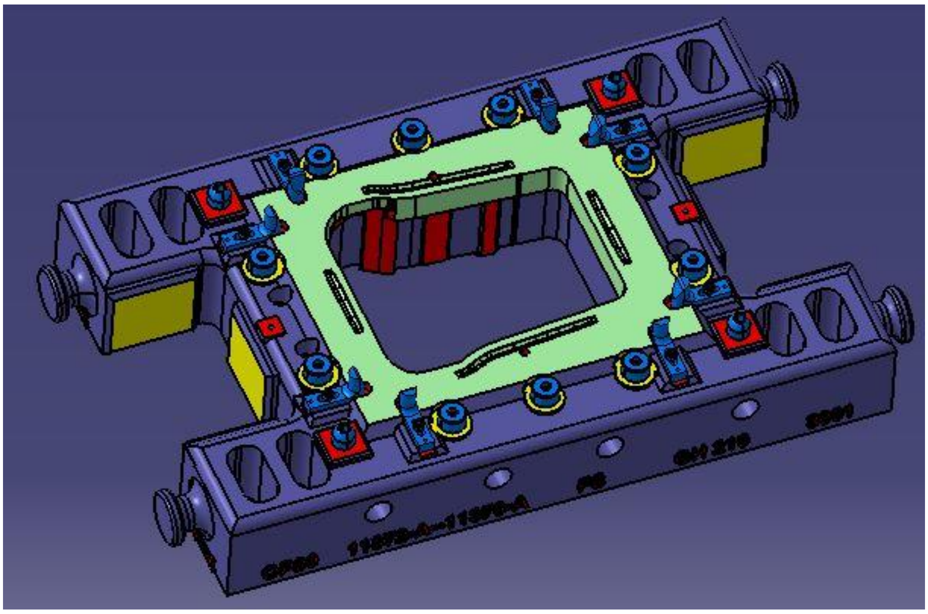
Kalıp yazıları için yardımcı bir Drawing ve Dxf dosyasından yararlanılmıştır. Catia içerisine çağrılan Dxf dosyası üstündeki bilgiler çizim içerisine yapıştırılmış ve oluşturulan çizim katılaştırılmıştır.



Şekil 83: Kalıp Yazılarının ve Okların Tasarıma Eklenmesi

5.1.11 Boşaltma Deliklerinin Açılması

Tasarımda ağırlık azaltma çalışmaları kapsamında döküm gruplarında uygun görülen kısımlara belirli çap değerlerinde delikler açılmıştır.



Şekil 84: Pot Çemberindeki Boşaltma Delikleri

5.1.12 Kalıp Kontrollerinin Yapılması

Tasarım tamamlandıktan sonra bazı kalıp kontrollerinin uygulanması gerekmektedir. Bu kontroller şunlardır;

Kesit kontrolü

Renklendirme Kontolü

Standart Parça Kontrolü

Clash Analysis

5.2 Tasarımda Kullanılan Komutların Saptanması Ve Kullanım Sıklığı Fazla Olan Komutların Belirlenmesi

Belirlenen bir döküm kalıbın Catia programında tasarımının yapılması çalışmasında kaydedilen videolar incelenmiştir. Tasarım esnasında kullanılan bütün komutlar saptanmıştır. Bu komut listesi aşağıdaki gibidir;

- New Part
- New Product
- Geometrical Set

- Point
- Plane
- Axis System
- Publication
- Copy
- Paste
- Offset Constraint
- Measure Between
- Measure Item
- Fix Component
- Kumpas (Komut değil)
- Join
- Disassamble
- Boundary
- Extrapolate
- Fill
- Paste Special
- Formula
- Rectangle
- Sketch
- Constraint
- Exit Workbench
- Pad
- Pocket
- Hole
- Chamfer
- Graphic Properties
- Painter
- Split
- Project 3D Elements
- Offset
- Elongated Hole

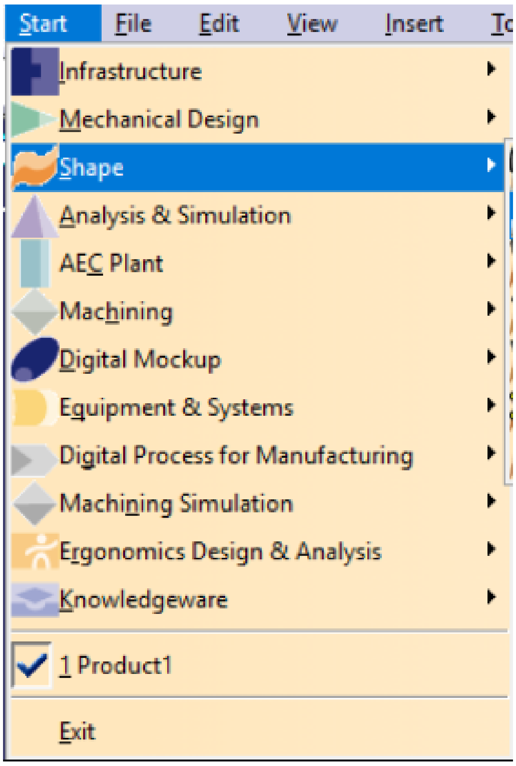
- Line
- Body
- İnstantiate From Document
- Mirror
- Fix together
- Circle
- Assemble
- Add
- Edge Fillet
- Thickness
- Remove Face
- Translation
- Symmetry
- User Pattern
- Normal View
- Views
- Update All
- Undo Empty Selection
- Redo
- Positioned Sketch
- Measure Inertia
- Clash
- Sectioning
- Enhanced Scene
- Graph Tree Reordering
- Coincidence Constraint
- User Pattern
- Snap
- Graphic Properties (Opacity, Color..)
- Trim
- Quick Trim
- Translate

Videolar incelenip değerlendirildiğinde sıklıkla kullanılan 5 komutun aşağıdakiler olduğu görülmektedir;

- Pad
- Constraint
- Pocket
- Rectangle
- Assemble

5.3 Tasarım Sürecinde Kullanılan Modüller

Tasarım sürecinde Catia programı içindeki Sketcher , Part Design, Assembly Design, Generative Shape Design modülleri kullanılmıştır.



Şekil 85: Catia Modül Geçişi

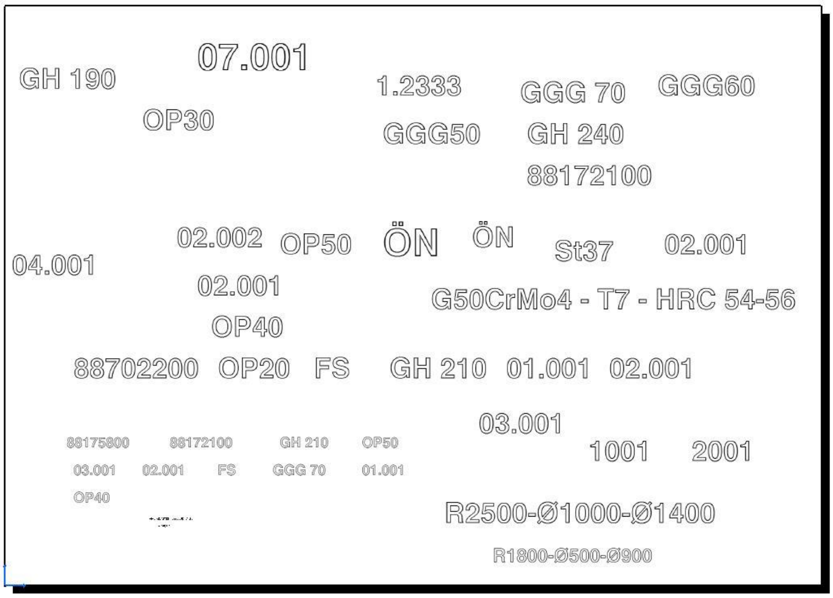
- Sketcher; tasarlamak istenilen elemanın 2 boyutlu geometrisinin çizilmesinde,
- Assembly Design Modülü; ürün ağacı yapısını oluştururken bileşen eklenmesinde, bileşenin taşınmasında, döndürülmesinde, parçaların çoğaltılmasında, standart elemanların konumlanması için ilişki tanımlanmasında, referans geometrilerin oluşturulmasında (çizgi, nokta, eksen takımı, düzlem),

- Generative Shape Design; kalıbın erkeği ve dişisini oluştururken bölme işleminde, parça yüzeylerini budama işlemlerinde,
- Part Design; 2 boyutlu çizimin 3 boyutlu hale getirilmesinde, radyus, pah kırma, çoğaltma işlemlerinde, referans geometrisi eklenmesinde kullanılmıştır.

5.4 Tasarım Sürecinde Zorlanılan Noktaların Saptanması Ve Öneriler

Solidworks programı ile yapılan tasarım çalışmasında bazı noktalarda zorlanılmış ya da tasarım esnasında bir takım öneriler getirilmiştir. Tasarım videolarının incelenmesi sonucunda ortaya konulan saptamalar aşağıda açıklanmaya çalışılmıştır. Bu saptamalar, kullanılan Solidworks programının değerlendirilmesi, tasarımcı gözüyle eleştriler, komutların kullanım kolaylığının değerlendirilmesi, kalıp tasarımında gerekli özelliklerin saptanması vb. durumların değerlendirilmesiyle ortaya konulmuştur.

- Programda kalıp yazılarının oluşturulması için ayrı bir komut yoktur. Kalıp yazılarının oluşturulması için yardımcı bir Drawing Dosyası kullanılmaktadır. Kalıp oku ise çizim ortamında oluşturulmuştur. Kalıp tasarımının bu aşamasında yazı yazma ve şekil eklemek için ayrı bir komut bulunması aradaki gereksiz işlemleri ortadan kaldırarak tasarımcının harcadığı zamanı azaltacaktır.



Şekil 86: Kalıp Yazılarının Oluşturulmasında Kullanılan Dwg Dosyası

5.5 Tasarım Çalışmasına Revizyon İşlemlerinin Uygulanması

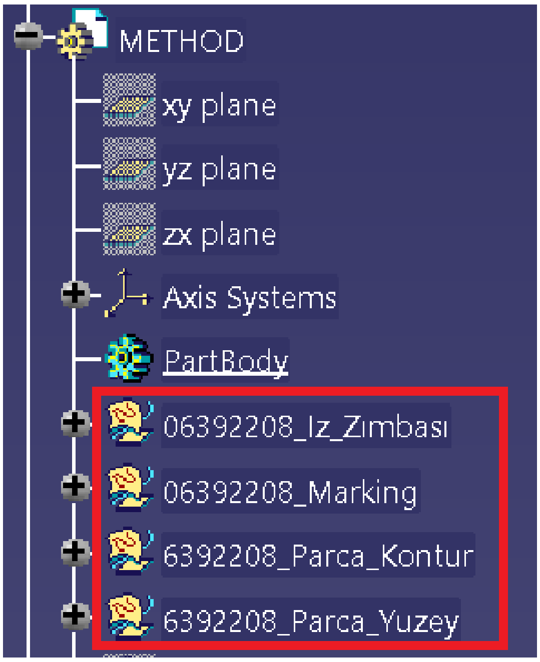
Raporun bu bölümünde kalıp tasarımına revizyon uygulanmak istenmesi ya da değişiklik yapılması ihtiyacı hissedildiği durumlarda tasarımcı tarafından değişikliklerin uygulanma kolaylığı değerlendirilmiştir.

Kalıp tasarımında kullanılan standart parçaların kalıba montajını ve konumlaması sağlayan bir çok ekleme, frezeleme, bağlantı delik bileşenleri bulunmaktadır. Çalışmada linkli çalışma esas alındığı için herhangi bir değişiklik yapılması istendiği durumda kolayca uygulanabilecektir.

Tasarımda tasarımsal bilgiler (düzlem, yükseklik, çizimler) 'Geometrical Set' adlı bir unsurun içerisinde bulunduğundan değişiklik yapmak istediğimiz çizime ya da düzleme ulaşabilmemiz kolay olacaktır.

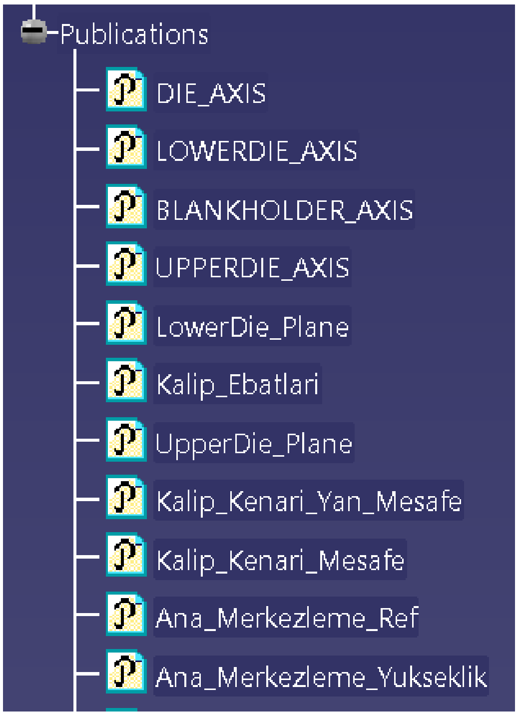
Catia programında kullanılan 'Center Graph' ve 'Parents Children 'komutları ile birlikte, ürün ağacında istediğimiz operasyona kolay ulaşabilecek ve böylelikle değişikliklerin uygulanması kolaylaşacaktır.

Catia programı ile parametrik tasarım gerçekleştirebildiğimizden kalıp ile ilgili kalıp kapalı yüksekliği, besleme yüksekliği, kalıp ebatları vb. değerlere hükmetmemiz kolay olacaktır.



Şekil 87: Catia Programı İçinde Geometrical Set Bileşeni Altında Tasarımsal Elemanların Gruplandırılması

Şekil 88: Catia Programında Parametre Atanan Değerler

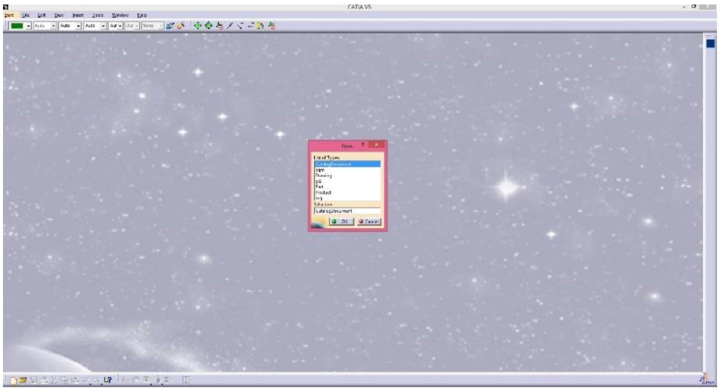


Şekil 89: Publication Komutu Uygulanmış Unsurlar

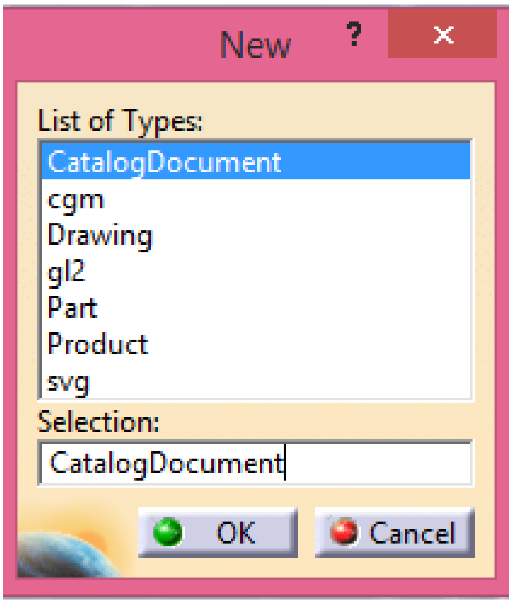
5.6 Catıa Programının Görsel Ve Kullanım Kolaylığı Yönleri Ele Alınarak Değerlendirilmesi

Catia programının kullanıcı gözüyle değerlendirildikten sonra varılan düşünceler aşağıda maddeler halinde sunulmuştur;

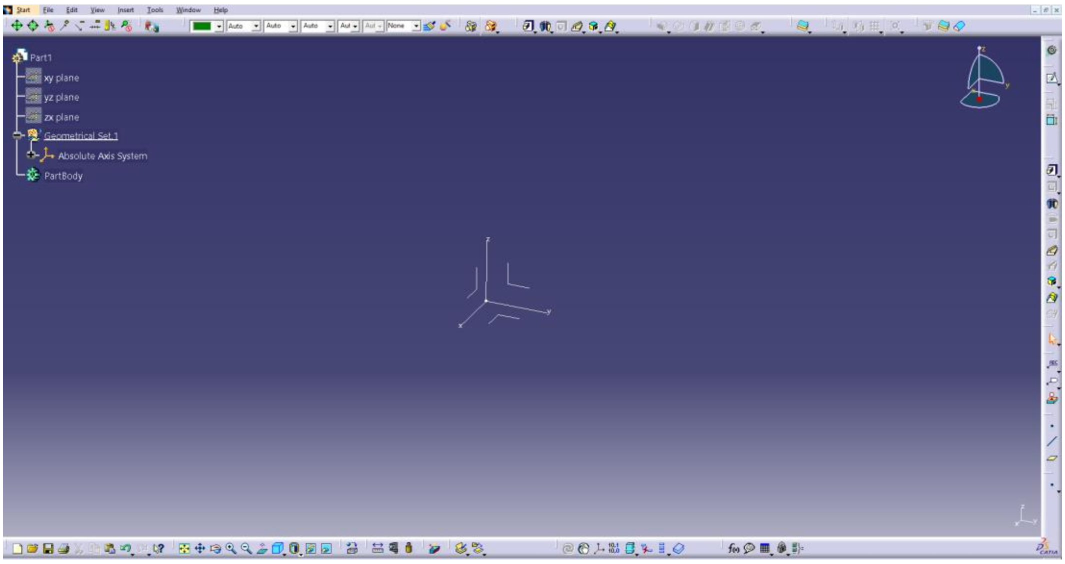
- Programın; pratik, anlaşılabilir, kullanımı kolay bir arayüze sahip olduğu görülmüştür. Çizim ortamına giriş, çıkış, komutların onaylanması, komut pencerelerinin yönlendirmeleri yeterli ve anlaşılır bulunmuştur. Komut ikonları görsel açıdan yeterli ve işlevleriyle uyumludur.



Şekil 90: Catia Programı Giriş Ekranı

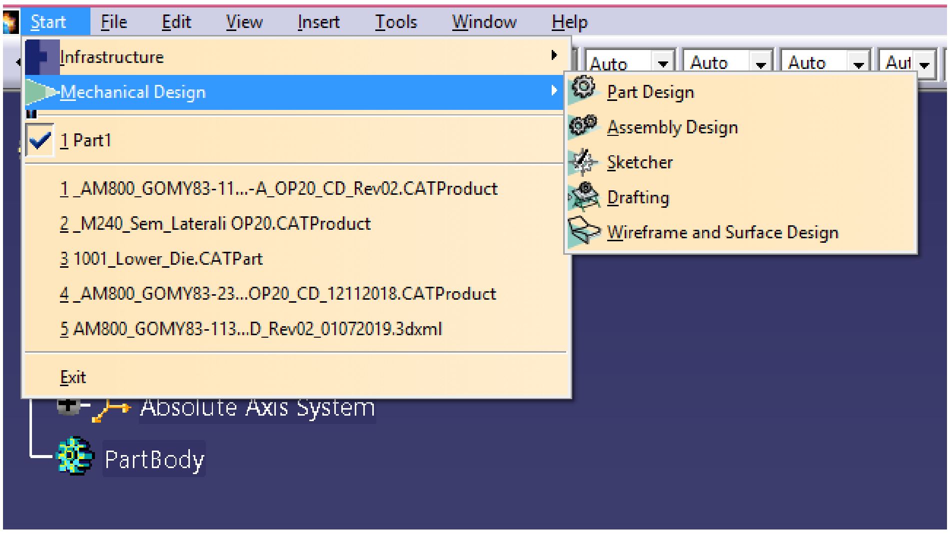


Şekil 91: Catia Programı Bileşen Açma Ekranı



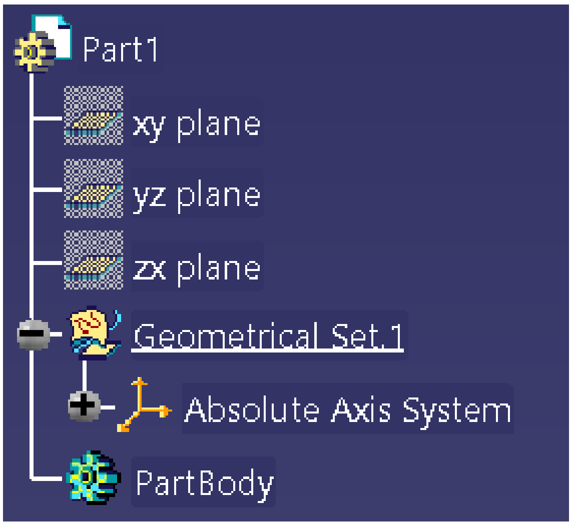
Şekil 92: Catia Programı Parça Ortamı

- Modüller arası geçişin diğer tasarım programları ile kıyaslandığında kullanışlı olmadığı görülmüştür.



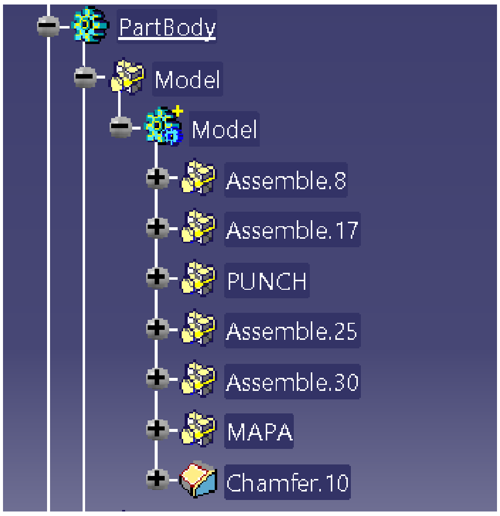
Şekil 93: Catia Programı Modül Geçişi

- Ürün ağacı yapısı işlevsel ve görselliği iyidir. Katı, yüzey vb. unsurları gruplandırabilme ve body mantığıyla çalışabilme özelliğinin bulunması tasarımı kolaylaştırmıştır.



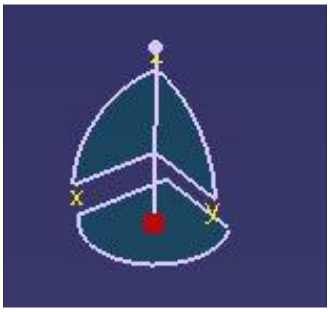
Şekil 94: Catia Programı Part Bileşeninin Ürün Ağacı Yapısı

- Ürün ağacında komutlar arasında gruplandırma yapılabilmiştir. Assemble mantığıyla çalışabildiği için komutlar arasında bir düzenleme yapılabilmiştir.



Şekil 95: Üst Döküm Parçasının Ürün Ağacı

- Programda eklenen parçaların konumlandırılması için taşıma elemanı ve komutları bulunduğu için preslerin ve metod dosyasının konumlandırılması, kalıbın açık ve kapalı durumlarının görüntülenebilmesi vb. işlemleri kolayca yapılabilmiştir. Bu işlemler için Catia'da ki kumpas elemanı bu açıdan işlevsel bulunmuştur. Kumpas elemanının dışında programın taşıma, döndürme komutları yeterli görülmüştür.



Şekil 96: Kumpas

- Yüzey komutlarının ve linkli çalışmanın sağlanması için gereken komutlar konusunda program yeterlidir. Yüzeyleri onarma konusunda yeterlidir, fakat öteleme yüzeyi oluşturma aşamasında Power Copy'den yararlanılmıştır.
- Programda kalıp üstünde ölçü alınmasını sağlayan komutlar kullanışlı bulunmuştur.

Şekil 97: Ölçüm Komutları

6. SONUÇ

	SOLIDWORKS	ÇİZEN	VISICAD	CATİA
Arayüz Değerlendirmesi	Modüller tek bir araç çubuğu üstünde yatay olarak gruplandırılmıştır. Ortam değişikliği ihtiyacı duyulmadan modüller arası geçiş kolayca yapılabilmektedir.	Tüm modüllere ait araç çubukları aynı ortamda bulunmakta, komutlara tek bir ortamdan kolay ulaşım sağlanabilmektedir.	Modüller tek bir araç çubuğu üstünde yatay olarak gruplandırılmıştır. Ortam değişikliği ihtiyacı duyulmadan modüller arası geçiş kolayca yapılabilmektedir.	Modüller arası geçiş diğer tasarım programları ile karşılaştırıldığında kullanışlı bulunmamıştır.
Ağaç Yapısı	Ürün ağacı, parça ve montaj olmak üzere iki bileşenden oluşabiliyor. Parça bileşeni gövdelere bölünerek her gövdeye ayrı işlem uygulanmasına izin veriliyor. (Diğer programlarda ki body mantığı) Gövde işlevi kullanışlı değil. Parça içinde komut, eğri,yüzey gruplandırılması yapılamıyor	Ayrı ayrı partlar oluşturularak birbirine ekleme ve birbirinden çıkarma yapılabiliyor. İçe aktarılan eğri, yüzey vb. geometriler "GeometrySet" içerisinde toplanıyor fakat gruplandırma yapılamıyor.	Ağaç yapısı yoktur. Layer sistemi vardır.	Ürün ağacı part, product, component gibi çeşitli bileşenlerden oluşabiliyor. Part bileşeni içerisinde body mantığı ile çalışılabilmesi tasarımı kolaylaştırmaktadır. Assemble komutu ile tasarım içerisinde komutlar arasında gruplandırma yapılabiliyor.
Sketch	Sketch modülü kullanışlı, sketch eksenleri	Sketch modülü kullanışlı, bazı fonksiyonlar eksiktir.	Sketch ile yapılacak işlemler kolaylıkla yapılabilmektedir.	Sketcher modülü kullanışlı bulunmuştur. Sketch

Ortamı	ölçülendirme, aynalama vb. komutlar için kullanılamıyor. (Sketch içine eksen çizgisi çizmek gerekiyor.)	(Örneğin; iz düşüm, geri alma vb.)	Ayrı bir sketch ortamı yoktur. Bütün işlemler tek bir ortamda yapılmaktadır.	eksenleri; sketch , part, product ortamında bütün komutlar için etkin olarak kullanılabiliyor.
Montaj ve Alt Montaj Dosyaları Oluşturma	Montaj dosyası içerisine sonsuz sayıda parça ve montaj dosyası açılabilir. Ürün ağacı karmaşıktır, ayırt edilebilirliği zor bir görselliğe sahip.	Montaj dosyası içerisine sonsuz sayıda parça ve montaj dosyası açılabilir. Tanıtımlarımızda görsellik konusunda olumlu görüş verenler oldu.	Montaj dosyası yoktur. Gruplandırma vardır.	Montaj dosyası içerisine sonsuz sayıda parça ve montaj dosyası açılabilir. Ürün ağacı düzenli bir görünüm ve yapıya sahip.
Kalıp Seti Oluşturma	Kalıp tasarımı yapılırken, kalıp içerisinde birden fazla yerde kullanılacak bileşenleri bir referans parçadan yayınlamak tasarım sürelerini büyük oranda kısaltmayı mümkün kılmaktadır. Solidworks'te bu özellik sketchler, eğriler vb. için mümkündür fakat düzlem bilgisi, yüzey bilgisi yayınlanamadığı için tam bir kalıp seti oluşturulamamıştır. Bileşenler eklendikten sonra her bir döküm grubu sırayla ve kısmen bağımsız olarak tasarlanmıştır	Kalıp tasarımı yapılırken, kalıp içerisinde birden fazla yerde kullanılacak bileşenleri bir referans parçadan yayınlamak tasarım sürelerini büyük oranda kısaltmayı mümkün kılmaktadır. ÇİZEN'de bu özellik olmadığı için kalıp seti oluşturulmamış ve tek tek parça dosyaları oluşturulup montaj dosyası içine alınarak tasarıma devam edilmiştir.	Kalıp tasarımı yapılırken, kalıp içerisinde birden fazla yerde kullanılacak bileşenleri bir referans parçadan yayınlamak tasarım sürelerini büyük oranda kısaltmayı mümkün kılmaktadır. VISICAD'de bu özellik olmadığı için kalıp seti oluşturulmamış bütün parçalar tek bir ortamda çizilmiştir.	Catia programında linkli ve parametrik çalışma mümkün olduğu için bileşenler kalıp içerisinde yayınlanabilmektedir. Bu sebeple Catia'da kalıp seti oluşturulabilmiştir.
Standart Parçaların Kalıp İçinde Kullanımı	Montaj ortamında parçalar eksenlere bağlanmıştır fakat linkli çalışma yapılamadığı için parçaya ait ekleme, frezeleme, delik vb. unsurların tek tek oluşturulması tasarımı zorlaştırmıştır.	Montaj ortamında standart parçaların eksen veya noktalara bağlanamaması büyük bir eksik.	Her bir standart parçaya ait işlemler tek tek oluşturulmuştur.	Standart parçalar eksen, çizgi, düzlem vb referanslara bağlanabilmekte ve bu parçaların ekleme, frezeleme gibi unsurları publication komutu ile kalıp içerisinde yayınlanabilmektedir.
Montaj Ortamında Taşıma, Döndürme İşlemleri	Montaj ortamında parçaların konumlandırılması için kullanılan taşıma/ döndürme komutu işlevsel bulunmadı. Preslerin ve metot	Montaj ortamında parçaların konumlandırılması için kullanılan İlişkilendirme operasyonlarında seçimler ve seçimlerin görünürlüğü anlamında bazı	Montaj ortamı yoktur. Bütün işlemler tek bir ortamda yapılmaktadır. Parçalar birbirine bağlantılı olmadığı için taşıma ve	Montaj ortamında parçaların konumlandırılması için kullanılan taşıma/ döndürme komutu işlevsel bulundu. Bunu dışında kumpas

	dosyasının konumlandırılmasında zorlanılmıştır.	geliştirmeler yapılmalıdır.	döndürme işlemleri kolay bir şekilde yapılabilmektedir.	elemanı ile konumlandırma ve hareket sağlanabilmektedir.
Komutların Yeterliliği ve İşlevselliği	Komutlar genel itibariye kullanışlı fakat yetersizdir. Yüzey komutları yetersiz kalmıştır.	Komutlar genel itibariye kullanışlı fakat yetersizdir.	Yüzey komutları haricinde komutlar kullanışlı ve yeterlidir.	Komutlar genel itibariyle kullanışlı ve yeterlidir.
Değişiklikle rin Uygulanması	Linkli çalışma ve parametrik tasarım yapılmadığı için değişikliklerin uygulanması zordur.	Linkli çalışma yapılmadığı için değişikliklerin uygulanması imkansız.	Değişiklikler yüzeyde ise uygulanması biraz zor ama diğer işlemler için hızlı bir şekilde sonuç alınabilir.	Linkli çalışma ve parametrik tasarım yapılabildiği için değişikliklerin uygulanması mümkündür.
Parametrik Çalışma	Parametrik çalışma bulunmuyor. (Ya da tasarım içinde kullanılamadı.)	Parametrik tasarım konusunda üstün özelliklere sahip	Parametrik çalışma bulunmuyor.	Parametrik çalışma yapılabiliyor.
Linkli Çalışma	Linkli çalışma yapılamadı.	Linkli çalışma özelliği bulunmamaktadır.	Linkli çalışma özelliği bulunmuyor.	Linkli çalışma özelliği bulunmaktadır.
Import/Export (İçe-Dışa Dosya Aktarımı)	Solidworks programında parça ya da montaj dosyası içerisine eğri ve yüzey dosyalarını Solidworks dosyası halinde almak mümkündür. Parça içerisine alınan eğri ve yüzeyler alınmış adıyla ürün ağacında görünmektedir. Kalıp revizyonlarının yapılması mümkündür.	İçeri aktarılan eğri ve yüzeyler için Çizen yeni bir parça dosyası açmakta ve stepten alınan bilgileri bu dosyada kullanıcıya sunmaktadır. Daha önce açılan bir parça dosyası içerisine eğri ve yüzeyleri içeri aktarmak mümkün değildir bu yüzden öncelikle eğri ve yüzeyler içeri aktarılmalı, sonra geometrik çizimler yapılmalıdır. Bu yüzden kalıp revizyonlarının yapılması mümkün değildir.	Visicad programının içerisine herhangi bir dosyayı almak mümkündür. Alınan dosya layer isminde görünmektedir. Kalıp revizyonları yapılması mümkündür.	Catia programının içerisine desteklenen belirli formatlarda herhangi bir dosyayı almak mümkündür. Bu sebeple revizyonlar uygulanabilmektedir.