

DOKUZ EYLÜL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCE

DETERMINATION OF THE WELD DIMENSIONS
IN PRESS MACHINE MANUFACTURING BY
THE FINITE ELEMENT METHOD

by

Kemal Koray ÖZTAYDAŞ

June, 2010

İZMİR

**DETERMINATION OF THE WELD DIMENSIONS
IN PRESS MACHINE MANUFACTURING BY
THE FINITE ELEMENT METHOD**

**A Thesis Submitted to the
Graduate School of Natural and Applied Sciences of Dokuz Eylül University
In Partial Fulfillment of the Requirements for the Degree of Master of Science
in Mechanical Engineering, Machine Theory and Dynamics Program**

by

Kemal Koray ÖZTAYDAŞ

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M.Sc THESIS EXAMINATION RESULT FORM

We have read the thesis entitled “**DETERMINATION OF THE WELD DIMENSIONS IN PRESS MACHINE MANUFACTURING BY THE FINITE ELEMENT METHOD**” completed by **KEMAL KORAY ÖZTAYDAŞ** under supervision of **ASSOC. PROF. DR. ZEKİ KIRAL** and we certify that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

.....
Assoc. Prof. Dr. Zeki KIRAL

Supervisor

.....

(Jury Member)

.....

(Jury Member)

Prof. Dr. Mustafa SABUNCU

Director

Graduate School of Natural and Applied Sciences

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Kemal Koray ÖZTAYDAŞ

DETERMINATION OF THE WELD DIMENSIONS IN PRESS MACHINE MANUFACTURING BY THE FINITE ELEMENT METHOD

ABSTRACT

Standard (C and H type) presses are produced in variant sizes for different capacities. Steel sheets are formed in desired shape by laser or plasma cutting processes. Some roughnesses may appear on edges during the operation. In order to analysis the situation, come across in real production conditions, by the numeric methods acceptable spaces are left between parts while body is being created by assembling in SolidWorks assembly environment. Thereby the behaviour of weld seam is examined and exerting the loads on weld seams is provided.

In this study, for generating the solid parts which compose the C type press body and body assemble automatically and parametrically by SolidWorks, a software has been developed by using VisualBasic and SolidWorks API (Application Programming Interface). Weld seams between steel sheets which compose press body are parametrically designed and located on related regions on body with developed software. By the program is run, the FEM (Finite Element Model) of press body solid model which is automatically composed in desired size by user is acquired by CosmosWorks interface and numerical analyses are performed for static condition. Natural frequencies of the press body are determined both numerically and experimentally. The displacements which consist on body are measured by laser sensors, the strains occur on critical regions, where strain values are high, are measured by strain gauges and the results are compared with the results obtained by CosmosWorks.

Keywords: Parametric design, C type eccentric presses, weld seam, finite element method, laser sensors, strain gauges.

PRES MAKİNASI İMALATINDA KAYNAK DİKİŞİ BOYUTLARININ SONLU ELEMANLAR YÖNTEMİ İLE BELİRLENMESİ

ÖZ

Standart tipteki (C ve H Tipi) presler farklı kapasiteler için farklı boyutlarda üretilmektedirler. Çelik plakalar lazer veya plazma kesim işlemleri ile istenilen boyutlara getirilmektedir. Kesim işlemleri sırasında parça kenarlarında düzgünsüzlükler oluşabilmektedir. Gerçek imalat koşullarında karşılaşılabilen bu durumun sayısal olarak incelenebilmesi için SolidWorks montaj ortamında parçalar birleştirilerek gövde oluşturulurken parçalar arasında kabul edilebilir boşluklar bırakılmıştır. Böylece yüklerin kaynak dikişleri üzerine binmesi sağlanmış ve dikişlerin yüklemeler altındaki davranışı incelenmiştir.

Bu çalışmada, C tipi pres gövdesini oluşturan parçaların katı modellerinin ve gövde montajının SolidWorks katı modelleme programı ile parametrik ve otomatik olarak oluşturulması için VisualBasic programlama dili ve SolidWorks API (Application Programming Interface) uygulaması kullanılarak bir yazılım geliştirilmiştir. Geliştirilen yazılım ile pres gövdesini oluşturan çelik sac plakalar arasındaki kaynak dikişleri parametrik olarak modellenir ve gövde montajında ilgili bölgelere atanır. Programın çalıştırılması ile kullanıcı tarafından istenilen boyutlarda otomatik olarak oluşturulan pres gövdesi katı modeline ait sonlu elemanlar modeli CosmosWorks programı ile elde edilir ve sayısal analizler statik yükleme şartı için gerçekleştirilir. Pres gövdesine ait doğal frekanslar sayısal ve deneysel olarak hesaplanmıştır. Lazer sensörler yardımı ile pres gövdesinde meydana gelen yer değiştirmeler ölçülür, strain gauge (strain ölçer) yardımı ile de gövde üzerinde gerilme değerlerinin yüksek olduğu kritik bölgelerde strain ölçümleri yapılır ve CosmosWorks programı ile elde edilmiş sayısal analizler ile karşılaştırılır.

Anahtar Kelimeler: Parametrik tasarım, C tipi eksantrik presler, kaynak dikişi, sonlu elemanlar yöntemi, lazer sensörler, strain ölçerler.

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CHAPTER ONE

INTRODUCTION

From past to the present many different technologies have been used in production industry. In parallel with growth industry requirements and the priorities have been changed. The solutions developed for both specific and general purposes. Press machines are one of the fundamental industrial production machines and take place in general side for instance. Press machines are used for many applications in industry, classified by considering the capacity, dimension and the application field. Generally they are categorized as hydraulic, eccentric and special presses. In this study eccentric presses are focused on. Eccentric presses are produced in two type called C Type and H Type and are being used in many cases like cold sheet metal processing, cutting, broaching, pounding and forming. Due to the great amounts of load which applied by machine, great tensions occurs on body assembly parts especially during contact regions and lines where merged by weld. For production of body of press machines there exist two common ways; casting and welding. Generally welding is preferred way to create the body because of its low cost. Body design, harmony between parts which composing body and welding are considerable fundamental points to prevent undesirable effects of the tensions (tearing, plastic strain etc.). Optimization on shape of parts and variability of dimensions of weld are some of workspaces deal with the defects mentioned above. Especially, since the most defects exist on welding seams, determination of weld dimensions has a high significance in preventing and handling the defects. There are certain studies in the literature related to the parametric design and determination of some geometric parameters via these techniques.

There are very limited number of studies related to the parametric design of engineering structures in the literature. This study aims to contribute to the parametric design studies.

Zhao, Huang, Khoo & Cheng (2009) studied on slotted rectangular and square hollow structural section (HSS) tension connections without welding at the end of the gusset plate for different weld length ratio, slot orientation, weld size and level of

HSS corner strength compared to its flat segment. Finite element models for the parametric study were developed and validated by Zhao & et al. (2009) against test results of the connection with the tube slotted. The modified weld length ratio was found to be a better parameter than the modified eccentricity ratio in characterizing the net section efficiency of a slotted HSS tension member when the weld length is short.

Romeijn, Sarkhosh & Hoop (2009) presented a basic parametric study on steel girders with trapezoidally corrugated webs having cut outs. A finite element analysis is carried out to investigate the effect of cut outs in corrugated webs. The analytical study showed that the influence of geometry of corrugated sheets with cut outs on the load capacity and buckling behaviour of the girder can be significant. With the help of the finite element model, the eigenvalue buckling analysis is carried out for all parameter combinations.

Athanasopoulos, Ugail & Castro (2009) presented a surface generation tool designed for the construction of aircraft geometry. Each surface is generated by a number of curves representing the character lines of a given part of the aircraft shape that can be manipulated in real time. Different surfaces then blend to create the full shape of the airplane. An important function of the proposed tool is its ability to change the aircraft shape through the adjustments of parameters associated with the initial curves. The work presents detailed descriptions on the PDE method, parametric design and manipulation of aircrafts along with graphical demonstrations of its abilities and a series of examples to illustrate the capacity of the methodology implemented.

Deng, Liang & Murakawa (2007) performed some experiments to investigate the characteristics of welding deformation in the fillet-welded joint. In order to precisely predict welding deformation by numerical method, a 3-D thermal elastic plastic finite element computational procedure is developed. The simulated results are in a good agreement with the experimental measurements. The influence on welding deformation of the flange thickness is investigated by experiment and numerical simulation.

Chatzakos & Papadopoulos (2009) attempted to set the basis for a systematic approach in designing quadruped robots employing a dynamically stable quadruped running in the sagittal plane with a bounding gait, which is a simple model commonly used to analyze the basic qualitative properties of quadruped gaits that use the legs in pair. The study takes into consideration data from experimental biology and ground surface properties, while it is subject to the existing technological limitations and economic restraints, i.e., the fact that there is a limited number of motor/gearbox combinations available from a practical point of view. The findings from simulation results indicate that the proposed methodology can assist in the design of new and modifications of existing quadruped robots.

Koini, Sarakinos & Nikolos (2009) presented a software tool for the conceptual design of turbomachinery bladings named “T4T” (Tools for Turbomachinery). It provided the ability to interactively construct parametric 3D blade rows of various types, including for multistage machines. The design procedure is parametric and a variety of different rotating machinery components may be produced. The design parameters used for the blades as well as the hub and shroud surfaces construction correspond to 2D sections.

Low (2009) considered and discussed the biomimetic design and the workspace study of undulating fin propulsion mechanisms. For a parametric study, the geometry of a single fin segment of the assembled fin mechanisms and the fin wave generated are first developed. Next, the fin workspace of the single fin segment is derived based on a defined area ratio. By virtue of the obtained fin dimensions, a gymnotiform robot, Nanyang knifefish (NKF-II), has been designed and constructed.

To improve the understanding of Steel catenary risers (SCR) behaviour and increase the confidence in the design of such systems in deepwater harsh environments, a parametric study on a SCR connected to a semi-submersible was carried out by Xia, Das & Karunakaran (2008) in this paper to deal with the factors that mainly influence the loading condition and fatigue life of the riser. Weight-optimized configurations were applied during the course of riser design. The parameters affecting the efficiency and accuracy of the simulations have also been studied during the analysis process.

Based on the concepts of linear elastic fracture mechanics, the effects of weld geometry, load conditions and the boundary constraints on fatigue strength of a ferrite-pearlite steel lap joint were investigated by Li, Partanen, Nykanen & Bjork (2001) using the finite element method. Various weld geometry including the leg length, flank angle and the size of lack-of-penetration were considered during the calculation of fatigue strengths. For a lap joint, with a transverse fixed boundary constraint at the main plate, the fatigue strength increases with a decrease of weld size but the influence of flank angle depends on type of load carried. Li et al. (2001) also found that the size reduction in Finite Element model is significant influence on the calculated fatigue strength; the use of reduced size FE model gives much higher overestimate of fatigue strength of the joint.

The aim of this study is determination of the weld dimensions, trials on design of body and parts in press machines via software which has been written for the purpose of parametric design in the scope of this study. Algorithms have been written in VisualBasic 6.0 programming language with SolidWorks2007 Macro codes. SolidWorks has macro recording capability called API (Application Programming Interface). After preparation of the software, numerical analyses and experiments are executed in accordance with requests of the study. The effects of changing dimensions of weld regions and body parts have been observed and compared with the test results. This thesis is organized as follows In Section 2, parametric design and next steps followed through are described. In Section 3, numerical and experimental analyses on test sample are given. Numerical analyses and experimental studies for press body are given in Section 4. Finally, the conclusions are drawn in Section 5.

CHAPTER TWO

PARAMETRIC DESIGN

2.1 Introduction

In the scope of thesis, effects of weld seam dimensions on stresses, strains and displacements occurring on the press body have been studied. Furthermore, a computer code and an interface as seen in Figure 2.1 have been developed to design parts and weld seams parametrically which compose the press body by using API facility (Application Programming Interface) of 3D design software SolidWorks®.

Nowadays one of the most important criteria expected from the employee in R&D (Research and Development) department is compete with time. It is expected to be brought out the designs and projects that meet the expectations in limited period. Softwares are prerequisite for optimizing the existing model in a little while. Thus, all sections which are examined in the name of the effects on design are comprehended as parameter by software; the expected alteration is performed in software interface, regeneration of design is automatically enabled and the effect of alteration on design can be observed.

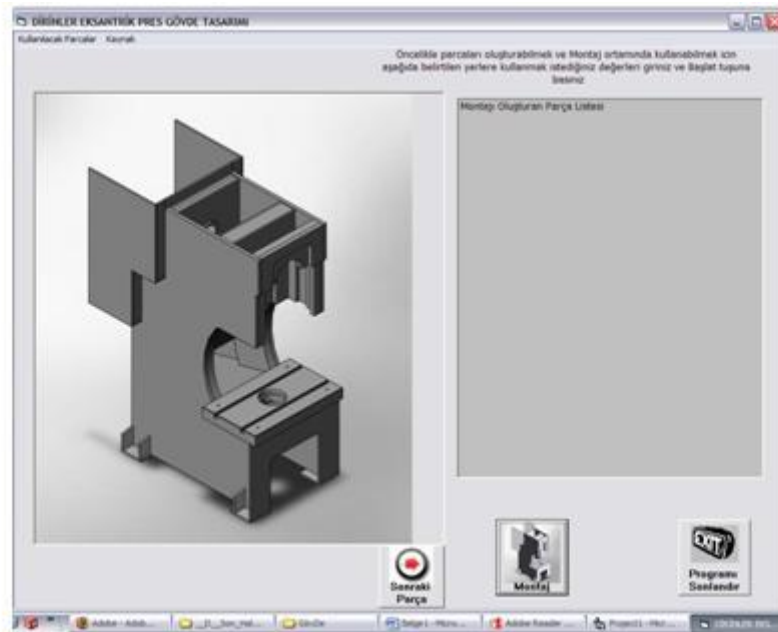


Figure 2.1 Parametric design interface

In case of performing the processes mentioned before manually, causes appreciable time loss. In order to compete with competitors, so as to supply the requirements in optimum way after testing by required tests and to rebuilt the existing design in customer-driven way, parametric design softwares are being needed.

2.2 Parametric Design

2.2.1 Design of Parts Composing the Body

Within the scope of this study, press machine with 80 tones capacity has been considered. Press body composes of different 65 parts which are assembled by welding. The fundamental aim of this study is examining the effects of dimensions of weld seams on press body under loading condition. Modifications on parts affect the stress, strain on weld seam and displacement on body as well. Considering the situation, initially worked on the parts that compose the body. Different program codes have been written in software interface which is developed to design the parts parametrically. "Govde Yan Sacı" is one of the most critical part among the parts creating press body. The interface and the program, which are developed for designing the "Govde Yan Sacı", are given as a sample.

2.2.1.1 Govde Yan Sacı

"Govde Yan Sacı" (Shown in Figure 2.2) has 30 mm thickness as default value sheet metal which surrounding the press body from right to left and carrying many parts on itself. "Govde Yan Sacı" is backbone for press body. Hence, alterations on it affects many parts directly.

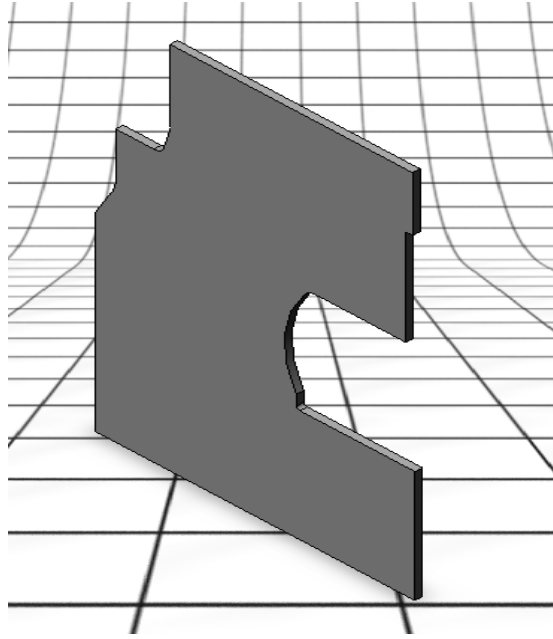


Figure 2.2 "Gövde Yan Sacı"

The interface and the codes used for building the part up are explained in detail below. In this study, the property of SolidWorks which saves all activities as Macro is used as seen in Figure 2.3.

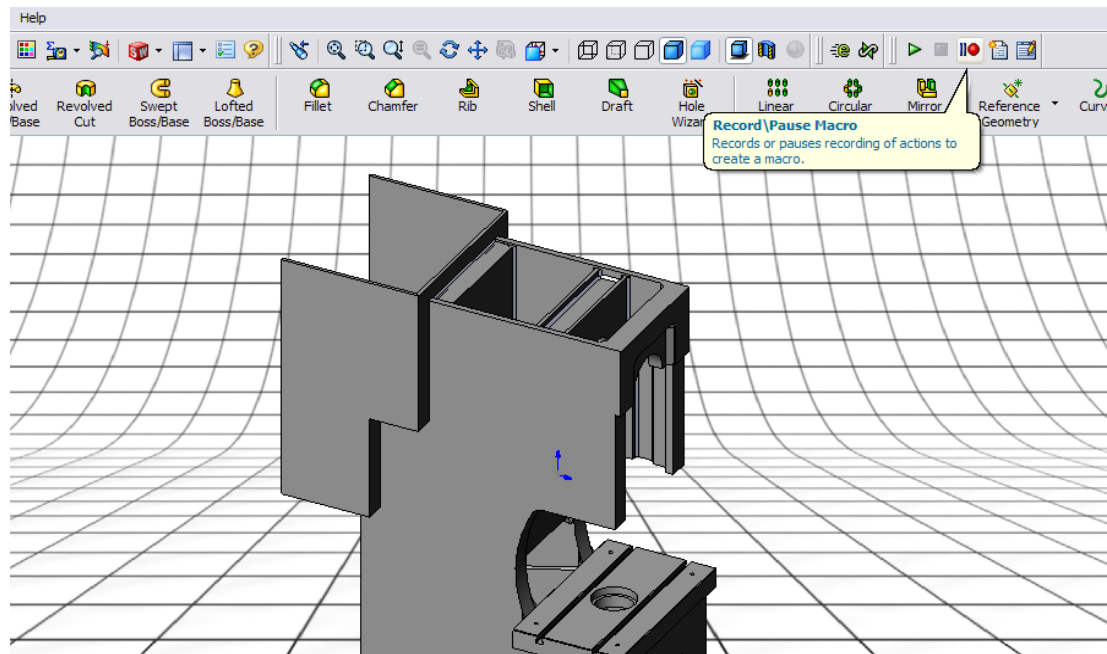


Figure 2.3 SolidWorks macro toolbar

Primarily the part is modeled in manual way. Meanwhile studies had been done before were recorded with Macro/Record option. The codes had been recorded by macro are compatible with VisualBasic language. These codes are used for parametric design in the developed interface. Sample codes are given in Figure 2.4.

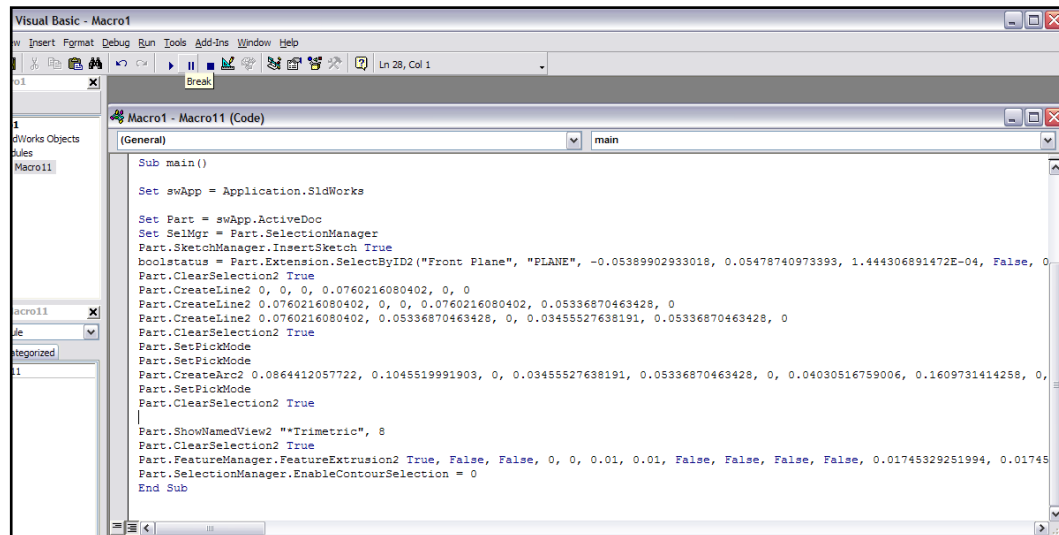


Figure 2.4 SolidWorks macro sample

The codes belong to "Govde Yan Sacı" are checked out with details, place of use and intended use of the codes are explained with the sample shown in Figure 2.5.

```

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

Dim msg As String

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, 0, 0, gys_1, 0, 0
part.ViewZoomtofit2
part.CreateLine2 gys_1, 0, 0, gys_1, gys_2, 0
part.CreateLine2 gys_1, gys_2, 0, (gys_1 - gys_3), gys_2, 0
part.CreateArc2 (gys_1 - gys_3), (gys_2 + r1), 0, (gys_1 - gys_3), gys_2, 0, (gys_3 + r1), (gys_2 + r1), 0, -1
part.ViewZoomtofit2
part.CreateLine2 (gys_1 - gys_3 - r1), (gys_2 + r1), 0, (gys_1 - gys_3 - r1), gys_4, 0

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

```

Figure 2.5 Codes of "Gövde Yan Sacı"

The sample codes and their explanations are given in Figure 2.6.

```
Call VeriOkuma 'Yandaki kod yardımı ile başka bir Subroutine çağırılarak programın kullandığı parametreler okunur Bkz.Şek.3.8
Set swApp = GetObject("sldworks.application") ' Yandaki kod takımı ile SolidWorks Programında
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#) 'Part dosyası açılır ve yeni bir Part Dosyası oluşturmak
Set part = swApp.ActiveDoc ' mümkün olur
Call snap ' Snap subroutine'i ile Programdaki otomatik ilişkilemler kapatılır.
```

Figure 2.6 Codes of "Gövde Yan Sacı"

After the codes mentioned above in Figure 2.6 are read sketch commands appear. With sketch codes all activities in sketch environment in SolidWorks could be done. For example; opening a sketch, plane selection, drawing line in selected plane, drawing spline and drawing circle. With the help of these codes shown in Figure 2.7 the sketch of "Govde Yan Sacı" is drawn and designed in two-dimensional environment.

```
part.SketchManager.InsertSketch True ' Sketch ortamında yeni bir sketch açılmasını sağlayan komuttur.
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0) ' Sketch ortamında bir düzlem-
(plane) seçilmesini sağlayan komuttur.
part.CreateLine2 0, 0, 0, gys_1, 0, 0 ' Çizgi çizilmesini sağlayan komuttur.
part.ViewZoomtofit2 ' Görüntüyü ekrana fit hale getirir.
part.CreateLine2 gys_1, 0, 0, gys_1, gys_2, 0
part.CreateLine2 gys_1, gys_2, 0, (gys_1 - gys_3), gys_2, 0
part.CreateArc2 (gys_1 - gys_3), (gys_2 + r1), 0, (gys_1 - gys_3), gys_2, 0, (gys_3 + r1), (gys_2 + r1), 0, -1
```

Figure 2.7 Codes of "Gövde Yan Sacı"

The coordinate of one end of straight line drawn in sketch environment is x_1, y_1, z_1 , and if the coordinate of the other end of straight line is x_2, y_2, z_2 , the required codes for drawing the line will be as follows in command.

Part.CreateLine ($x_1, y_1, z_1, x_2, y_2, z_2$)

"gys_1", which is following the command CreateLine, denotes a parameter. Parameters used in the codes are stored in Text files.

When the software needs to use any parameter, reaches the required text file and reads the required parametric values. For example assumed default value of parameter "gys_1" is 1300 mm or assumed default value of parameter "gys_3" is 467 mm. Since the codes in Figure 2.7 checked up any numerical value has not been used directly. Required values are obtained by algebraic operations in accordance with parameter's model requirements. That logic circuit is valid for all equations and algorithms used in prepared software. Thus, when any parameter of program is changed, software detects that alteration and rebuilt the design to create according to changing. In Figure 2.8 with a sample subroutine it is shown how to read parametric values.

```

Sub gys_sol_parametre_okutma()
Open f10 + "gys_sol_parametrik_degerler.txt" For Input As 1
Input #1, msg
Line Input #1, inp1: gys_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: gys_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: gys_3 = Val(Left(inp3, 8)) / 1000
.
.
Line Input #1, inp18: r1 = Val(Left(inp18, 8)) / 1000
Line Input #1, inp19: r2 = Val(Left(inp19, 8)) / 1000
.
Line Input #1, inp24: parca_kalinaligi = Val(Left(inp24, 8)) / 1000

Close #1
|
End Sub

```

Figure 2.8 A sample data read code under "Veri Okuma" subroutine

The parameters saved in "gys_sol_parametrik_degerler.txt" file are able to be changed by user by reaching the text file in Figure 2.9.

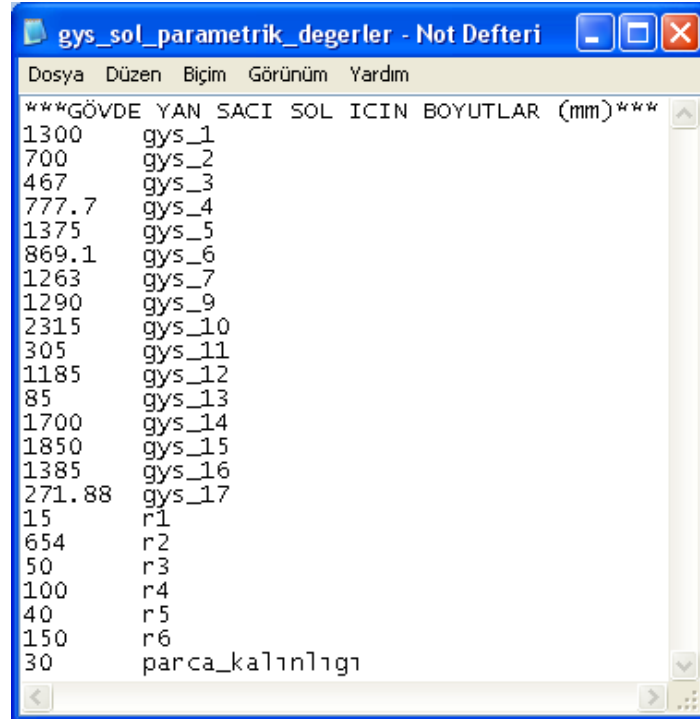


Figure 2.9 Sample text file where the parameters are stored

The radius (R2) between the table-ram of "Govde Yan Sacı" is drawn by using Newton-Raphson method as shown in Figure 2.10. Iteration provides the alterations of parameters to affect the drawing radius on sheet metal.

```

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1 'Sol tarafta yer alan deęişkenler newtonR adlı bir sub rutin de kullanılmaktadır.
yp2 = gys_4
R = r2

Call newtonR ' Newton Raphson methodu kullanılarak Gövde Yan Sacı'nın Masa-Koç mesafesi arasındaki Radius
' sadece parametreler deęiştirilerek yazılım tarafından çizilebilmektedir.

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)

part.CreateArc2 gys_merk_x, gys_merk_y, 0, xp2, yp2, 0, xp1, yp1, 0, -1 ' Üç Nokta Kullanılarak Yay çizme komududur.

```

Figure 2.10 Codes of "Gövde Yan Sacı "

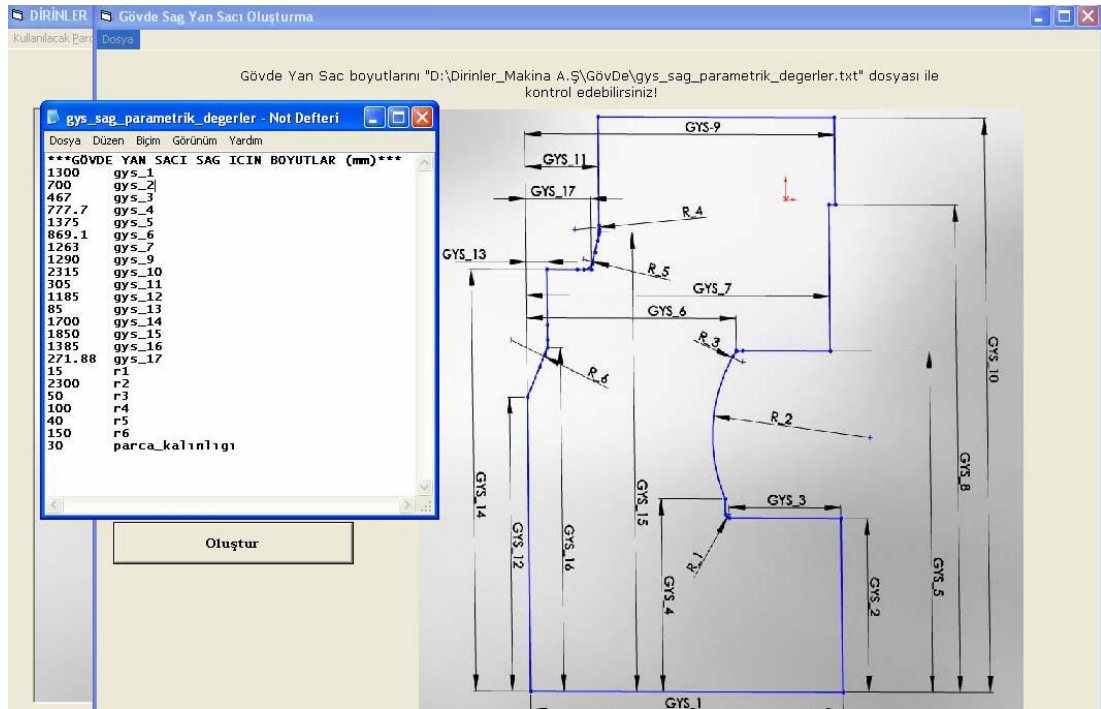


Figure 2.11 Design interface of "Gövde Yan Sacı"

"part.CreateArc2" is used for drawing a circle with three known coordinate points. Coordinate of arc center is gys_merk_x , gys_merk_y and gys_merk_z ($gys_merk_z=0$). The coordinate of one end of arc is $xp2$, $yp2$, 0 and the other end of arc belongs to coordinate $xp1$, $yp1$, 0. "-1" value used in the end of code is used to show drawing direction; "-1" is for counter clockwise drawing and "1" is for clockwise drawing. It is shown below in sample commands.

Part.CreateArc2. gys_merk_x , gys_merk_y , 0, $xp2$, $yp2$, 0, $xp1$, $yp1$, 0, -1

Part.CreateArc2. Mx , My , 0, Bx , By , 0, Ax , Ay , 0, -1

The radius R2 is created with Three Points Arc method in SolidWorks program while "Govde Yan Sacı" is modeling. The coordinates belong to A and B points, which are end points of arc and shown in Figure 2.12, are able to be calculated in accordance with parts dimensions. The coordinate of arc center "M" is obtained by solving non-linear algebraic equations. In solution of non-linear algebraic equations Newton-Raphson method has been used.

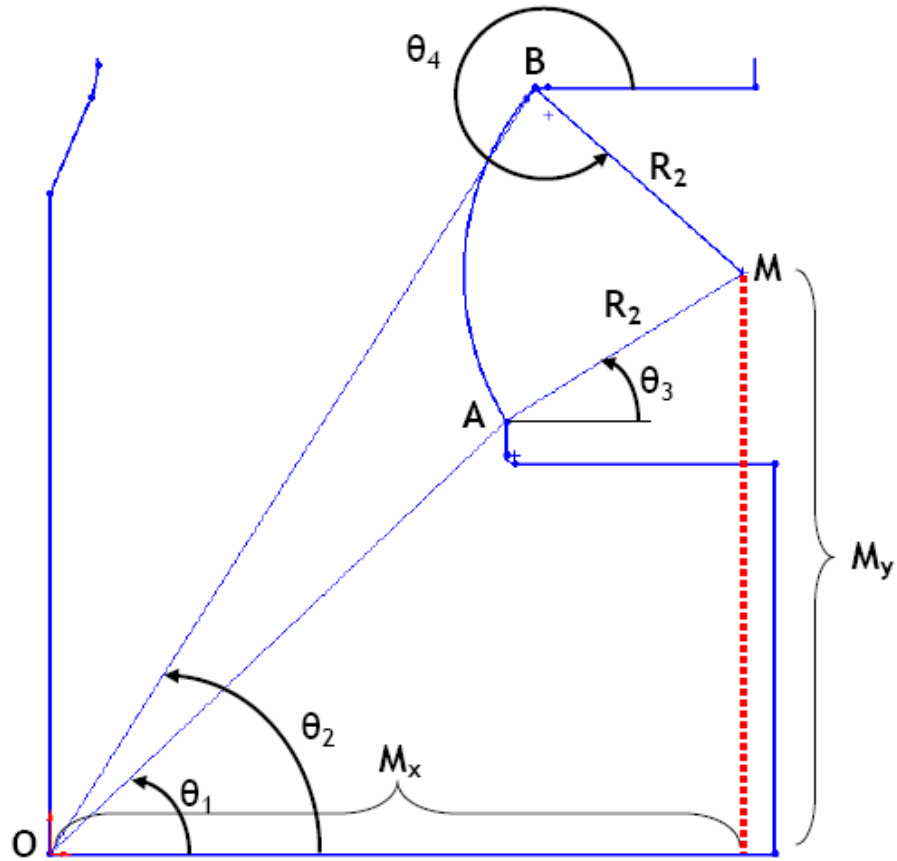


Figure 2.12 Geometry of R2 of "Gövde Yan Sacı"

The Equation 1 presenting below may be written in relation to geometry given in Figure 2.12:

$$\overline{OA}e^{i\theta_1} + \overline{MA}e^{i\theta_3} = \overline{OB}e^{i\theta_2} + \overline{MB}e^{i\theta_4} \quad (1)$$

Equations 2 and 3 may be written below as component of Equation 1:

$$\overline{OA} \sin \theta_1 + \overline{MA} \sin \theta_3 - \overline{OB} \sin \theta_2 + \overline{MB} \sin \theta_4 = 0 \quad (2)$$

$$\overline{OA} \cos \theta_1 + \overline{MA} \cos \theta_3 - \overline{OB} \cos \theta_2 + \overline{MB} \cos \theta_4 = 0 \quad (3)$$

θ_3 and θ_4 are variables and may be determined with Newton-Raphson method. With the solution of equations, coordinates x and y of point "M" according to point "O" may be determined as follows as given in Equation 4 and 5.

$$M_x = \overline{OA} \cos \theta_1 + \overline{MA} \cos \theta_3 \quad (4)$$

$$M_y = \overline{OA} \sin \theta_1 + \overline{MA} \sin \theta_3 \quad (5)$$

After determining coordinate of radius center, the radius R2 is built up with command “Part.CreateArc2”. Newton-Raphson method is also used for solid modeling of some parts belong to press body.

The other commands used for creating "Govde Yan Sacı" are “SketchFillet”, “FeatureExtrusion” and “CreatePlaneAtOffset”. With the command “SketchFillet” it is possible to give radius on vertex where two line intersect. As codes are given in Figure 2.13. Vertex number 20 on sketch had been selected and the value of "r6" parameter 150 mm is assigned as radius value.

```
boolstatus = part.Extension.SelectByID2("Point20", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r6, 1 ' Bu komut ile Sketch üzerindeki istenilen köşe(nokta) seçilir ve bu köşeye radyus verilir.
part.ClearSelection2 True
```

Figure 2.13 Sample SketchFillet command

It is possible to transform a two-dimensional sketch drawing to a three-dimensional solid model with command “FeatureExtrusion” as shown in Figure 2.14. Primarily active sketch named "Sketch1" is selected and have it volumed by “Extrusion” command.

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, parca_kalınlığı, 0, False, False, False, False, 0, 0
,False, False, False, False, 1, 1, 1, 0, 0, False
' Bu komut ile "parca_kalınlığı" parametresinin değeri kadar 30 mm parçaya
' kalınlık verilir ,böylece iki boyutlu bir çizim üç boyutlu katı bir modele dönüşmüş olur.
```

Figure 2.14 “Extrude” command as code

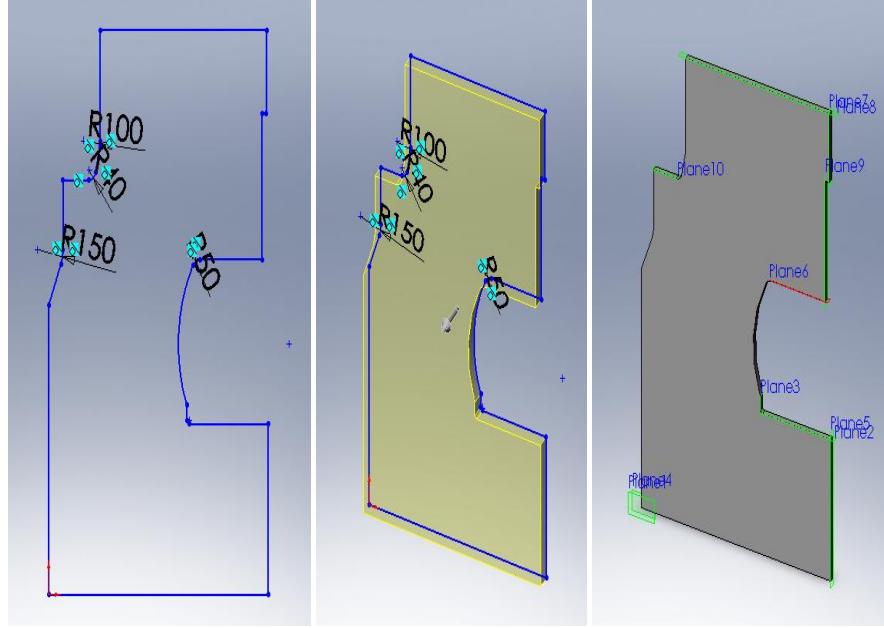


Figure 2.15 “Extrude” process

With the command "CreatePlaneAtOffset" it is possible to create planes which are necessary in Mate process for intersecting the faces together as shown in Figure 2.15 and Figure 2.16.

```
boolstatus = part.Extension.SelectByID2("", "FACE", gys_2, gys_2, parca_kalnlığı, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True ' Bu komut "Plane" atanacak yüzey seçilir ve bu yüzeye "Plane" atanır.
' Secilen Yuzey için "gys_2" - X, "gys_2" - Y ve "parca_kalnlığı" - Z eksenindeki koordinatlardır. Bu yüzeyden 0 mm uzakta
"Plane" oluşturulmuştur.
```

Figure 2.16 “CreatePlane” command as code

In software prepared in the scope of thesis, any part that belong to press body can be created by using the codes mentioned above. However, user will execute the modeling by using the developed software. Creation of "Govde Yan Sacı" with developed software will be possible by following the way explained below.

At first, SolidWorks is activated. Then from the developed interface, the part desired to be created may be reached by using the toolbar or using the buttons named “Sonraki Parça” and “Önceki Parça” as shown in Figure 2.17.

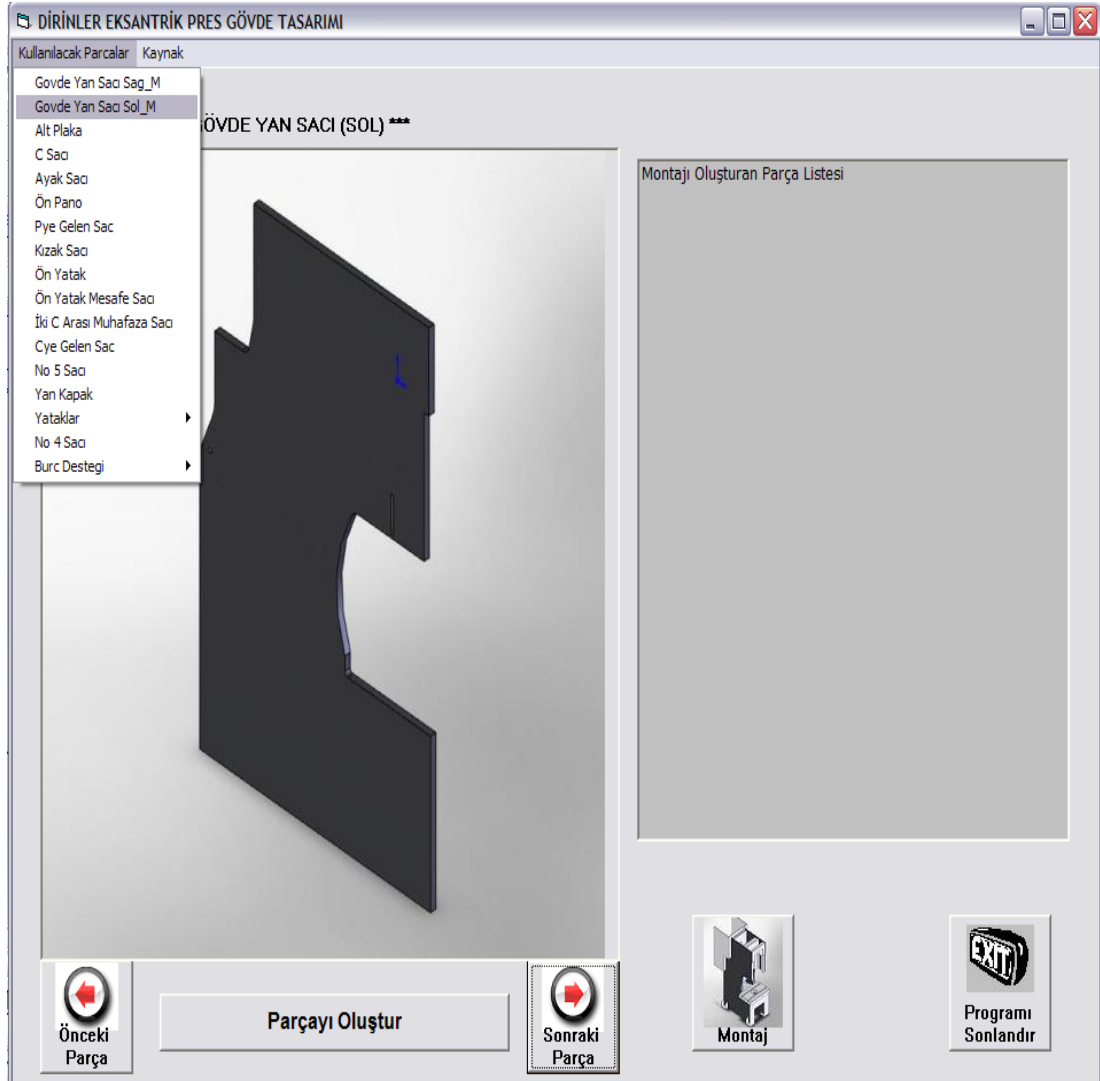


Figure 2.17 Program Interface

Following the selection of part which is going to be designed on interface, clicking the button named “**Parçayı Oluştur**” helps to activate the form which belongs to part. Parameters for modeling the part may be changed from “**Dosya**” on toolbar in form belongs to part and with the button “**Oluştur**”, solid model of part is created in SolidWorks environment by using the dimensions predefined. In Figure 2.18 progressing program interface of “Govde Yan Sacı” and related parameters for part are shown.

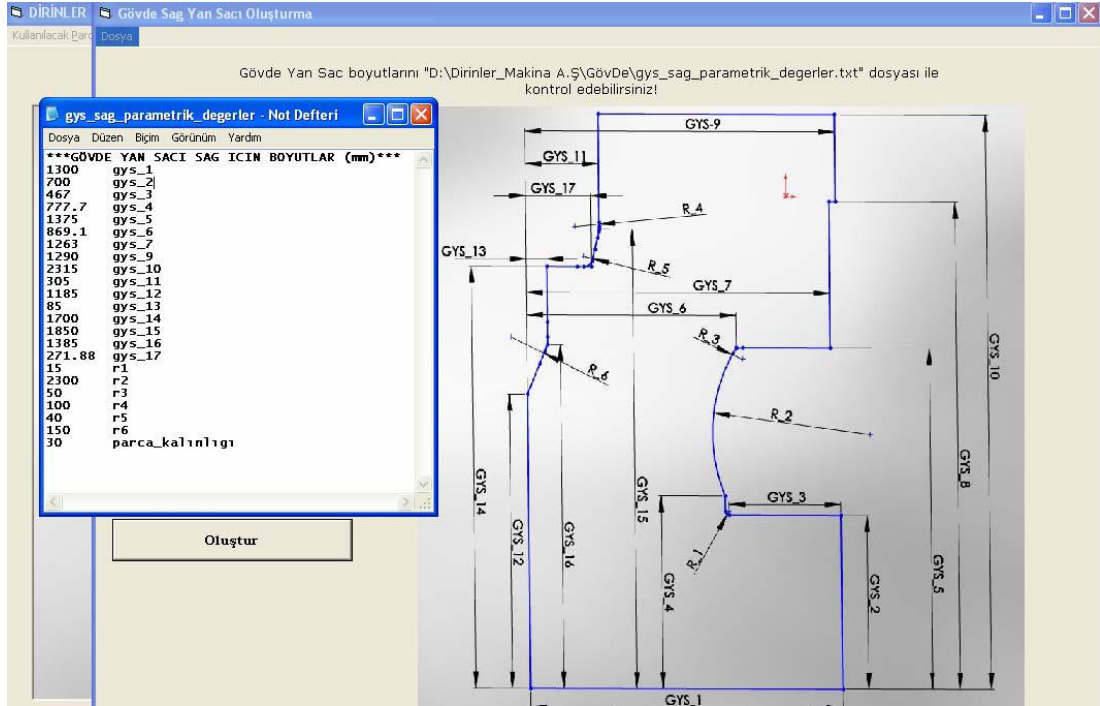


Figure 2.18 Gövde Sağ Yan Sacı creation form

In case one of the basis dimension "R2" from those parameters belongs to "Govde Yan Sacı" takes different values 2300 mm, 900 mm and 500 mm, different solid models created by SolidWorks are shown in Figure 2.19.

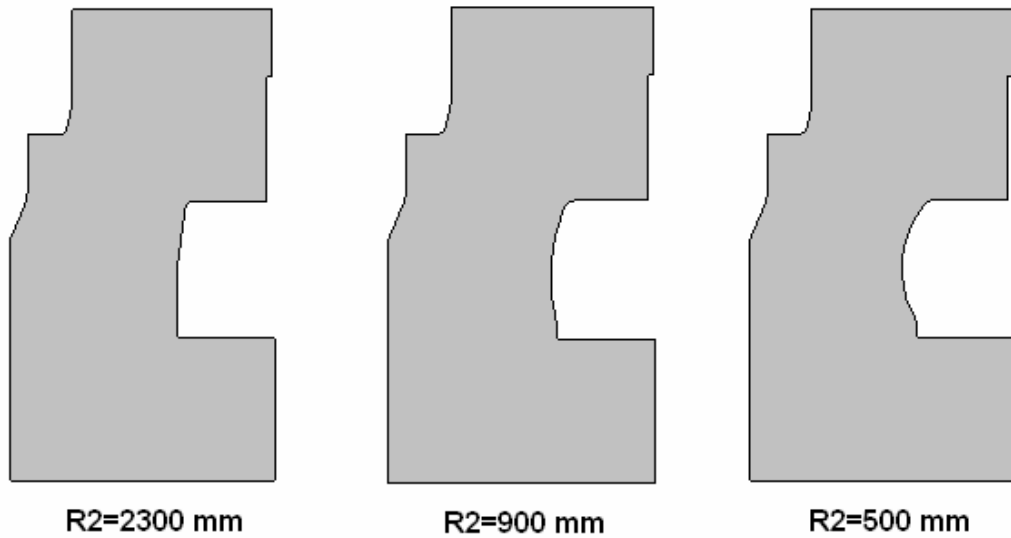


Figure 2.19 Different solid models for Gövde Sağ Yan Sacı with different "R2" values

2.2.1.2 C Sacı

C Sacı is one of the most important structural part of the eccentric press under consideration. Some of codes related with "C Sacı" are influenced by alteration of "Govde Yan Sacı" explained in section 2.2.1.1, and these have been given in Figure 2.20. In case of analysing the codes, the parameters used in software seem to be sufficient for modeling the "C Sacı". "C Sacı" with "Gövde Yan Sacı" has the same internal radius. As well as internal radius, also outer radius has been drawn by using Newton-Raphson method for solving non-linear equations.

```

Private Sub Command1_Click()

Call VeriOkuma
*****GYS Olusumundan Buraya Yansiyacak Degerler Icin*****

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
*****
*****

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

Öteleme_x = ((gys_1 - gys_3 - r1) - cs_8)
Öteleme_y = (gys_2 - cs_10)

xp1 = cs_2
yp1 = cs_1
xp2 = 0
yp2 = 0
R = cs_r1

Call newtonR

```

Figure 2.20 Codes for "C Sacı"

"C Sacı" creation form is shown in Figure 2.21. Internal radius of "C Sacı" has the same dimension with internal radius of "Govde Yan Sacı" and it's value automatically assigned by using the value of internal radius of "Govde Yan Sacı".

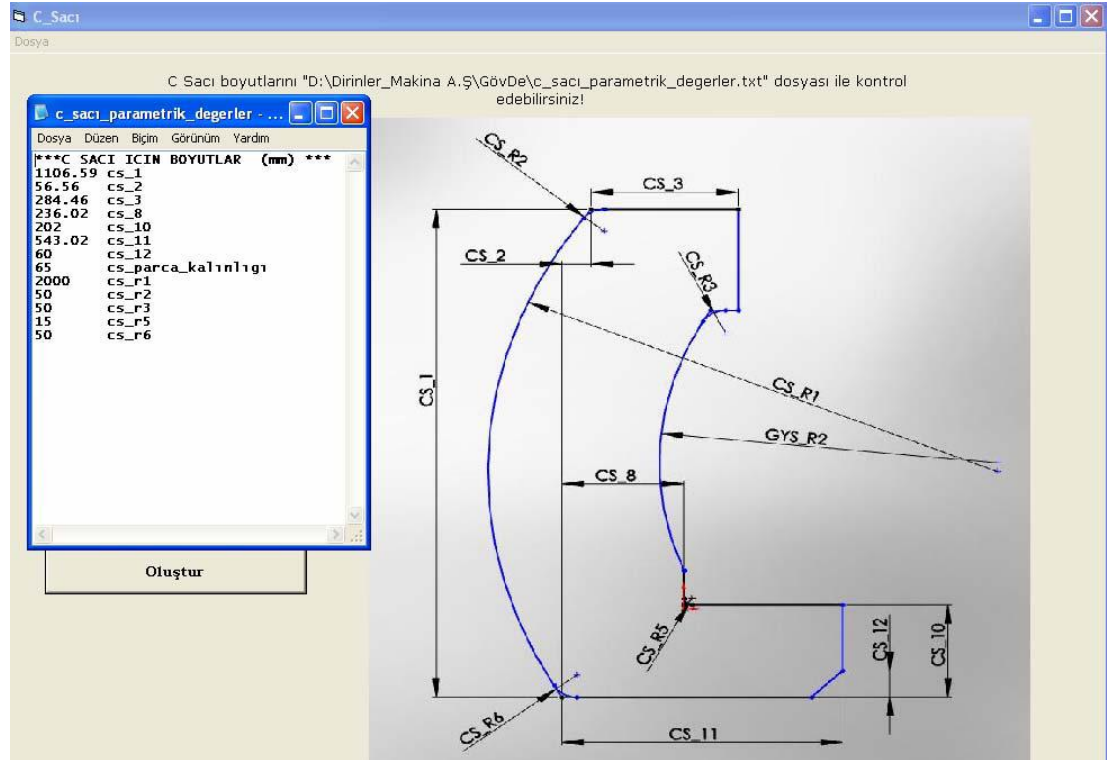


Figure 2.21 The form for creation of C Sacı

Different sizes of "C Sacı" created by developed software are shown in Figure 2.22.

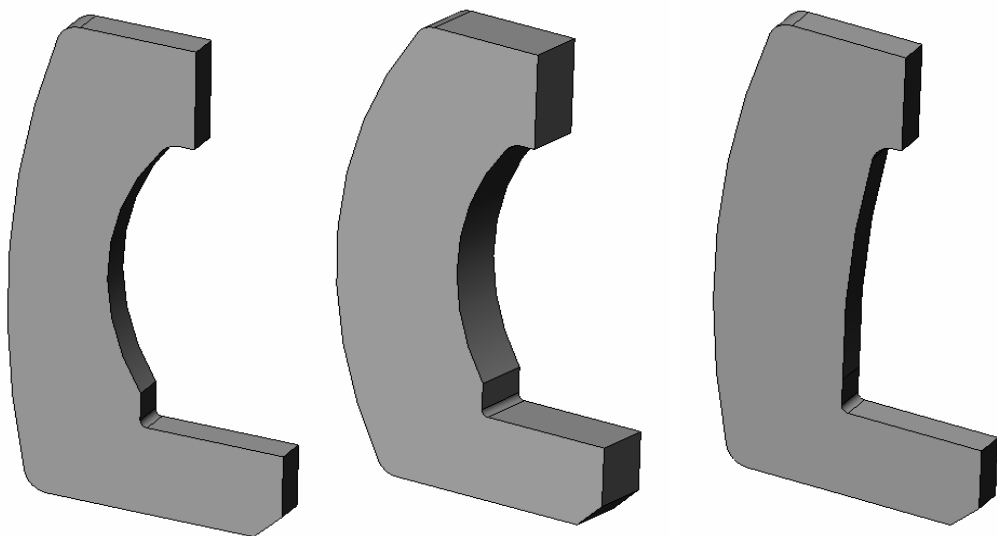


Figure 2.22 Samples for "C Sacı" modeled by developed software

The other parts composing the C type press body are also modeled with similar method in creation period of body. After modeling the all parts, by pushing the button "**Montaj**" located in main form, parts belong to press body are brought together to compose press assembly by using the geometric relations between the parts. In Figure 2.23 different sized C type press bodies modeled by developed software are shown.

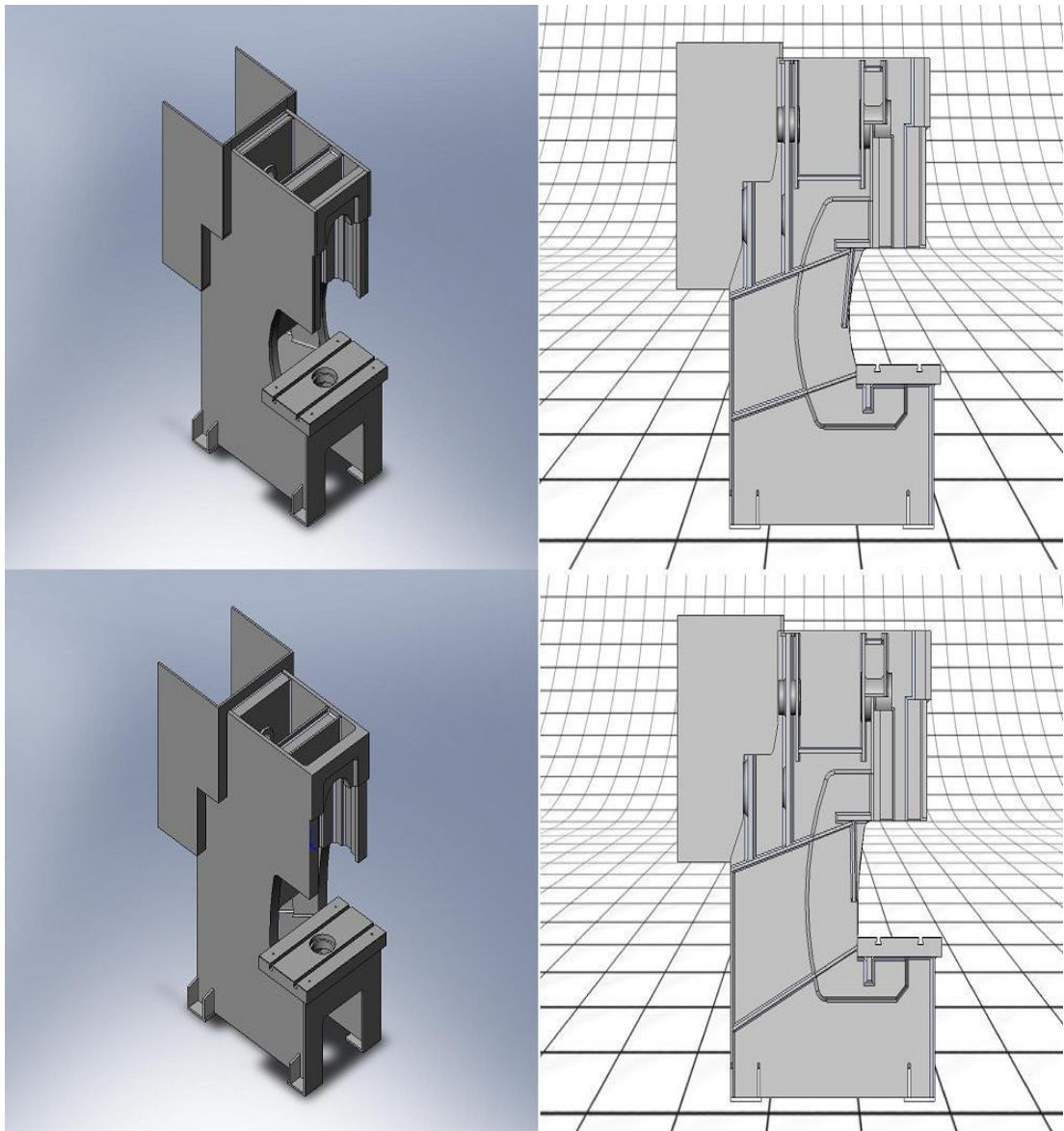


Figure 2.23 Effects of different parameter values on press body

2.2.2 Body Assembly

After creation of solid model, the obtained model is automatically saved in order to be used in press body assembly with a name indicated before. All parts composing the press body are created by this way and if required different combinations for press body may be tested by changing their parameters. Assembly process may be performed after the parts are created and saved with the names indicated before. After pushed down the button "**Montaj**" in program interface shown in Figure 2.17, the interface shown in Figure 2.24 is activated.

The screenshot shows a software interface titled "Montaj". It features a grid of input fields for various dimensions and a central technical drawing of a mechanical assembly. The drawing includes dimension lines labeled X_1 through X_7 and Y_1, corresponding to the input fields. The "Başlat" button is highlighted with a red border.

Parameter	Value (mm)
"Eksen_1" Krank Orta Yatak Eksen	1191
"Eksen_2" Ön Yatak ile Arka Yatak Arası Eksen	450
"Eksen_3" Ön Yatak ile Orta Yatak Arası Eksen	13
X_1 (mm)	253
X_2 (mm)	36
X_3 (mm)	123
X_4 (mm)	15
X_5 (mm)	34
X_6 (mm)	213
X_7 (mm)	114
Gövde Yan Sacları Arası Mesafe "Mesafe"(mm)	580
Y_1 (mm)	258

Figure 2.24 Assembly Form

2.2.2.1 Inserting the Parts in Assembly

Just after the button "**Montaj**" is pushed down, primarily a subroutine created with VisualBasic commands is read and then the names of parts will be inserted in

the assembly environment which are written in list appearing on main form shown in Figure 2.17.

After listed parts will be inserted to the assembly, another subroutine, which is written for inserting parts to assembly, is read automatically as shown in Figure 2.25

```

Sub parcaları_cagir()
'Bulunması Gereken Satırlar, SolidWorks Montaj ortamı Aktif hale getirilir.
Set swApp = GetObject( "sldworks.application")
Set asmb1 = swApp.NewDocument(flsw + "assem.asmdot", 0, 0#, 0#)
Set asmb1 = swApp.ActiveDoc: nft = 1

Call snap

For i = 1 To n
  fl = parca(i) ' Burada motaj için kullanılacak parcalar listelendikten sonra numara sırasına göre çağırılır ve 50 nolu satırdaki
  GoSub 50 ' prosedür uygulanır.
Next i

asmb1.ViewZoomtofit2: bs = asmb1.EditRebuild3
Exit Sub
' Asagıda ,parcalar tek tek acılıp montaj ortamına atıldıktan sonra tek tek tekrar kapatılıyor ...
50 Set part = swApp.OpenDoc6(fl0 + fl, nft, 0, "", n1, n1)
  asmb1.AddComponent fl0 + fl, 0, 0, 0|
  'Form1.Print fl0 + fl
  Set part = swApp.ActivateDoc2(fl, False, n1): Set part = Nothing ' Yandaki komut ile açık olan ve montaj ortamına taşınan
  swApp.CloseDoc fl: Return ' part. dökümları yani parçalar kapatılır. Sadece Assembly
End Sub ' dökümanı açık kalır.

```

Figure 2.25 Inserting the parts in assembly

In this code,

$fl0 = "D:\Dirinler_Makina A.Ş\GövDe"$, $fl = "parca(i)"$ and

$flsw = "C:\Program Files\SolidWorks\lang\english\Tutorial"$ variables are defined in main form before.

$Set part = swApp.OpenDoc6(fl0 + fl, nft, 0, "", n1, n1)$

With the above command, $fl0$ which is given in the command above represents folder name where studies are saved, fl represents the name of parts inserted and nft represents type of model wanted to be inserted assembly. nft has to be "1" for Part and has to be "2" for assembly; otherwise inserting process fails. 0 shows whether the part inserted is read only or not and view only or not.

0 is used for inserted part to be available and viewable. If following variable set as shown above (" ") inserted part comes to assembly with last changing on part. However "n1" is another variable defined as long and used for Errors and Warnings as given in Figure 2.26.

retval = SldWorks.OpenDoc6 (filename, type, options, configuration, &Errors, &Warnings)		
Input:	(BSTR) Filename	Document name or full path if not in current directory, including extension
Input:	(long) Type	Document type as defined in swDocumentTypes_e
Input:	(long) Options	Mode in which to open the document as defined in swOpenDocOptions_e
Input:	(BSTR) Configuration	Model configuration in which to open this document <ul style="list-style-type: none"> • Applies to parts and assemblies, not drawings • If this argument is empty or the specified configuration is not present in the model, the model is opened in the last-used configuration.
Output:	(long) Errors	Load errors as defined in swFileLoadError_e
Output:	(long) Warnings	Warnings or extra information generated during the open operation as defined in swFileLoadWarning_e
Return:	(LPDISPATCH) retval	Pointer to a Dispatch object, the newly loaded ModelDoc2 , or NULL if failed to open

Figure 2.26 Variables for swApp.OpenDoc6 command

```
asmb1.AddComponent fl0 + fl, 0, 0, 0
```

With this command given parts are inserted to assembly and all inserted parts are located in reference point where the coordinate X, Y and Z are "0". Parts should be separated away and located in different coordinates. Because, after inserting the parts to assembly environment, distance between themselves and positions according to themselves are used in Mate operation. However, before the procedure explained above, existing work is saved as assembly file named "Dirinler_Pres" with command shown in Figure 2.27

```

Sub kaydet()
asmb1.SaveAs2 "D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres.SLDASM", 0, False, False
End Sub

```

Figure 2.27 Saving Subroutine

After saving procedure location of parts in assembly environment is like shown in Figure 2.28. Because firstly parts are inserted to 0,0,0 reference coordinate as mentioned before.

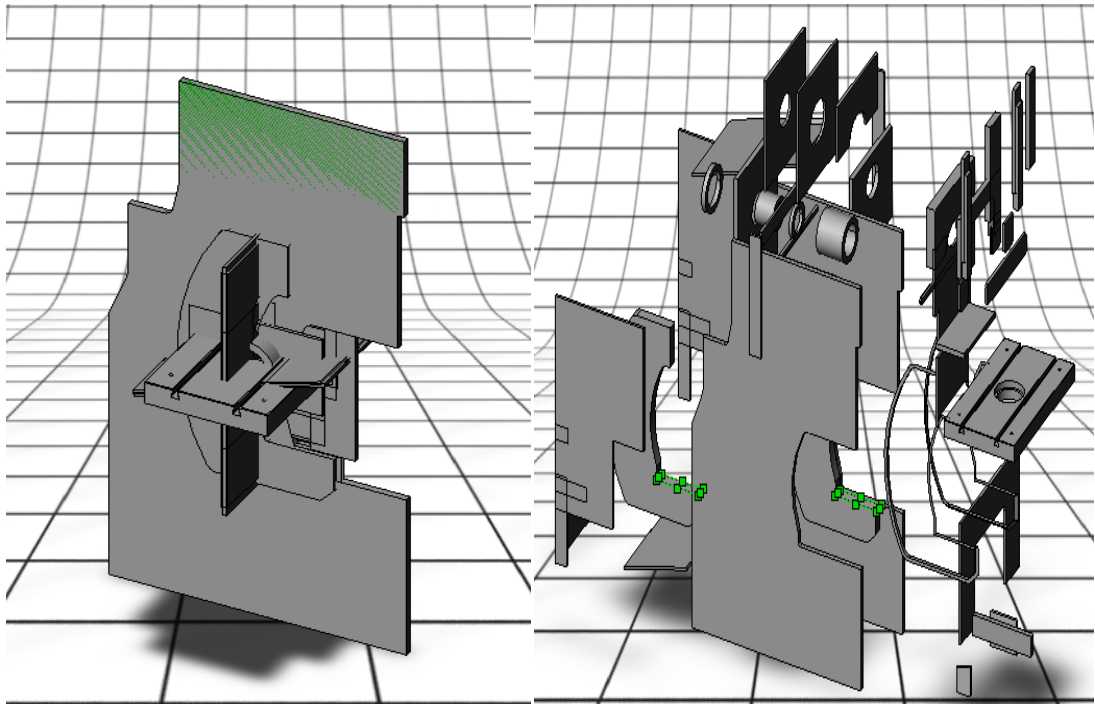


Figure 2.28 Location of parts in reference point and separation of parts

Just after pushing to button "**Başlat**" in assembly interface shown in Figure 2.24 procedure mentioned above works. Automatically working procedure continues with subroutine named "Parçaların Lokasyonu" for locating the parts. As it seems in Figure 2.28 that subroutine is used for setting the parts apart. In Figure 2.29 there are codes about listing the parts mentioned before. In assembly there are 65 parts totally and all parts have different numbers. If any part wanted to be inserted twice to assembly that part has to be numbered twice with different numbers and in Figure

2.29 there is a sample about that. Later on, these codes are used for the names take place in subroutine shown in Figure 2.30

```

Sub parcalistesi()
ana_form.List1.Clear
n = 65 'Parça adedi
ReDim parca(n) As String
parca(1) = "Govde Yan Sacı Sag_M.sldprt"
parca(2) = "Govde Yan Sacı Sol_M.sldprt"
parca(3) = "Alt Plaka.sldprt"
parca(4) = "C_Sacı.sldprt"
parca(5) = "C_Sacı.sldprt"
.
.
.
For i = 1 To n
ana_form.List1.AddItem (f10 + parca(i))
Next i
End Sub

```

Figure 2.29 Subroutine for Part List

```

Private Sub Form_Load()

Sub parcalarin_lokasyonu()
Set swApp = GetObject(, "sldworks.application")
Set asmdl = swApp.ActiveDoc

Call snap

Dim prc As String

prc = parca(1)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmc: cfix = "fix"
'Form1.Print flasmc
xt = 0 / 1000: yt = 0 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 180, tz)

prc = parca(4)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmc: cfix = "unfix"
'Form1.Print flasmc
xt = -1800 / 1000: yt = -1500 / 1000: zt = 450 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

prc = parca(5)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmc: cfix = "unfix"
'Form1.Print flasmc
xt = -1800 / 1000: yt = -1500 / 1000: zt = -900 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

Figure 2.30 Subroutine for Locating Parts

In subroutine shown in Figure 2.30 there are codes determining the parts how to be named in assembly list appearing in assembly environment. Every part initially assigned to a variable named "**prc**" with filename extension. Later on, the number of the character which are belong to part names are counted by command "**Len**". The character count of ".**sldprt**" 7 is deducted from number of total character. This value is used as number of character which has to be written from left side with command "**Left**". And, if the concerned part is wanted to be used once in assembly "1" is added at the end of name or "2" is added at the end if the part wanted to be used twice. "**flasmb**" is added as suffix. In codes of main form "**flasmb**" is defined as variable and it is the name given to assembly file.

```
f10 = "D:\Dirinler_Makina A.Ş\GövDe\"
flsw = "C:\Program Files\SolidWorks\lang\english\Tutorial\"
flasmb = "Dirinler_Pres"
```

For example if $parca(3) = Alt\ Plaka.sldprt$ in subroutine named "Parçaların Lokasyonu" it is going to be $comp = Alt\ Plaka-1$ and in SolidWorks assembly environment in assembly list is going to appear with that name.

With variable "**cfix**" coming after "**fix**" and "**unfix**" expressions, which specify first part is fix and the others are unfix, are assigned. In assembly environment, displacement of parts, from reference point to different coordinates with different angles, is enabled with algorithm composed of orientation matrix in subroutine. "**xt**", "**yt**" and "**zt**" are transformation components which are used in orientation matrix shown in Figure 2.30. However, "**tx**", "**ty**" and "**tz**" are components of rotation matrix which are used in orientation matrix. Stated in other words, while x, y and z coordinates are defined by transformation matrix shown in Equation 6, with rotation components rotation angles in x, y and z axis are defined. In Figure 2.31 there is another subroutine for location of parts. In this subroutine firstly variable "**comp**" mentioned above is selected by "**long**" variable named "**bs**". Later than, when the part is being inserted to assembly, it is inserted with their all properties in SolidWorks interface, in short, it is inserted with all features it contains.

```

Sub loc_comp(xtr, ytr, ztr, thx, thy, thz)
  Dim vtr(15) As Double
  bs = asmdl.Extension.SelectByID2(comp, "COMPONENT", 0, 0, 0, False, 0, Nothing, 0)
  Set swcomp = asmdl.SelectionManager.GetSelectedObjectsComponent3(1, 0) 'This method gets the component of the-
  ' selected object in assembly mode.

  If cfix = "fix" Then asmdl.FixComponent: cfix = "": Exit Sub
  If cfix = "unfix" Then cfix = "": asmdl.UnfixComponent

'Orientation hazırlanıyor
pi = 3.141592654
cX = Cos(thx * pi / 180): sx = Sin(thx * pi / 180)
cY = Cos(thy * pi / 180): sy = Sin(thy * pi / 180)
cz = Cos(thz * pi / 180): sz = Sin(thz * pi / 180)

vtr(0) = cY * cz: vtr(1) = -cY * sz: vtr(2) = sy
vtr(3) = sx * sy * cz + cX * sz: vtr(4) = -sx * sy * sz + cX * cz: vtr(5) = -sx * cY
vtr(6) = -cX * sy * cz + sx * sz: vtr(7) = cX * sy * sz + sx * cz: vtr(8) = cX * cY
vtr(9) = xtr: vtr(10) = ytr: vtr(11) = ztr
vtr(12) = 1: vtr(13) = 0: vtr(14) = 0: vtr(15) = 0
Set trans1 = swApp.GetMathUtility.CreateTransform((vtr)) 'oryantasyon matrisi oluşturuldu, This method creates-
'orientation tamamlandı new MathTransform object.

swcomp.Transform2 = trans1: asmdl.ClearSelection2 (All) ' This property gets or sets the component transform. It affects-
' the underlying model geometry| and the display of the component.

```

Figure 2.31 Subroutine for location of parts

The transformation matrix in SolidWorks is defined as

$$\mathbf{T} = \begin{bmatrix} c\theta_x c\theta_z & s\theta_x s\theta_y c\theta_z + c\theta_x s\theta_z & -c\theta_x s\theta_y c\theta_z + s\theta_x s\theta_z & x_{tr} \\ -c\theta_y s\theta_z & -s\theta_x s\theta_y s\theta_z + c\theta_x c\theta_z & c\theta_x s\theta_y s\theta_z + s\theta_x c\theta_z & y_{tr} \\ s\theta_y & -s\theta_x c\theta_y & c\theta_x c\theta_y & z_{tr} \\ 0 & 0 & 0 & 1 \end{bmatrix} \quad (6)$$

where C donates cosine and S denotes the sine.

2.2.2.2 Mate in Assembly

After pushed down the button **Baslat** shown in Figure 2.24 all steps aforementioned till now are performed automatically. Without pressing any other button. This period continues and includes the steps will be explained under assembly operation.

Following the period for part's location on different coordinates, Mate operation becomes a part of an activity automatically as seen in Figure 2.32. Right after the subroutine named "Parcaların Lokasyonu" Mate operation starts automatically as shown in Figure 2.32

```

Set asmb1 = Nothing
swApp.CloseDoc "Dirinler_Pres"

Call parcalistesi
Call parcalari_cagir
Call kaydet
asmb1.ViewZoomtofit2
asmb1.EditRebuild3
Call parcalarin_lokasyonu

Call asmb1.Extension.SelectByID2("Top@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.Extension.SelectByID2("Top@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.AddMate(0, 0, False, 0, 0)

Call asmb1.Extension.SelectByID2("Right@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.Extension.SelectByID2("Right@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.AddMate(0, 0, False, 0, 0)

Call asmb1.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.Extension.SelectByID2("Plane1@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.AddMate(5, 0, False, mesafe, 0)

```

Figure 2.32 Assembly subroutine sequence

In Mate operation features which belong to parts are used as well. These are the features can be used in Mate operation like plane, vertex, edge and face. Among the features aforementioned there are supplementary features, which are added later by user, besides the features assigned by SolidWorks owing to physical structure of them (vertex, point, edge and face). These supplementary features are plane, axis or features like specific local faces created by spline command.

The code about creating plane was shown and explained in Figure 2.16 which is located in previous section. Another feature, that is added by us to subroutine for creating part, is axis. Required code for creating axis is shown in Figure 2.33. In here

an axis, which is perpendicular to a plane and passing from the point specified before, is handled.

```

End If
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.SketchAddConstraints "sgVERTICAL2D" ' Seçili olan çizgiye Diklik özelliği atanmıştır
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 1)
part.SketchAddConstraints "sgPARALLEL" ' Seçili olan iki çizginin birbirlerine paralel olma özelliği atanmıştır.
part.ClearSelection2 True

part.FeatureManager.FeatureCut True, False, True, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False, False, 0, 1
part.SelectionManager.EnableContourSelection = 0 ' Aktif olan Sketch kullanılarak Kesme (Boşluk Oluşturma) işlemi yapılmıştır.

swApp.SetUserPreferenceToggle swSketchInference, False

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "VERTEX", 0, as_2, mesafe, True, 0, Nothing, 0)
part.InsertAxis2 True

swApp.SetUserPreferenceToggle swSketchInference, True

part.SaveAs2 fl0 + "Ayak Sacı.SLDPRT", 0, False, False

```

Figure 2.33 Subroutine for Ayak Sacı - Creating Axis

Meanwhile creating the axis, plane which is wanted to be perpendicular to axis, is selected at first. Next, the vertex coordinates, where axis pass through, X: 0, Y: *as_2* and Z: *mesafe* are selected by using the parameters indicated above. Finally an axis has been assigned which pass through the features mentioned above as shown in Figure 2. 34.

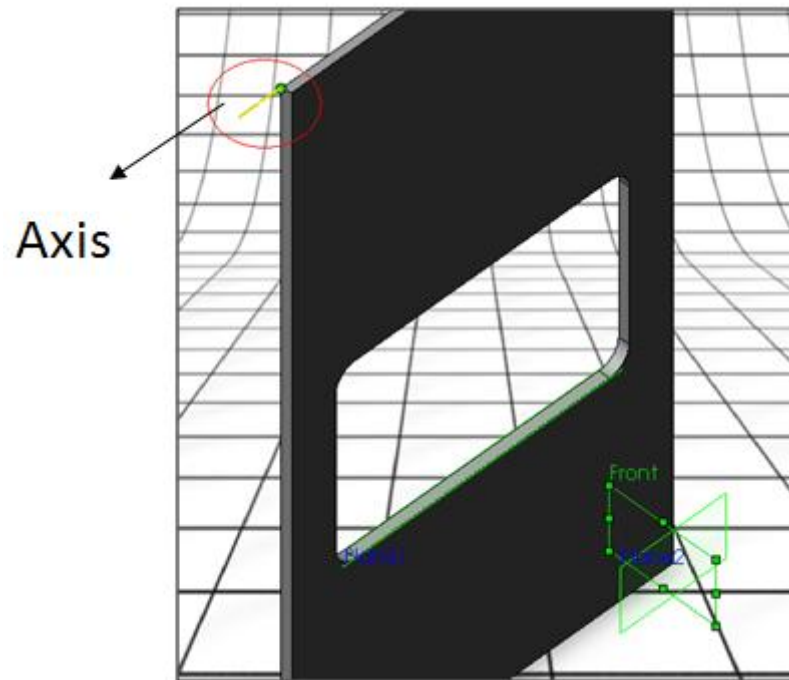


Figure 2.34 Ayak Sacı - Creating Axis

Mate operation, that is shown in assembly procedure in Figure 2.32, is made by using mentioned features. In case of handling the first 3 steps of mate which are shown in assembly procedure above;

```

1 Call asubl.Extension.SelectByID2("Top@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Top@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)

2 Call asubl.Extension.SelectByID2("Right@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Right@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)

3 Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.AddMate(5, 0, False, mesafe, 0)

```

Figure 2.35 Sample (AddMate) operation

In mate number one operation as shown in Figure 2.35 firstly top plane of a part named *Gövde Yan Sacı Sağ_M-1* is selected, and then top plane of a part named *Gövde Yan Sacı Sol_M-1* is selected. “Coincident” mate relation is defined between these two plane. In a similar way in mate number two, right planes of both two parts has been selected and coincident mate relation is defined between two plane. In case of analysing the code about mate operation as follows;

Asmbl.AddMate (MateType As Long, Align As Long, Flip As Boolean, Dist As Double, Angle As Double)

In code given the first component inside parenthesis is "Mate" type and mate types in SolidWorks are listed in Figure 2.36. In sample situated above "Coincident" mate type, presenting in list below, is used. And the number corresponding mate type is "0" as shown in list given below in Figure 2.36.

swMateType_e	
Specifies values for types of assembly mates.	
swMateCOINCIDENT	0
swMateCONCENTRIC	1
swMatePERPENDICULAR	2
swMatePARALLEL	3
swMateTANGENT	4
swMateDISTANCE	5
swMateANGLE	6
swMateUNKNOWN	7
swMateSYMMETRIC	8
swMateCAMFOLLOWER	9
swMateGEAR	10
swMateWIDTH	11
swMateLOCKTOSKETCH	12
swMateRACKPINION	13
swMateMAXMATES	14

Figure 2.36 Mate types in SolidWorks

In command “Asmbl.AddMate” second component inside parenthesis is "Alignment". "-1" represents "Aligned", "0" represents "Anti-Aligned" and "1" represents the "Closest". In parenthesis the third component "Flip" option. In case of adding coincident relation between two faces "Flip" is used for changing the side 180 by means of angle without changing the direction. With “*False*” rotated without, and with “*True*” mate operation is made by rotating. The fourth component inside the parenthesis is "Distance". While mate operation is in progress, distance has to be given in case the fifth type of mate is selected. In here the distance value between two elements is used. In prepared software only the parameters are used as a value of distance. The parameter "*mesafe*" is used as a distance between two parts in mate operation shown in Figure 2.35. Last component inside the parenthesis is "Angle". In case an angle wanted to be given between two parts, the angle value is given in radian as shown in Figure 2.37.

```
Call asmbl.Extension.SelectByID2("Planel@Birc Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmbl.Extension.SelectByID2("Planel@Govde Yan Saci Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmbl.AddMate(6, 1, True, 0, 0.5235987755983) "0.523598..." 30 derece acimin radyani
asmbl.ClearSelection2 True
```

Figure 2.37 Mate “Angle”

In Figure 2.37, primarily two different plane have been selected and an angle is given between these planes as well as in other mate commands. Any parameter has not been used because it is said that the angle between these two plane is fix and not a variable.

How mate types in Figure 2.35 come true as expected, may be seen in Figure 2.38. Firstly, a mate is assigned between two "Govde Yan Saci". And then the other parts are associated with each other as shown in figure. Similar codes have been used for other parts and finally there have been 162 piece of different mate type in the end of assembly process. In the end of mate process press body has got ready for creating the welding seams procedure.

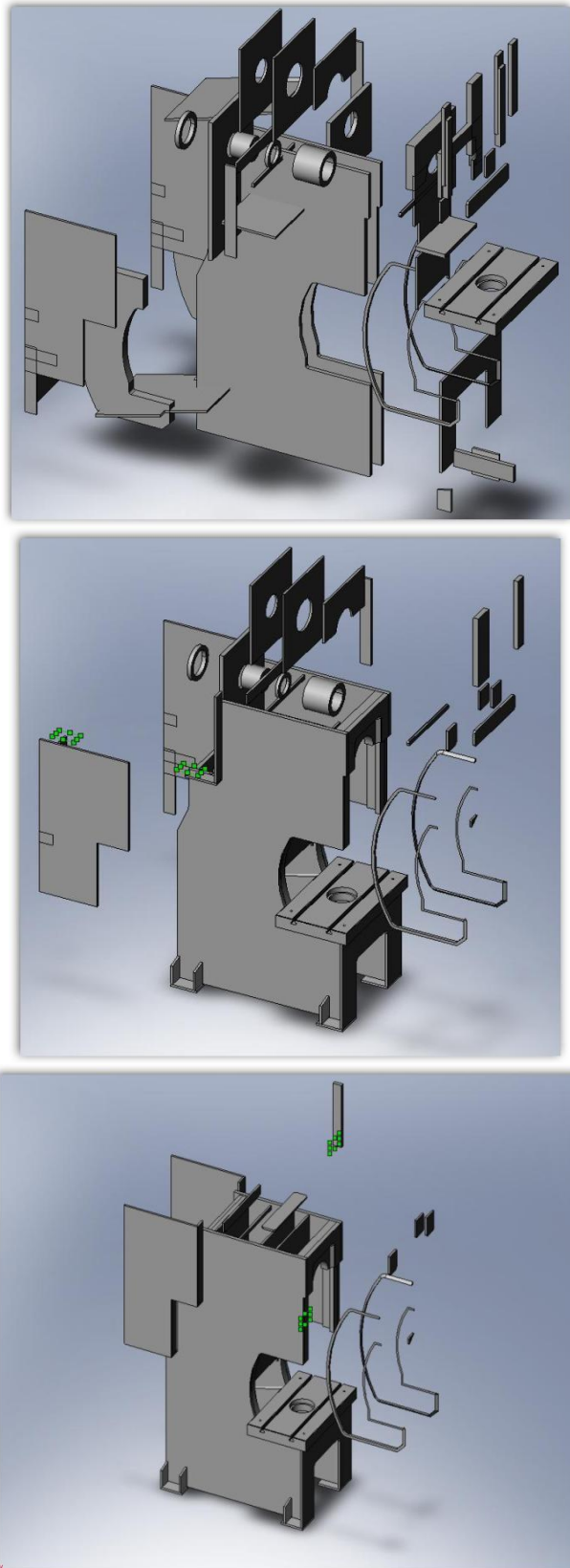


Figure 2.38 Assembling parts in progress

2.2.3 Definition of Welded Connections

In developed software, definition of the weld seams takes place in period of automatic assembly process. In this period weld seams are defined without intervention of user on the lines where two parts intersect. There are two types of weld seams as Fillet weld and Butt weld. There are some matters, before analysing the definition of weld seams in code format, should be considered by user.

In case of analysing about weld seams the space between two parts considered zero, in examined studies in literature. The space will able to be occurred between touching face of parts is shown in Figure 2.39 schematically. These parts are formed by Plasma/Laser cutting method. This space is generally non-uniform in practice as it is shown in the Figure. In order to examine the effects of the space between parts on static stress and strain behaviours of part obtained by welded connection, numerical analyses have been performed by using SolidWorks/CosmosWorks softwares. In consequence of meetings between university and company, it is decided to leave 0.1mm average space between combined parts in welding process, as it is used to be in practice also shown in Figure 2.39.b

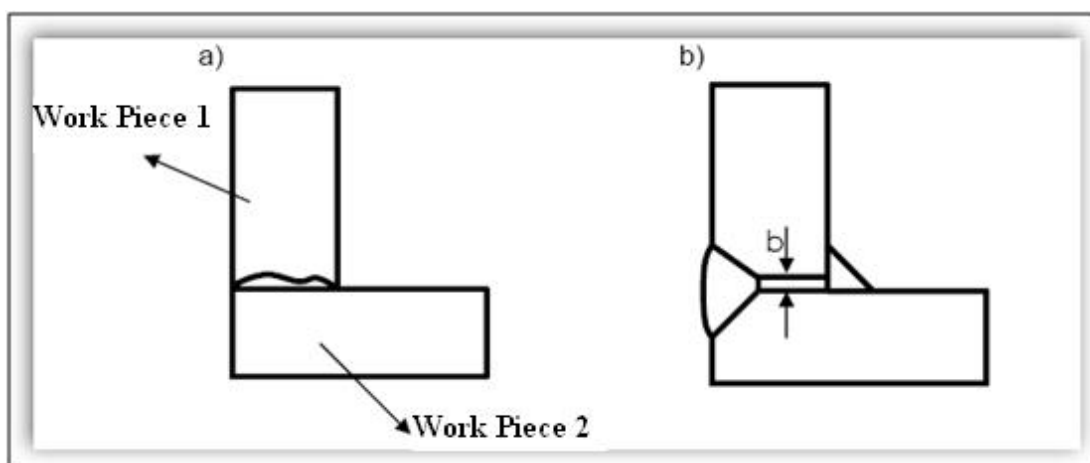


Figure 2.39 Space between work piece a) Real model b) Model for numerical analysis

Thereby, in the finite element analysis of press body under load, load carrying by weld seams is provided. Consequently the analyses on weld seams, which are one of the fundamental aims of this study, are carried out under real conditions.

In order to create the space between two parts going to be associated, some arrangements are made to the parts going to be used in press body assembly during the creation. These arrangements are added to the related subroutines. For example some cutting operations have been done on parts like "Govde Yan Sacı". 0.1 mm has been cut from the surfaces looking inside of press body. Thus, 0.1 mm space has been occurred between parts holding "Govde Yan Sacı" and "Govde Yan Sacı". In this case, the welded combination between any part and "Govde Yan Sacı" is going to be like it is shown in Figure 2.40

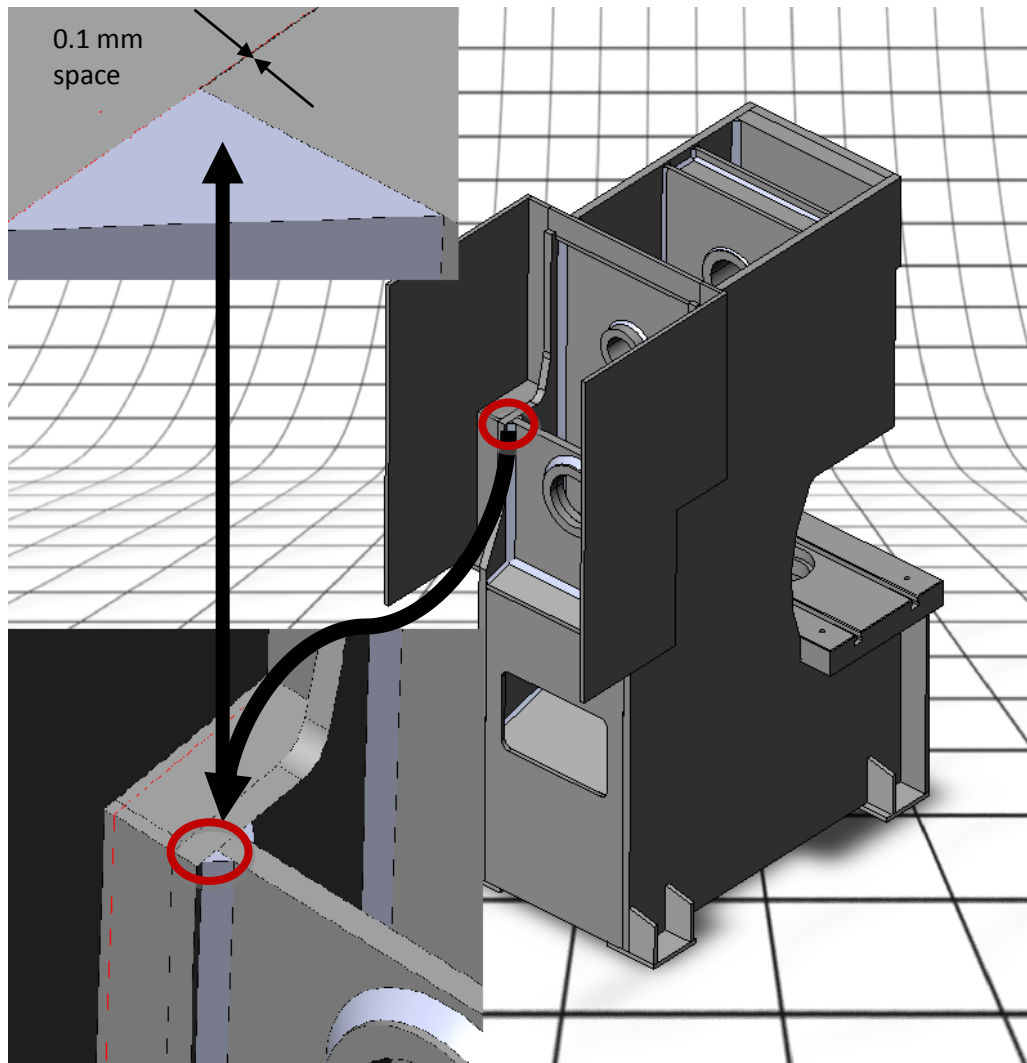


Figure 2.40 Space between parts in assembly

Before welding procedure, another arrangement during the parts are being created is made in regions where butt welds will take place. In order the required weld seams on press body to have acceptable forms as shown in Figure 2.41, inserting some additions about butt welds in the end of codes by which the parts are created, has been found acceptable.

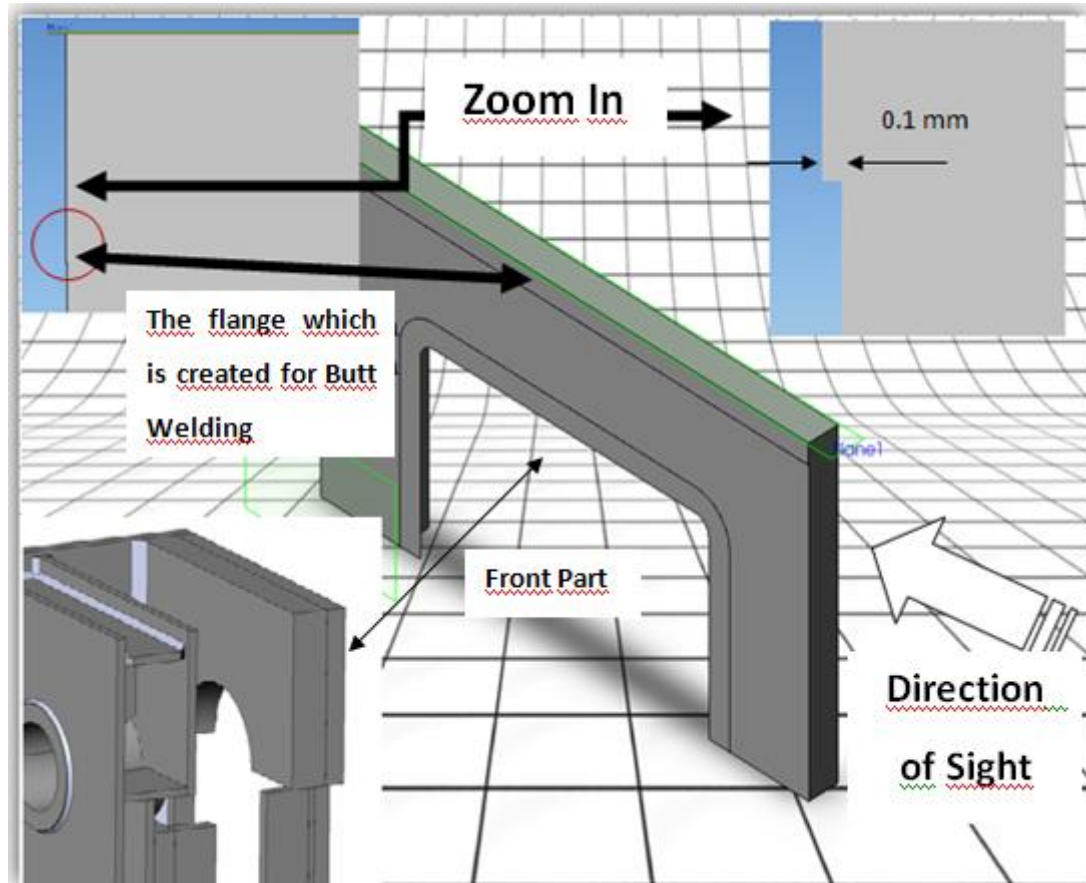


Figure 2.41 Sample Butt weld on "Ön Yatak Mesafe Sacı"

The target geometry is shown in Figure 2.42. In order to get the desired connection, keeping the 0.1 mm space between touching surface of two parts and making butt welding from the top of connection and penetrating sum of welding seam through parts are main steps have to be followed. The butt weld, of parts will be assembled in press body, has been modeled in accordance with these principals. Thickness of welding seam takes place in software as a parameter can be defined by user.

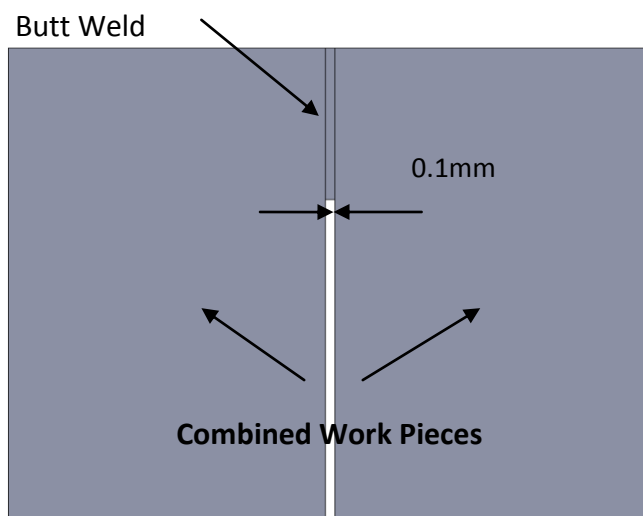


Figure 2.42 Butt weld between two parts

If the existing weld in welding region of press body is butt weld then it is made on part during creating process, else the fillet weld is made automatically in assembly environment by software. Fillet weld is defined across the line where two faces intersect. In order to define weld seam automatically by program, intersecting two faces have to be selected as shown in Figure 2.43

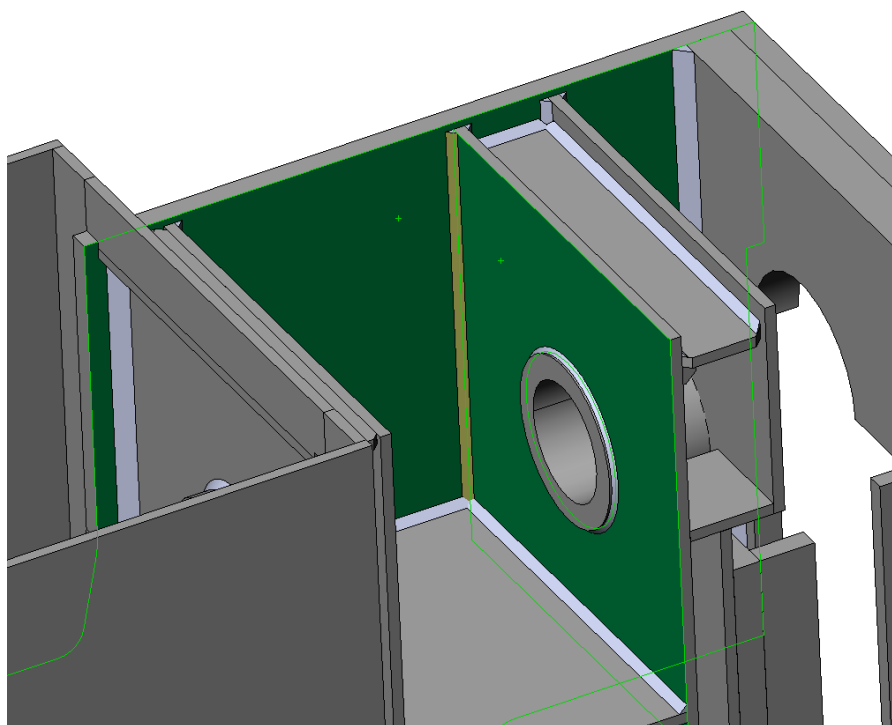


Figure 2.43 Fillet weld for intersecting two face

This operation is performed from the beginning of automatically running assembly operation period as well. After mate process primarily an another subroutine become a part of activity to define fillet weld seams. With the subroutine mentioned above the ledges shown in Figure 2.41 are suppressed in SolidWorks environment. Following the fillet welding process these ledges are Unsuppressed in SolidWorks environment. Codes by which the last mentioned procedures take place are shown in Figure 2.44 and Figure 2.45

```
Call asmdl.Extension.SelectByID2("Extrude2@No 1-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asmdl.EditSuppress

Call asmdl.Extension.SelectByID2("Extrude2@Karsı Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asmdl.EditSuppress
```

Figure 2.44 Suppressing the butt welds

```
Call asmdl.Extension.SelectByID2("Extrude2@No 1-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asmdl.EditUnsuppress

Call asmdl.Extension.SelectByID2("Extrude2@Karsı Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asmdl.EditUnsuppress
```

Figure 2.45 UnSuppressing the butt welds

When the codes are looked over it is obvious that the components Suppressed and Unsuppressed are features belong to parts. For example in first command line, the feature "*Extrude*" number 2 belong to part *No_1-1* in assembly named *Dirinler_Pres* is suppressed. Last mentioned extrude is a ledge for butt weld.

Here is the reason for the procedure mentioned above. In case of necessity of fillet welding across the line where two face intersect, it is mentioned before that the faces have to be selected. However, there has to be 0.1 mm space between two faces. SolidWorks is able to define a fillet weld between these two faces by ignoring (toleration) the space between. But, if the space between these two parts is 0.2 mm or more, SolidWorks is not able to define any Fillet Weld and gives error message indicating that two faces has no common edge by means of not intersecting.

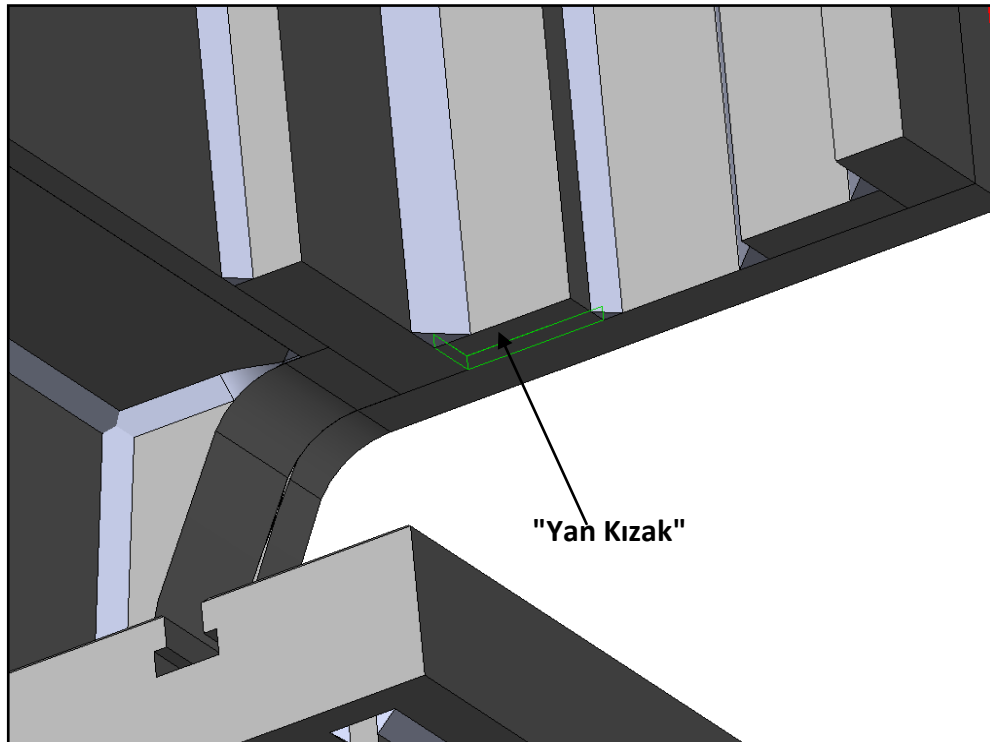


Figure 2.46 Yan Kızak - The part at which butt and fillet welds are used in the same time

The part named "Yan Kızak" shown in Figure 2.46 have both butt welding and fillet welding. In such a case, due to creation of butt welds on parts as explained above, SolidWorks gives an error message when Fillet welds are wanted to be defined in assembly environment. Underlying reason is existence of ledge on part. The ledge (extrude) feature causes to fill the space between parts partially, and it causes the 0.1 mm space disappear. In that case, while the space between two parts was straight line as shown in Figure 2.46.a, it changes form "Z" as shown in Figure 2.47.b

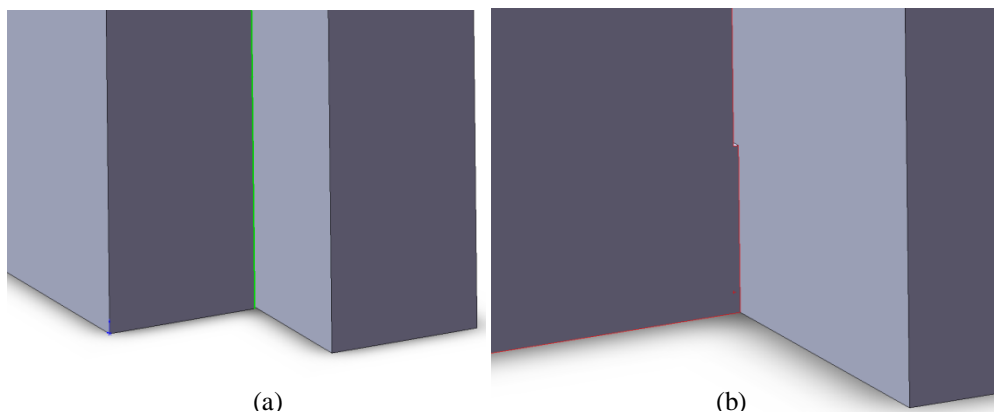


Figure 2.47 Welding connection for Yan Kızak

In defining weld seams process due to intersecting two selected faces the geometry is inconvenient for fillet welding process in such a case the intersection is only across the line as much as butt weld penetrate. Due to these reasons before starting the weld process, the ledges, which existing on parts as butt weld, are suppressed as shown in Figure 2.44. Right after fillet weld are defined automatically by developed software. Finally, the ledges, which existing on parts as butt weld, are unsuppressed as shown in Figure 2.45, in other words changed back in to original conditions.

2.2.3.1 Definition of Fillet Welds

As mentioned before in previous sections, welding process is all about selection of two face and automatically definition a weld seam as much as parameter value all across the line where two faces intersect in SolidWorks environment. It is better to explain these series of process by the help of codes and figures. The algorithm in Figure 2.48 is appropriate sample for defining the weld seam between selected two faces. In codes, it is seen in first line that a subroutine named "Kaynak" is called. This subroutine can be explained by codes take place in Figure 2.49

```
'75 muhafaza_saci---c_saci---ön

Kullanilan_Parca_Kalinligi = ms_parca_kalinligi
x_factor = 0.5
Call Kaynak

Call C_yegelen_Rutin
Call asmdl.Extension.SelectByID2("", "FACE", 0, 0, m_o_öteleme_z + cs_parca_kalinligi, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asmdl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalinligi ^ 2) - cygs_r * Tan(ct_alfa)) - 0.25 *
Else
Call asmdl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalinligi ^ 2) + cygs_r * Tan(ct_alfa)) - 0.25 *
End If
Call asmdl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead75.sldprt")
asmdl.ClearSelection2 True
```

Figure 2.48 Codes for definition of welding

```
Sub Kaynak()
```

```
' x_factor = kaynak dikişi işleminde tek taraftan kaynak yapılıyorsa 0.7 , Cift taraflı kaynak yapılıyorsa 0.5 alınır.
```

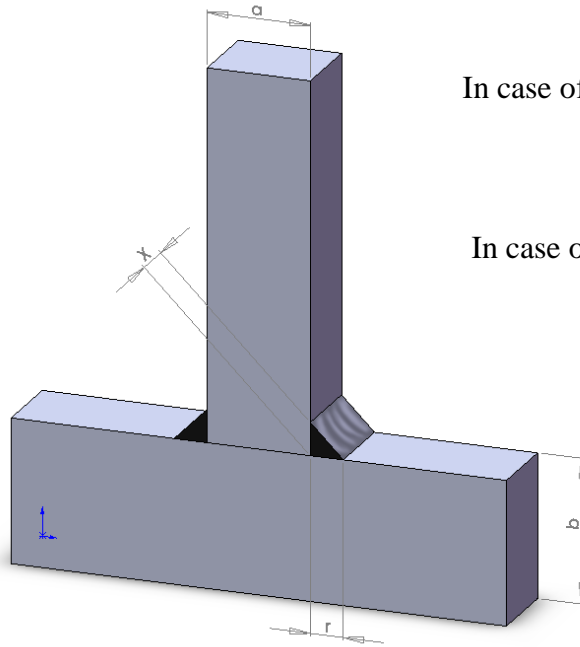
```
Kaynak_Dikisi_Boyutu = Kullanilan_Parca_Kalinligi * x_factor
```

```
Kaynak_Dikis_Boyu = Kaynak_Dikisi_Boyutu * Sqr(2)
```

```
End Sub
```

Figure 2.49 Kaynak Subroutine

As it is shown in Figure 2.50; the explanation stated by producer in first command line in Figure 2.49 is a common application used in determination of weld seams.



If $a < b$;

In case of welding both side of wok piece;

$$(X = 0.5 \times a)$$

In case of welding one side of work piece;

$$(X = 0.7 \times a)$$

Figure 2.50 Determination of weld seam size

The radius value entered to program during welding process in SolidWorks environment is shown with "r" in Figure2.50. The weld seam size is entered to the program considering the formula in Figure 2.51

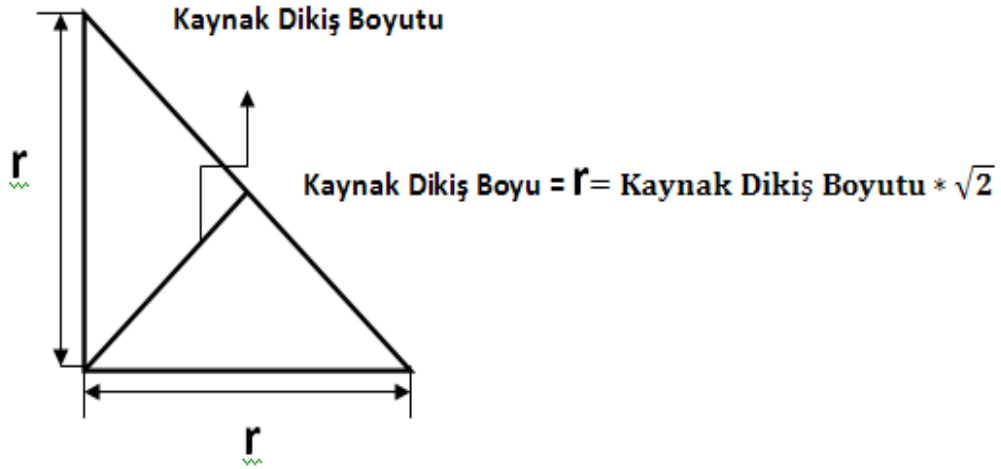


Figure 2.51 Weld seam geometry and parameters “Kaynak Dikiş Boyutu”, “ r ”, and “Kaynak Dikiş Boyu” which are used in “Kaynak” subroutine

As it is shown in Figure 2.48 the subroutine for defining weld seams includes different parameters and equations for every welded region. The code takes place in Figure 2.47 is only for the weld seam number 75 and it is defined between "C Sacı" and "Muhafaza Sacı" as shown in Figure 2.52. When the subroutine for weld seam number 75 is looked over firstly a subroutine named *C_ye gelen Rutin* and then an another subroutine used in creation of Muhafaza sacı are called. In this subroutine, there are some algorithms to change the inclination existing on "C' ye Gelen Sac" in accordance with parameter changing and some algorithms to keep the contact between faces where "C'ye Gelen Sac" and "C Sacı" intersect. These mentioned codes and methods are all available in appendices. Later on there is another subroutine for selection the faces where weld seams are going to be defined. In case the inclination angle of "Muhafaza Sacı" shown in Figure 2.52 and which is able to change in accordance with changing parameters, higher than 90° or lower than 90° , coordinate of the selectable point is determined in non-linear relation. In any case of changing the mentioned algorithm is used in order the face to be selected.

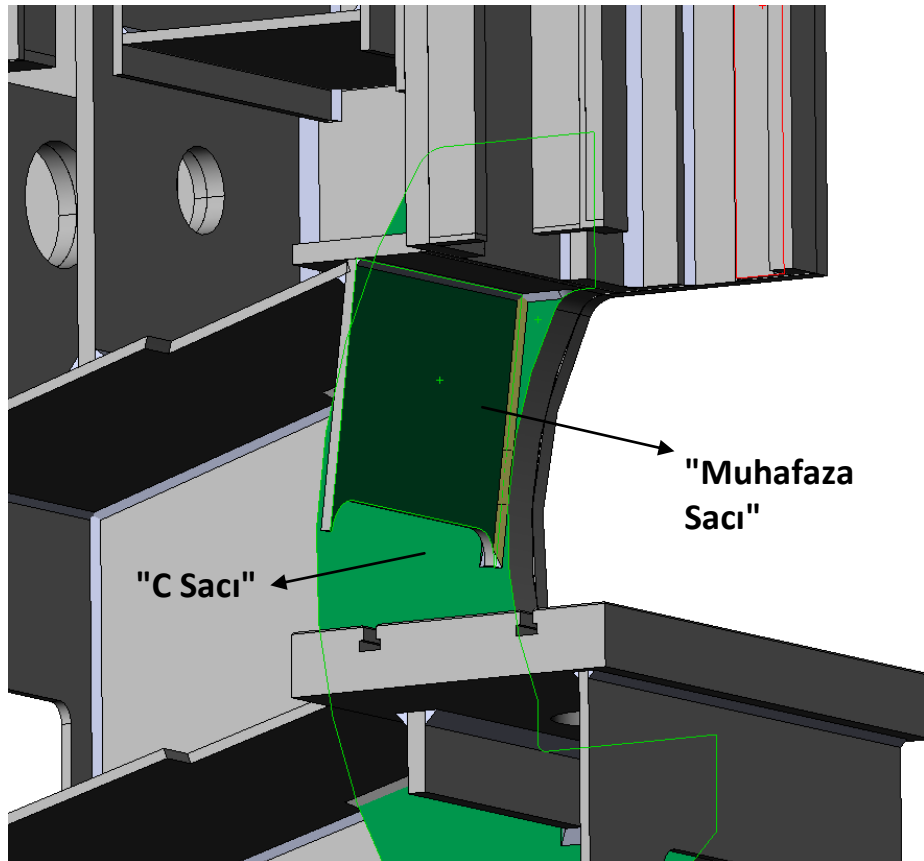


Figure 2.52 Number 75 weld seam

Due to not capturing the view completely in Figure 2.48, the algorithm inside the code is given below again.

If $ct_x < (\pi / 2)$ Then

```
Call asmb1.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 +
ms_parca_kalın1ığı ^ 2) - cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + Sqr(ms_x ^ 2
+ ms_parca_kalın1ığı ^ 2) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) +
m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
```

Else

```
Call asmb1.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 +
ms_parca_kalın1ığı ^ 2) + cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + Sqr(ms_x ^ 2
+ ms_parca_kalın1ığı ^ 2) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) +
m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
```

End If

Weld seams are defined by using parameters that can be changed by user. In developed software there are two different parameters in order to change the weld seams. One of them is the thickness of parts, and the other is “*x_factor*” take place in the first command line in Figure 2.48. Parameters are both able to be changed by user. Due to the fact that the fundamental purpose of this study is analyzing the weld seams in press body, three different press bodies having different weld seam sizes are presenting in Figure 2.53.

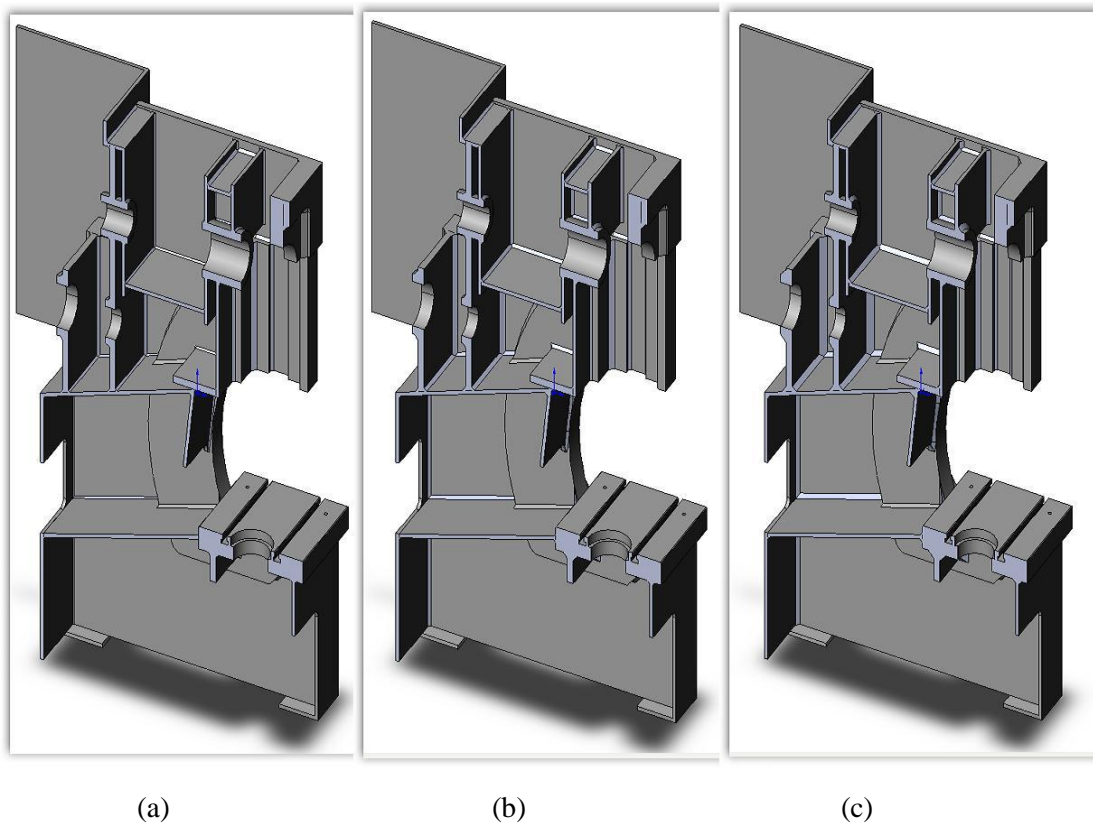


Figure 2.53 Press body section views having different weld seam sizes.

In sample take place in Figure 2.52(a) *X_Factor*: 0.3, in (b) *X_Factor*: 0.5 and in (c) *X_Factor*: 0.7. The results of analyses for the samples given above are examined in details in the next sections.

As mentioned in automatic assembly period, welding process is finished after suppressing the butt welds defining the fillet welds and unsurprising the butt welds. Press body is automatically saved in assembly format with last changing.

2.2.4 Making the Press Body Ready for Analyses

Automatically progressing period for assembly continues by pushing down only one button. After press body is saved in assembly format some series of procedures are applicated. The main purpose of doing this is performing the analysis of press body without any problem. It is better the analysed part to be in a single piece in order to mesh the part for the finite element analysis with SolidWorks 2009 Simulation (CosmosWorks) used for analysis procedure. In case press body is in assembly format, in other words, in case press body composes of many different parts, mesh operation for finite element analysis fails. Due to the fact that press body composes of too many parts and due to the complex geometry of body causes analysis operation to fail and existing hardware and software are incapable for mesh process in assembly format. In order to eliminate last mentioned problem some series of codes, going to be explained below in Figure 2.54, are added in software

```

Call kaydet
asmb1.ClearSelection2 True

    asmb1.ViewZoomtofit2
    asmb1.EditRebuild3

Call part_olarak_kaydet

Call Assembly_Gövdeyi_Kapat

Call Part_Gövdeyi_Cagır

Call Kesme

Call Birlestirme

Call Simetrik_Mirror_ile_Tekrar_Birlestirme

```

Figure 2.54 Preparation of press body for analysis

After saving process, the press body is saved as "Part" format this time as seen from the Figure 2.55.

```

Sub part_olarak_kaydet()

asmb1.SaveAs2 "D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres.SLDPRT", 0, True, False

End Sub

```

Figure 2.55 Saving the press body as "Part"

Afterwards, the press body in assembly format which is still active on work screen is closed owing to will be worked on single file. And then press body which is saved as "Part" format is recalled. (Shown in Figure 2.56 and Figure 2.57)

```

Sub Assembly_Gövdeyi_Kapat()

Set asmb1 = Nothing
swApp.CloseDoc "Dirinler_Pres"

End Sub

```

Figure 2.56 Closing the assembly file

```

Sub Part_Gövdeyi_Cagır()

Dim longstatus As Long, longwarnings As Long

Set part = swApp.ActiveDoc

Set part = swApp.OpenDoc6("D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres.SLDPRT", 1, 0, "", longstatus, longwarnings)
Set part = swApp.ActivateDoc2("Dirinler_Pres", False, longstatus)

End Sub

```

Figure 2.57 Calling part file

Press body has a symmetric geometry and it is approved to use this property of press body. The advantage of symmetric body is possibility of duplication all existing applications made on one side to the other side. By taking the advantages of this situation, all weld seams are defined on Left side of press body. Because of this the press body which is saved as "part", has been cut from the plane of symmetry as shown in Figure 2.58

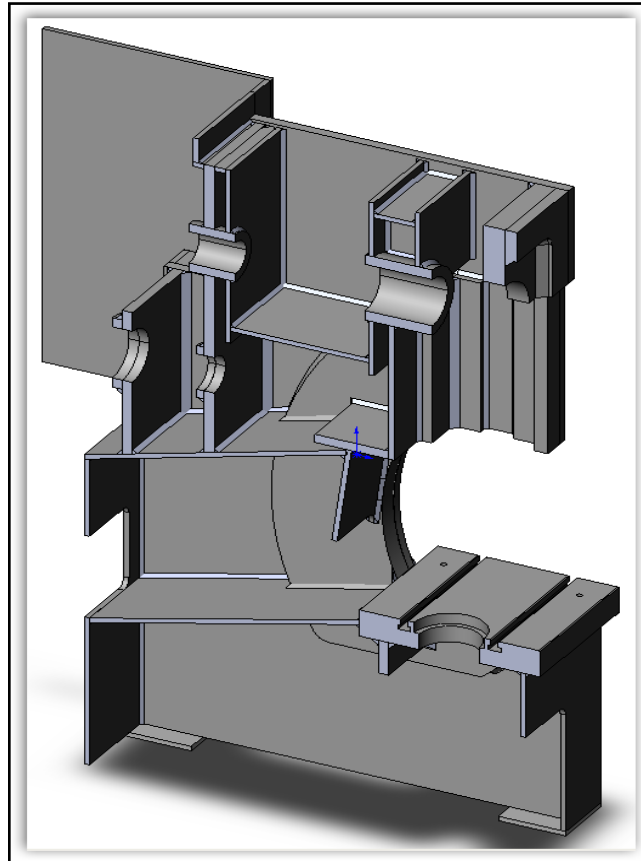


Figure 2.58 Half model of press body

After cutting process there are 122 different parts in "Part" format press body. The features occurred after cutting process and the weld seams are included. As mentioned before, in order to define finite element mesh without any problem it is better the part, which is going to be analysed, to have single piece volume. So as to provide this condition, "Combine" feature in part interface is used. With "Combine" feature it is possible to unite all parts and to have single piece volume as shown in Figure 2.59

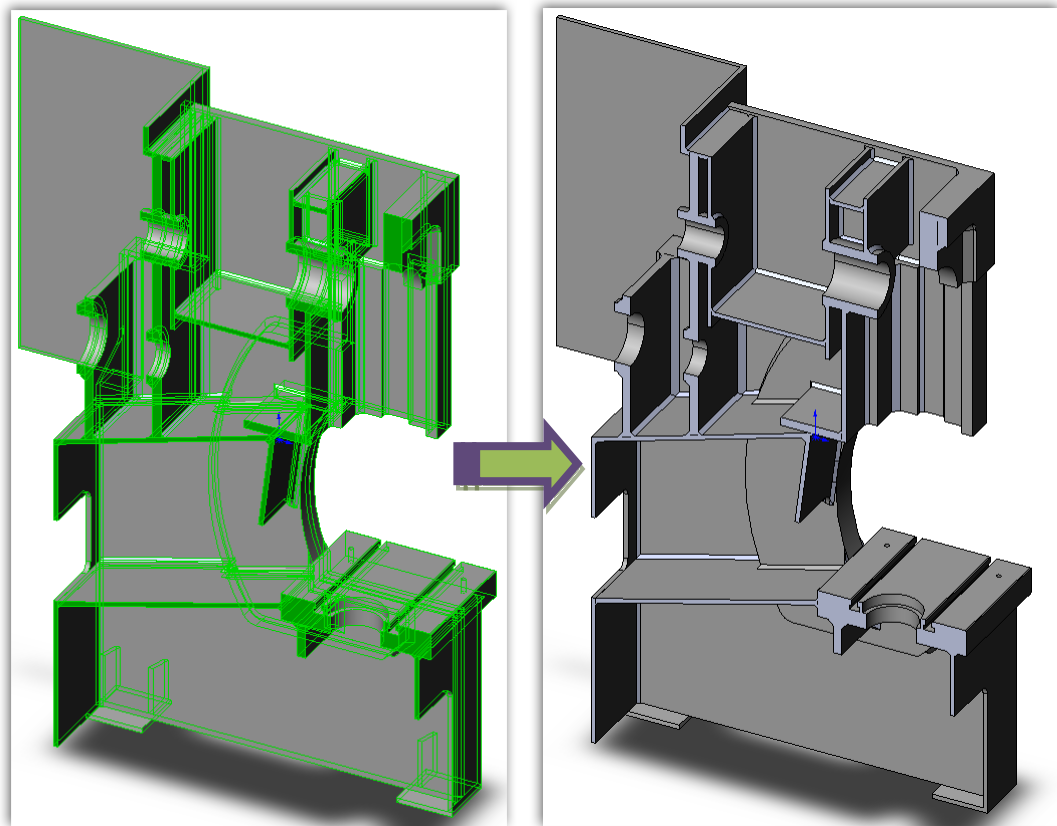


Figure 2.59 Single piece volumed press body by Combine process

The codes for Combine process are shown in Figure 2.60. Firstly, all parts, which are different from each other, are selected. Later, with the command "Combine" press body is become monoblock solid body. In Figure 2.60 some of selected parts are presented as sample. With software 122 piece of solid parts are selected one by one and finally, with command "*InsertCombineFeature*" all of them are combined.

```
boolstatus = part.Extension.SelectByID("Govde Yan Sacı Sag_M-1-solid1", "SOLIDBODY", 0, 0, 0, False, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID("C_Sacı-2-solid1", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID("Ayak-1-solid1", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID("Ayak-3-solid1", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID("Ayak Federi-4-solid1", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
part.FeatureManager.InsertCombineFeature 15903, Nothing, Empty
```

Figure 2.60 Codes for Combine process

After combining process, finally "Mirror" feature takes place in procedure. Thus, left side which is occurred as single piece body by combining after welding process finished, is mirrored and all properties belong to left side copied to right side as well. In this way, the press body which has the similar properties at both sides of body is created. As shown in Figure 2.61 with the feature "Mirror" complete press body is obtained.

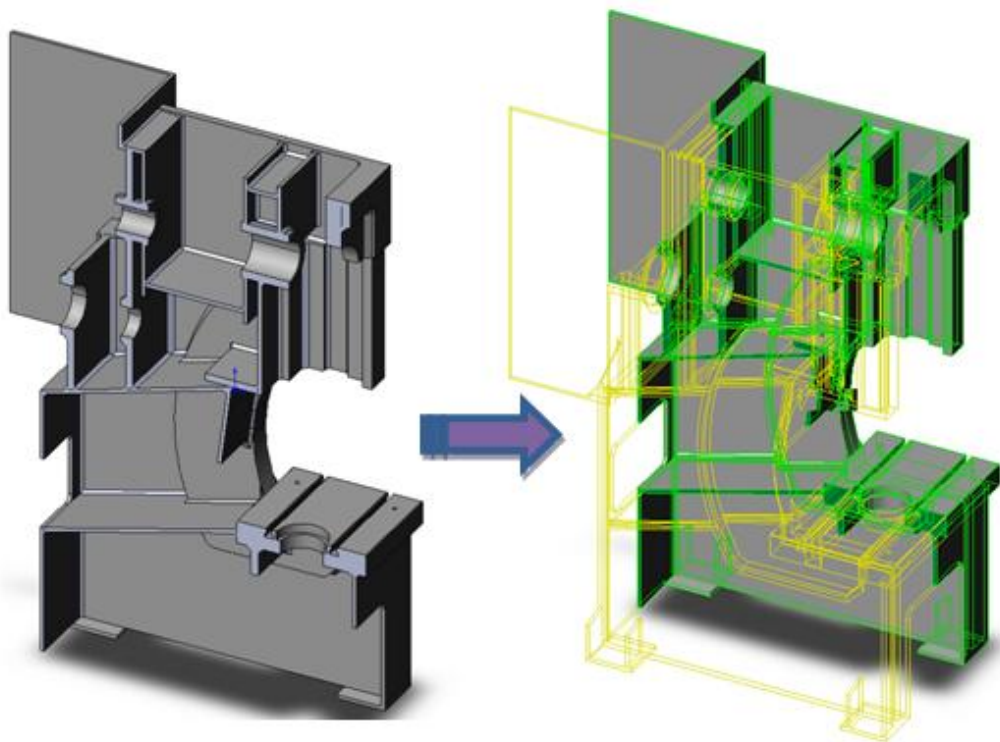


Figure 2.61 Mirror Process

The "Mirror" command is used in codes as shown in Figure 2.62.

```
Sub Simetrik_Mirror_ile_Tekrar_Birlestirme()
```

```
' Asagıda mirror yapılırken secilen yuzey, ayak sacının sol alt kosesidir. Bu yuzeyin ve kullanılan koordinatlar, her  
"Modelde, her farklı modelde yer alacağı düşünülüğü için kullanılmıştır.
```

```
m_o_öteleme_x = -gys_1 / 2
```

```
m_o_öteleme_y = -gys_10 / 2
```

```
m_o_öteleme_z = (parca_kalınlığı / 2 - parcalar_arasi_bosluk / 2)
```

```
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", m_o_öteleme_x + 0.01, m_o_öteleme_y + 0.01, m_o_öteleme_z + mesaf
```

```
boolstatus = part.Extension.SelectByID2("Combine1", "SOLIDBODY", 0, 0, 0, True, 256, Nothing, 0)
```

```
part.FeatureManager.InsertMirrorFeature True, False, True, True
```

```
part.Save2 False
```

```
End Sub
```

Figure 2.62 Codes Mirror command

A surface which is selectable in any case of combination of press body, is selected. Due to the last process is Combine operation, SolidWorks defines this feature as *Combine1* automatically. Owing to only solid body is *Combine1*, *Combine1* is selected as solid body. Finally, mirror operation is performed by using the selected items. Herewith, press body is ready to be analysed as given in Figure 2.63.

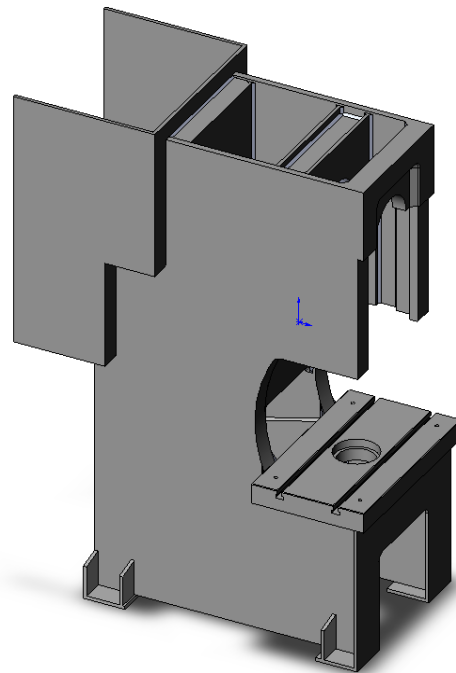


Figure 2.63 The press body ready for analysis

2.2.5 Scanning the Parameters

An algorithm has been developed in order to determine in which subroutines the used parameters are active. The main purpose of mentioned algorithm is able to see which parameter is directly effective in creation in which part. Thus, the parts required to be recreated, which are effected by the changed parameters, are listed on the user interface. The mentioned parameter scanning takes place on part creation display as shown in Figure 2.64.

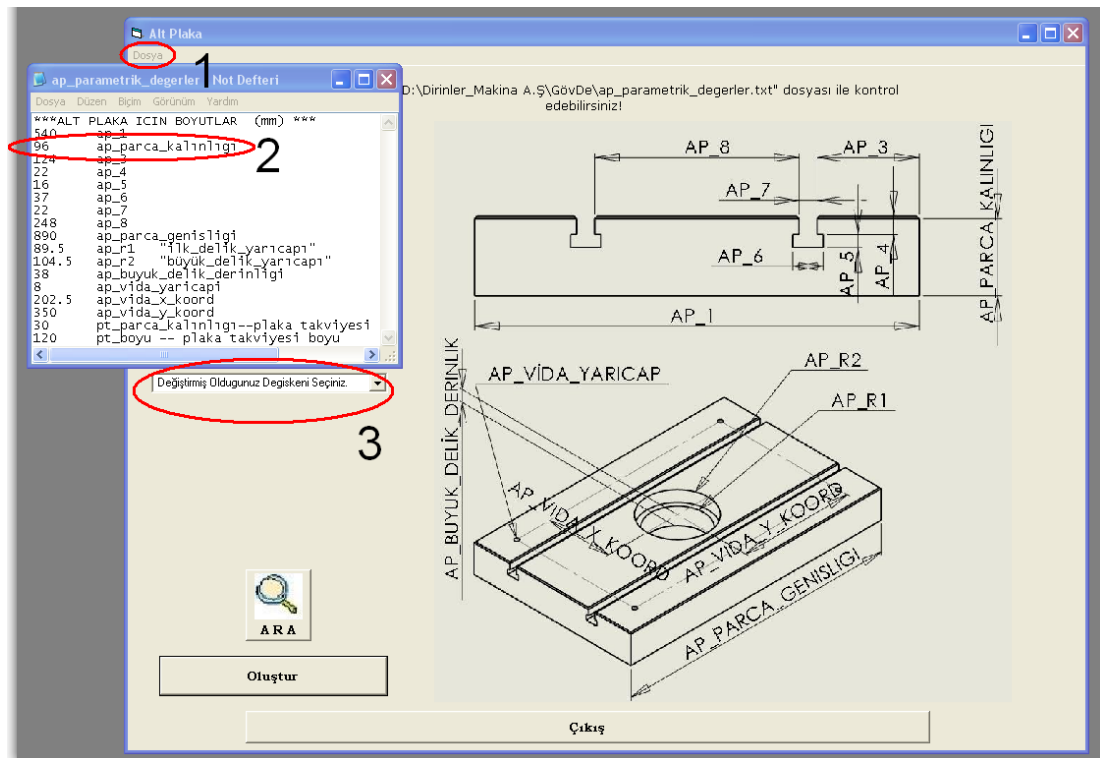


Figure 2.64 Parameter Scanning on Interface

At first, the "text" file where the parameters are stored is opened by clicking the "Dosya" on toolbar on part creation interface as numbered as 1 in Figure 2.64. The parameter needed to be changed is changed in text file as shown in 2. Finally, the text file is closed after saved. In region where shown as 3 in Figure 2.64, there is a "ComboBox". The name of variable parameters, which belong to displaying part and store in the openable text file "dosya" appearing on the top of interface, are listed in "Combobox" as shown in Figure 2.65

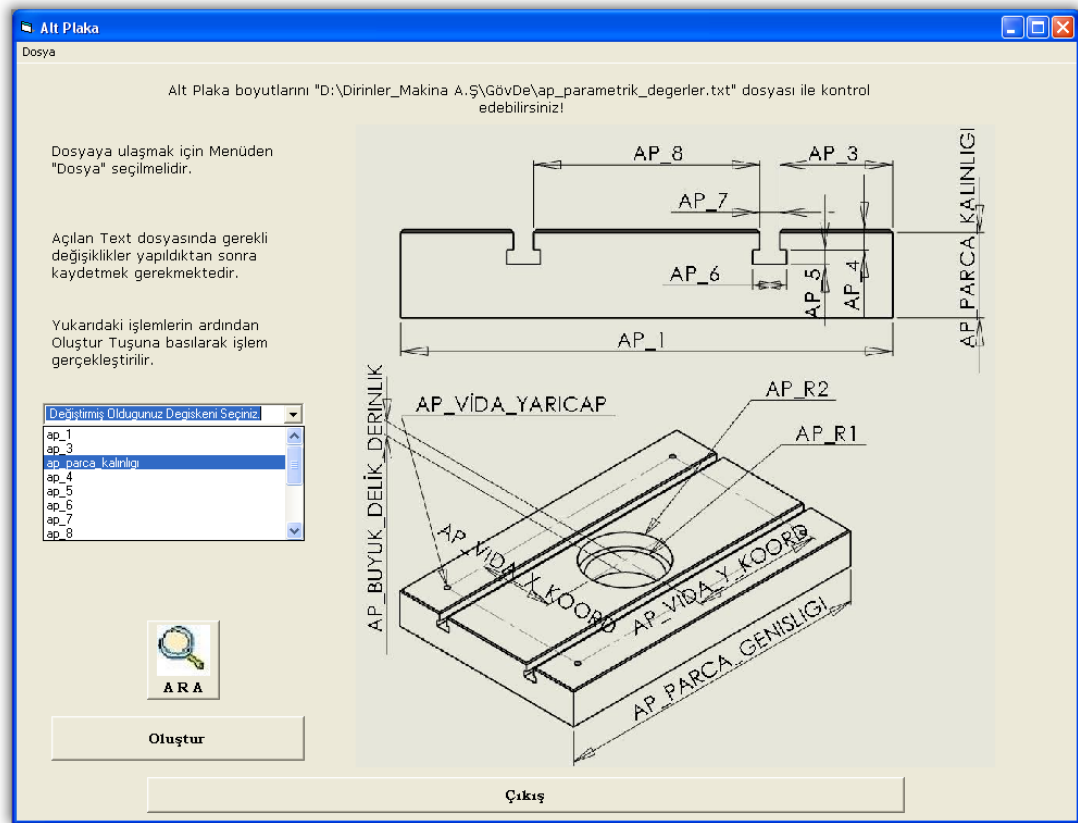


Figure 2.65 Parameter scan on Interface – Parameter Selection

On condition that the parameter named “ap_parca_kalınliği” is the one has changed by user, the parameter which has changed before is selected from "ComboBox" locating on part creation form and the button "Ara" is pushed down as shown in Figure 2.65. Search result is listed in the list locating underneath the "ComboBox" which is given in Figure 2.66. The alteration which is made on parameter, directly affects the parts listed in the list. The parts listed in the list have to be recreated by the user. Otherwise in assembly environment, the unchanged part will not able to be assembled to press body properly and during the assembly SolidWorks is going to give "Mate Errors". Because of these, parameter scan should be done and the part affected by changed parameter should be recreated. On condition that any parameter has been changed, the listed parts are recreated and if it is followed by the other different parameter changing, furthermore if the part recreated before takes place again between the parts listed after parameter scan, aforementioned part should be recreated again.

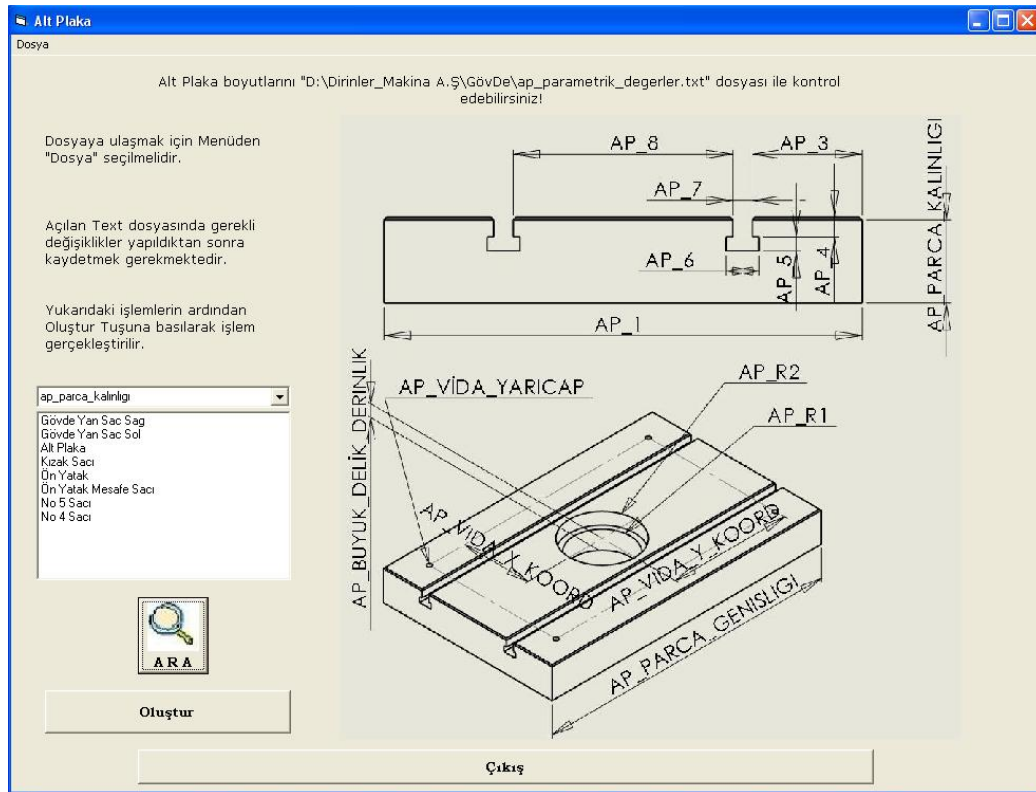


Figure 2.66 Parameter scan on Interface and Part List

Scanning has to be done after any parameter changing. Parts have to be created in accordance with the last values of parts to prevent any error in assembly environment. How the parameter named “ap_parca_kalnlığı” affects the parts take place in list, may be explored as sample shown in Figure 2.66

```

part.CreateArc2 gys_merk_x, gys_merk_y, 0, xp2, yp2, 0, xp1, yp1, 0, -1
part.ViewZoomtofit2
part.CreateLine2 gys_6, gys_5, 0, gys_7, gys_5, 0
part.ViewZoomtofit2
boolstatus = part.Extension.SelectByID2("Point8", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r3, 1
part.ClearSelection2 True

part.CreateLine2 gys_7, gys_5, 0, gys_7, gys_2 + ap_parca_kalnlığı + eksen_1 - eksen_3, 0
part.CreateLine2 gys_7, gys_2 + ap_parca_kalnlığı + eksen_1 - eksen_3, 0, gys_9, gys_2 + ap_parca_kalnlığı + eksen_1 - eks

```

Figure 2.67 Usage of “ap_parca_kalnlığı” in creation of Govde Yan Sacı

In Figure 2.67 it is shown that how the parameter named "ap_parca_kalınlgı" has been used in command lines for Govde Yan Sacı. The other sample is about usage of same parameter in command lines of part named Kızak Sacı shown in Figure 2.68.

```
part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", mesafe / 2, ((eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı) / 2), 0, True, 4, Noth
part.FeatureManager.FeatureExtrusion2 True, False, 0, 0, ks_parca_kalınlgı, 0, False, False, False, False, 0, 0, False, False, False, False, 1, 1, 1, 0, F
part.SelectionManager.EnableContourSelection = 0
```

Figure 2.68 Usage of “ap_parca_kalınlgı” in case Kızak Sacı is created

The parameter “ap_parca_kalınlgı” has been used during the creation of part named Alt Plaka. Sketch selection is required in order to use "Extrude" feature. Also, in last sample given above, the parameter “ap_parca_kalınlgı” has been used while sketch was being selected.

As it is shown from the figures (samples) given above, controlling the changes in parameters, exploring the relations between parts and parameters and performing the required alterations on parts are the most critical steps supposed to be followed by the user. Otherwise facing with unexpected problems and error messages are foregone conclusion as a result of chain reactions.

CHAPTER THREE
NUMERICAL AND EXPERIMENTAL ANALYSES
ON TEST SAMPLE

3.1 Test Rig and Test Samples

In the scope of thesis, in order strain and laser displacement systems to be tested and in order to make preparations for the measurements going to be made, displacement and strain measurements have been carried out with experimental parts having different welding properties. The test rig is shown in Figure 3.1

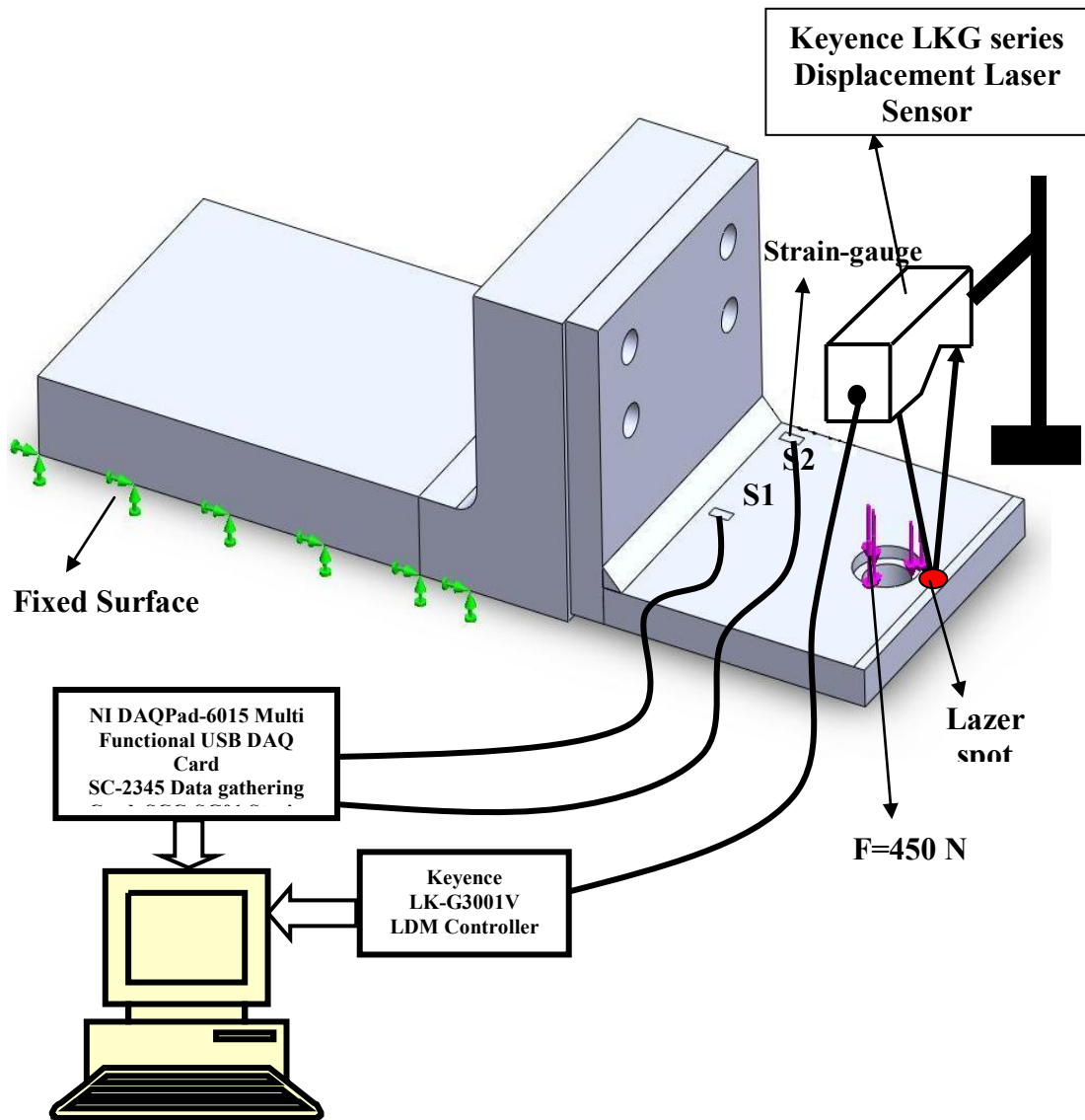


Figure 3.1 Test Rig for welded test sample L plate (Schematic)

Real measurement system is shown in Figure 3.2. The L shaped welded test plate is loaded by using six pieces of loads each weighs 9 kg. Total load is 450 N.

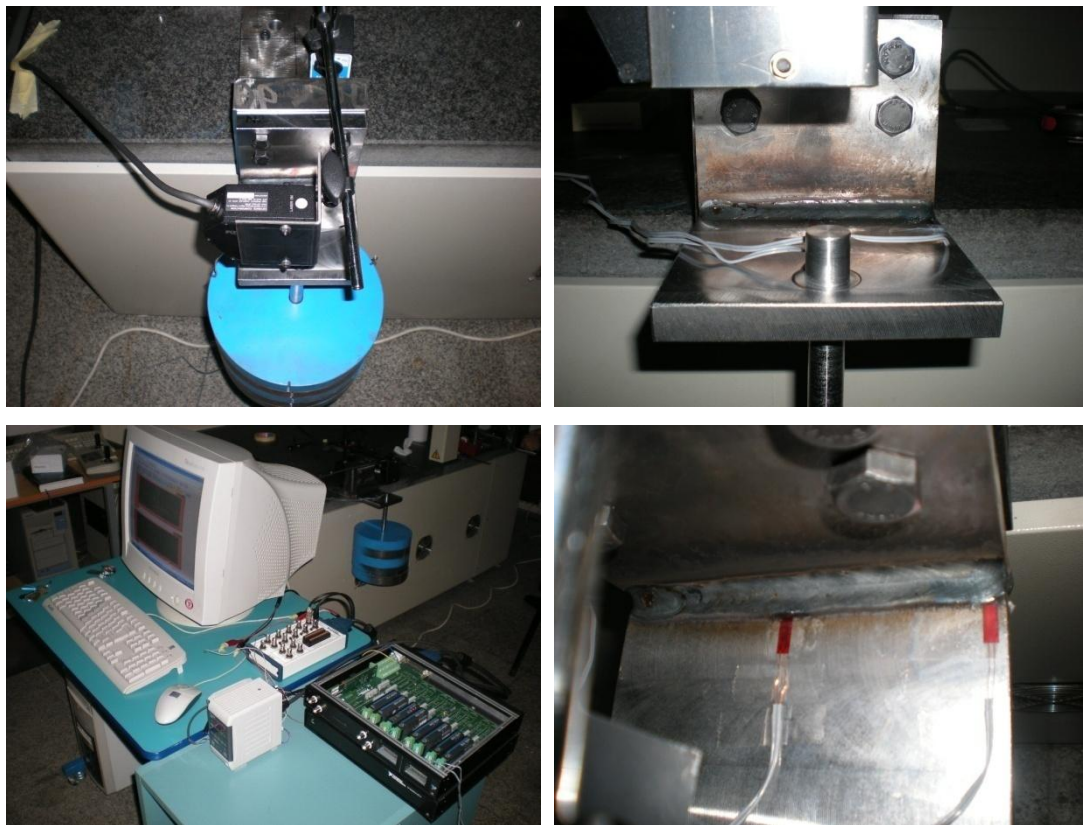


Figure 3.2 Test rig for welded L plate

Five different models have been produced for welded L plate. Weld sizes and welding methods have been changed in these models presented below in Figure 3.3.

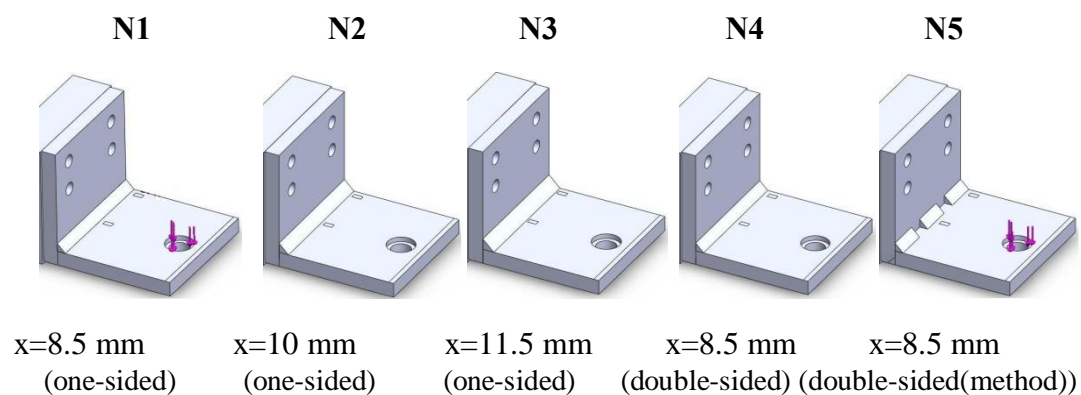


Figure 3.3 Solid models of welded L shaped test samples

Real image belong to test samples is presented in Figure 3.4.



Figure 3.4 Welded L shaped test samples

3.2 Test Results

Prepared test samples are subjected to 450 N vertical loading. Strain values of two regions close to weld seam and the displacement values on edge of plate are measured with laser displacement and strain gauge measurement systems which are purchased in the scope of thesis. Loading is carried out putting 9 kg of disks in sequence. Sampling frequency is chosen 500 Hz for strain measurements and 1000 Hz for displacement measurements. Strain and displacement results obtained from the test are given below.

3.2.1 Test Sample 1 (N1)

Displacement and Strain results for N1 are given in Figure 3.5, Figure 3.6 and in Figure 3.7.

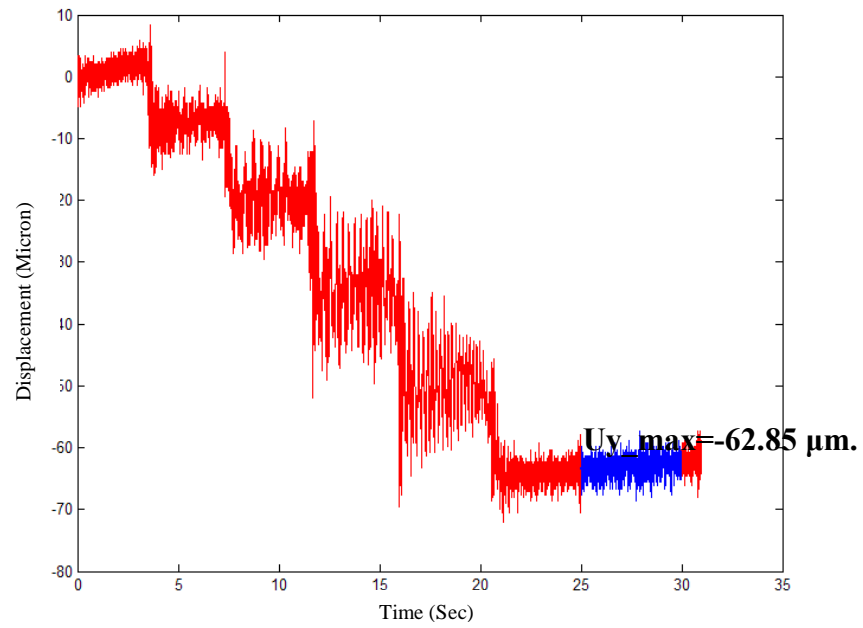


Figure 3.5 Displacement result for N1

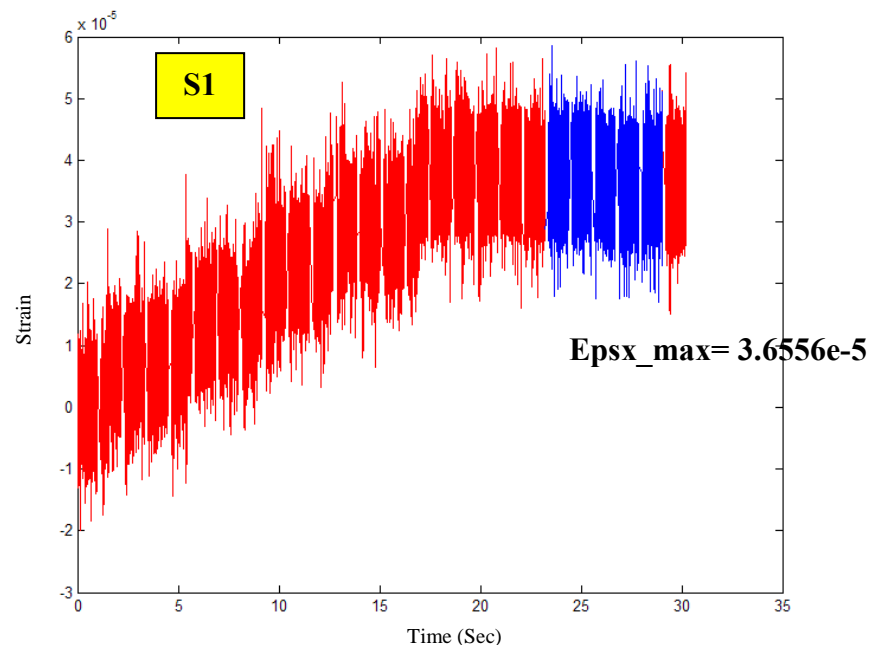


Figure 3.6 Strain result for N1, strain-gauge S1

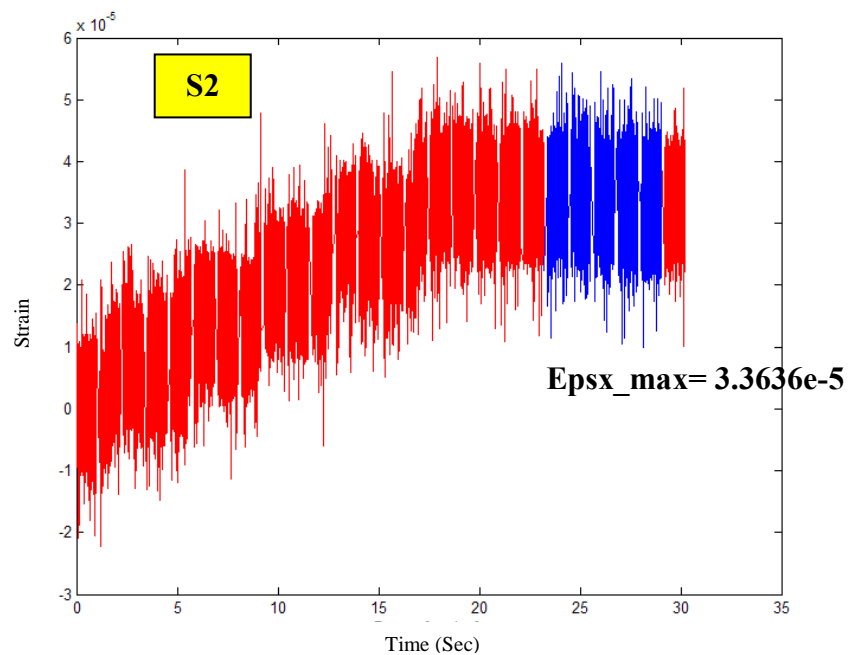


Figure 3.7 Strain result for N1, strain-gauge S2

3.2.2 Test Sample 2 (N2)

Displacement and Strain results for N2 are given in Figure 3.8, Figure 3.9 and in Figure 3.10.

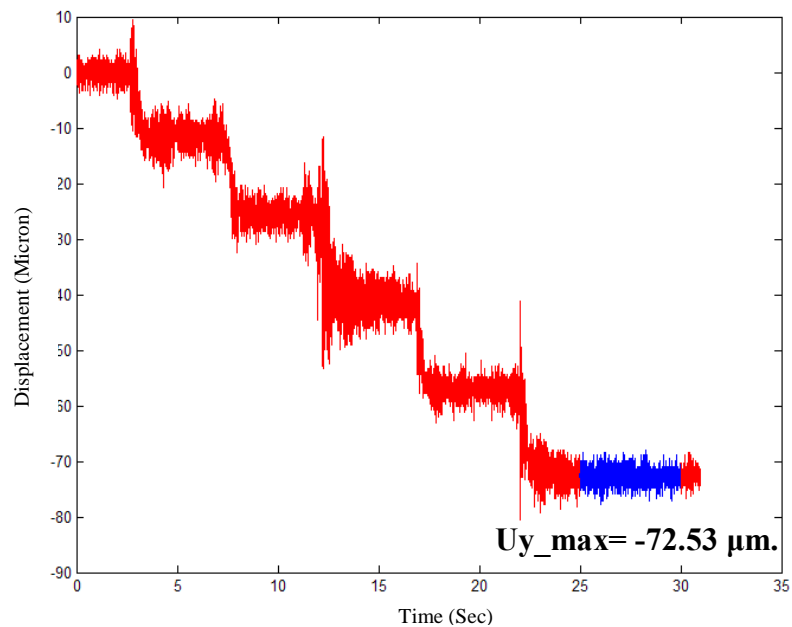


Figure 3.8 Displacement result for N2

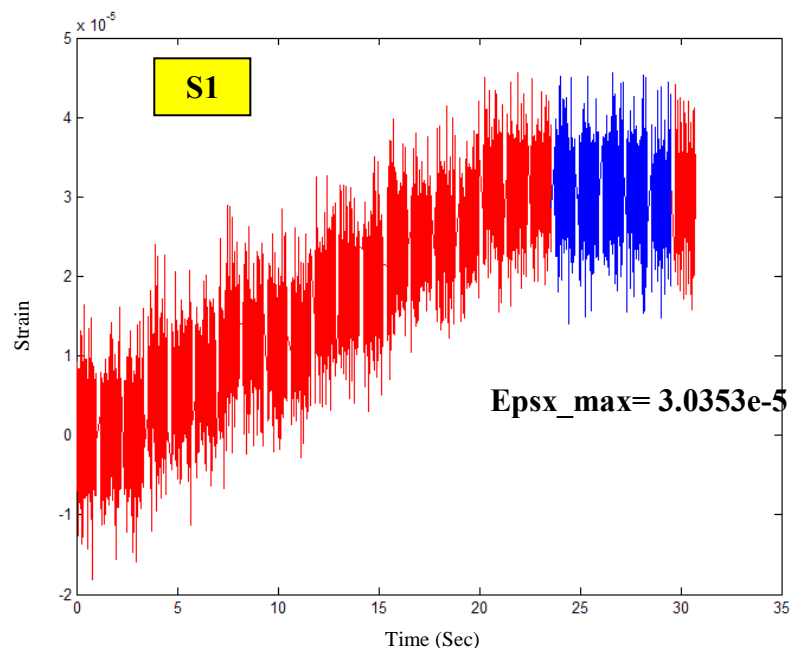


Figure 3.9 Strain result for N2, strain-gauge S1

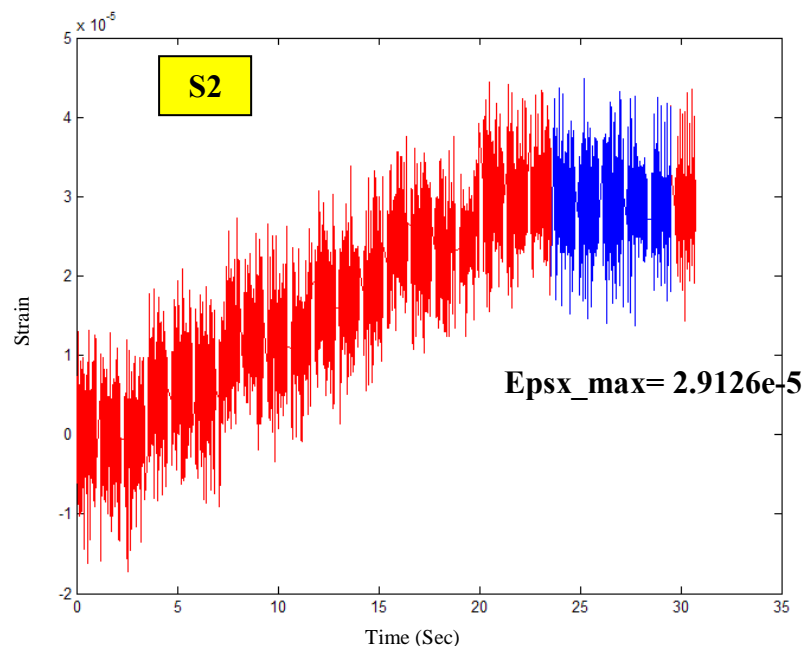


Figure 3.10 Strain result for N2, strain-gauge S2

3.2.3 Test Sample 3 (N3)

Displacement and Strain results for N3 are given in Figure 3.11, Figure 3.12 and in Figure 3.13.

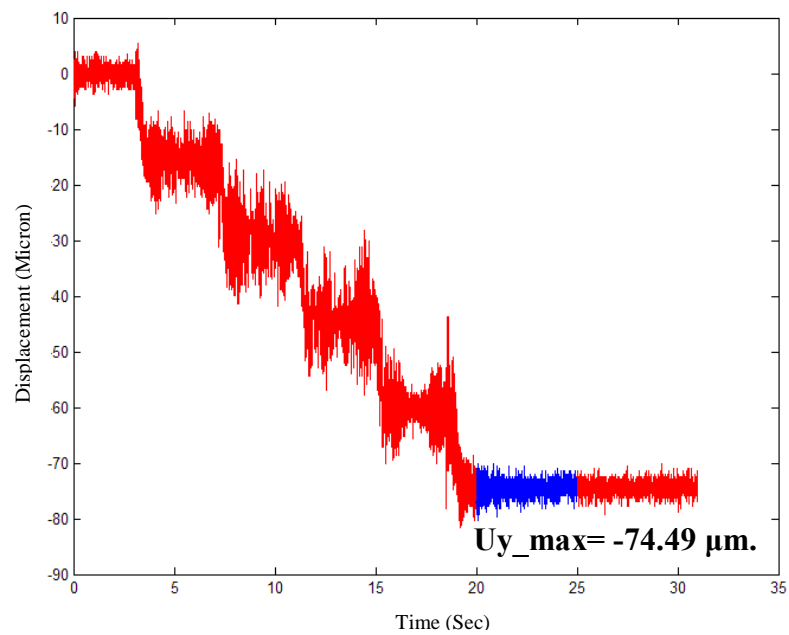


Figure 3.11 Displacement result for N3

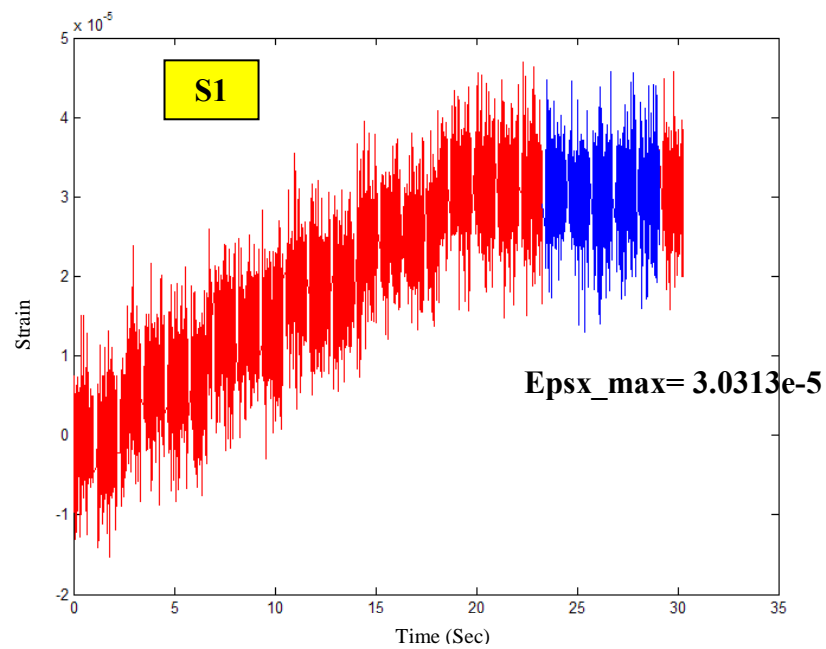


Figure 3.12 Strain result for N3, strain-gauge S1

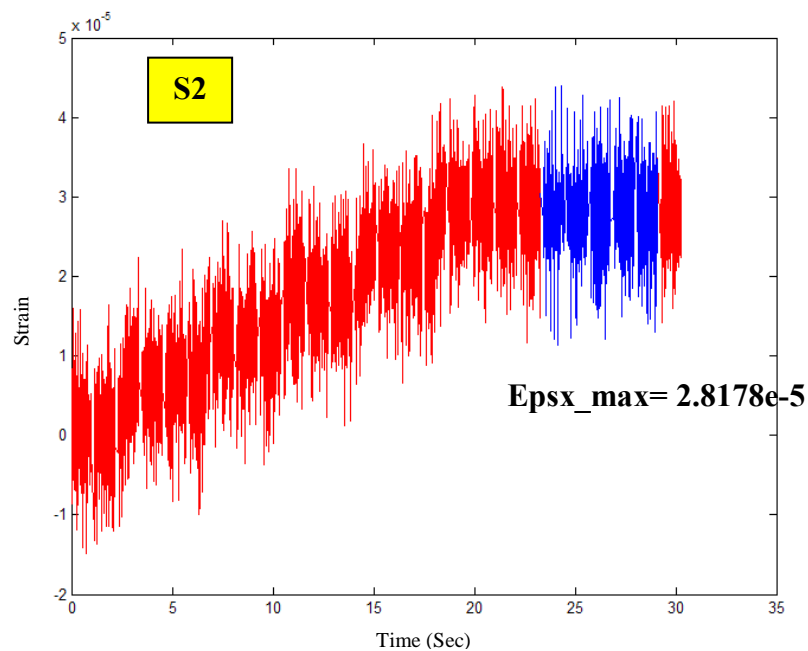


Figure 3.13 Strain result for N3, strain-gauge S2

3.2.4 Test Sample 4 (N4)

Displacement and Strain results for N4 are given in Figure 3.14, Figure 3.15 and in Figure 3.16.

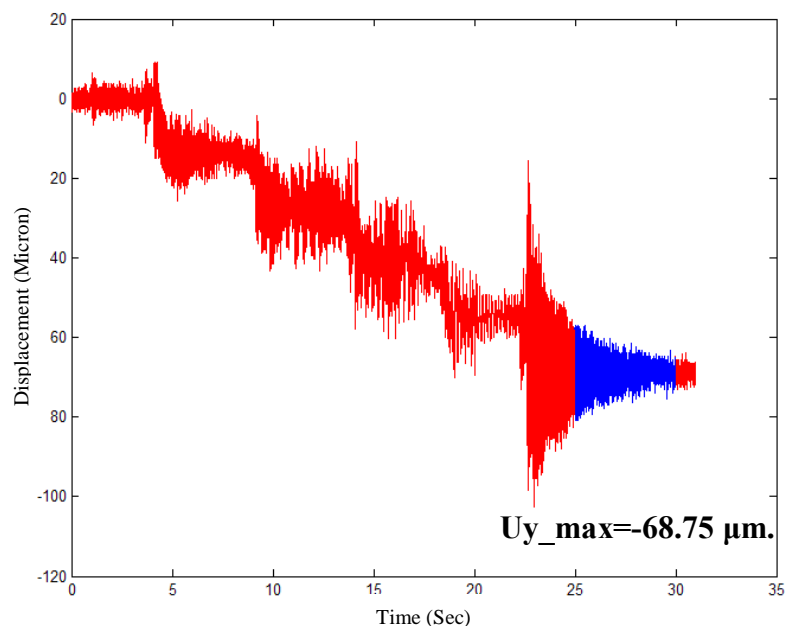


Figure 3.14 Displacement result for N4

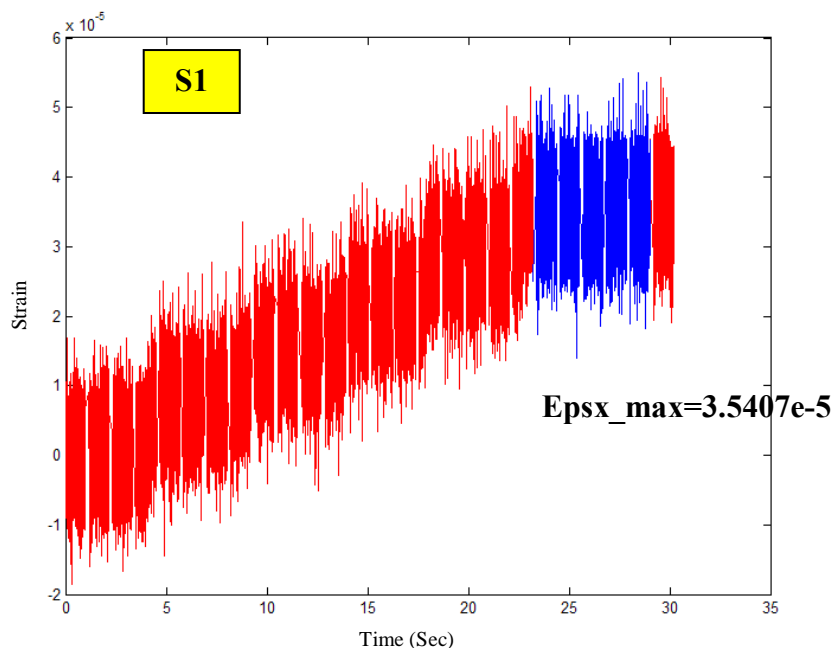


Figure 3.15 Strain result for N4, strain-gauge S1

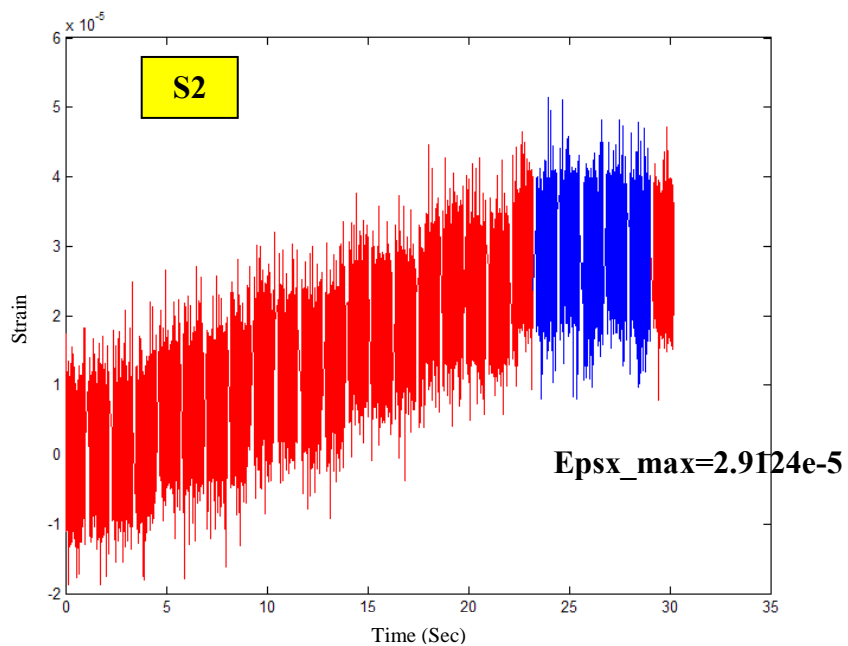


Figure 3.16 Strain result for N4, strain-gauge S2

3.2.5 Test Sample 5 (N5)

Displacement and Strain results for N5 are given in Figure 3.17, Figure 3.18 and in Figure 3.19.

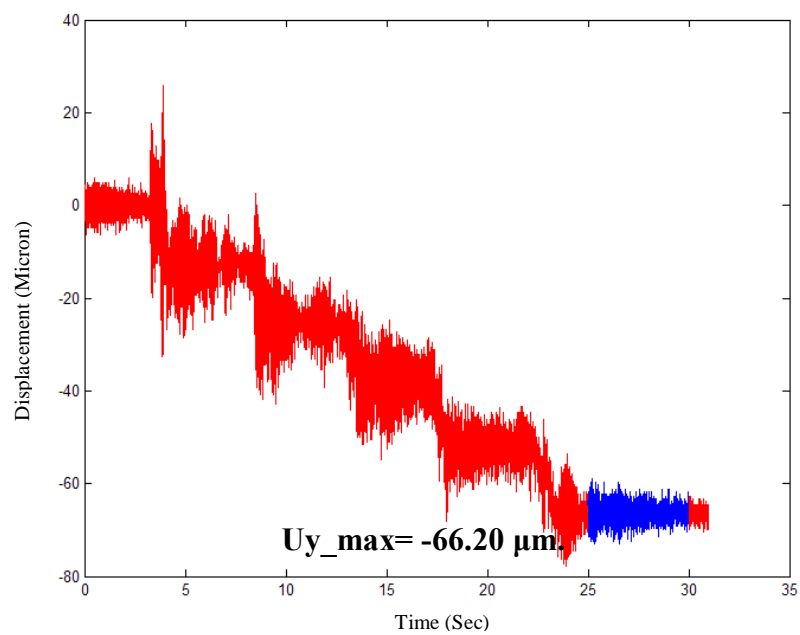


Figure 3.17 Displacement result for N5

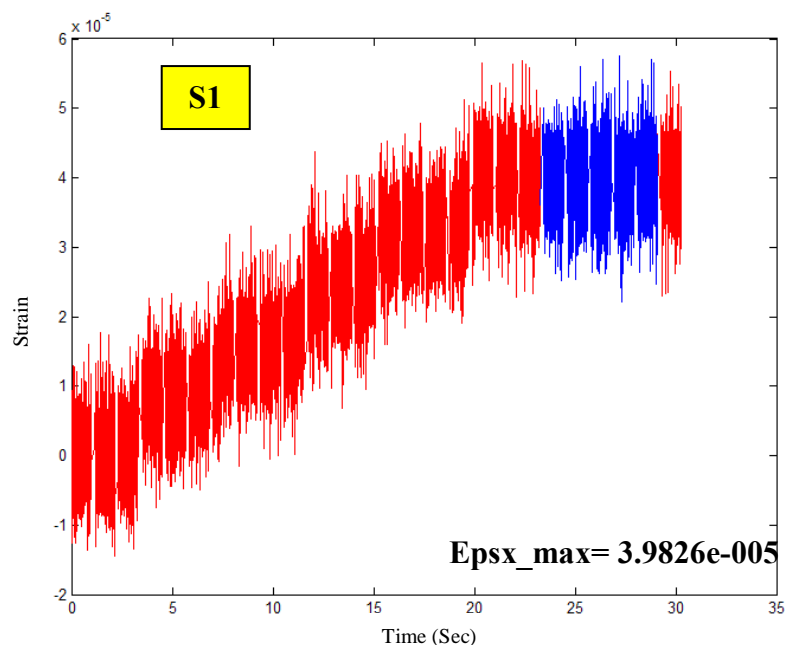


Figure 3.18 Strain result for N5, strain-gauge S1

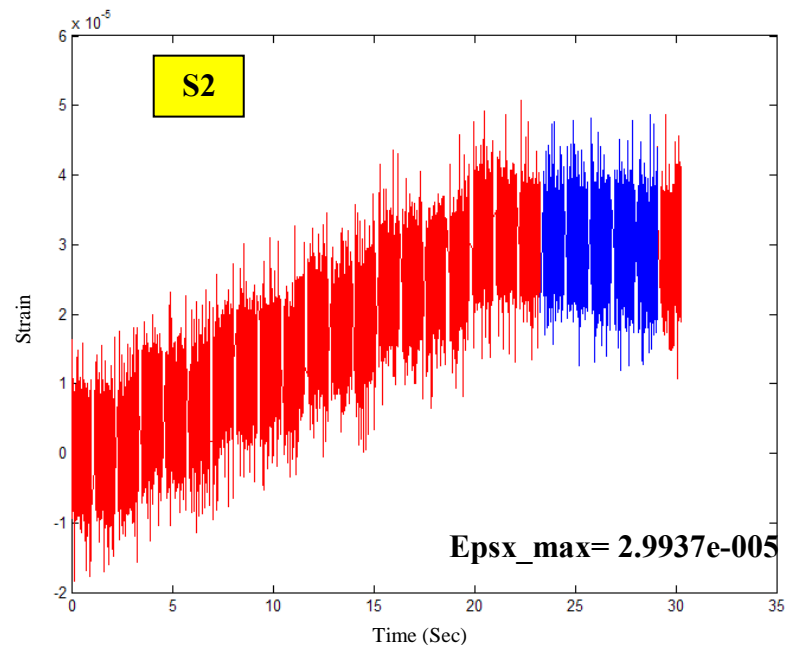
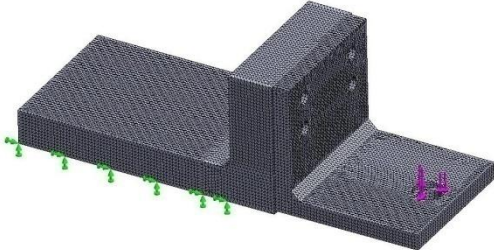
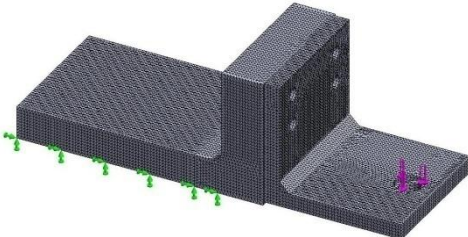
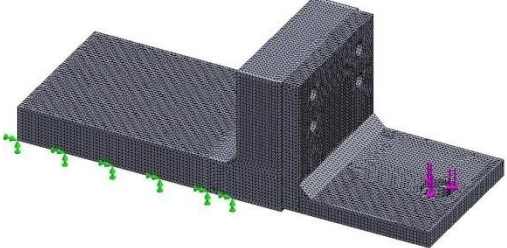
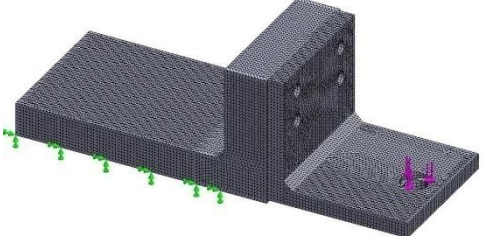
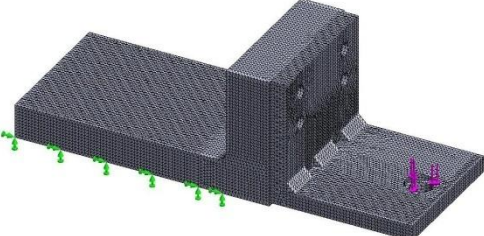


Figure 3.19 Strain result for N5, strain-gauge S2

3.3 Finite Element Analyses

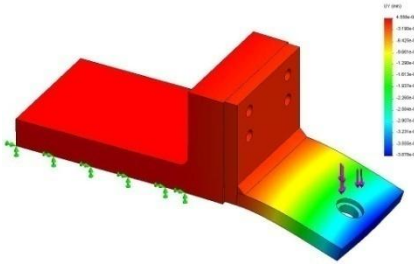
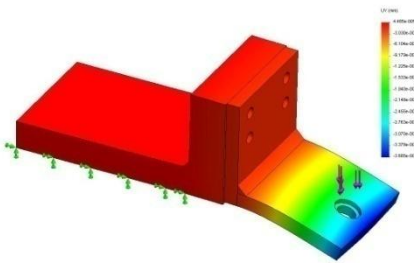
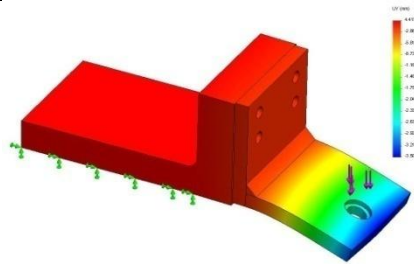
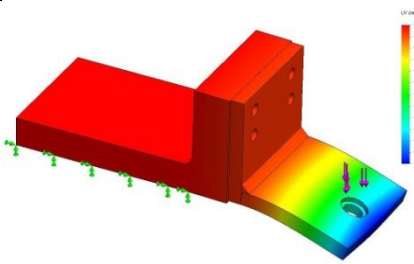
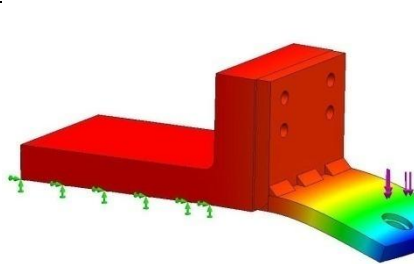
In order to compare the strain and displacement results obtained by the measurements of welded L test sample with numerical values, analyses have been performed with CosmosWorks Finite Element package. Parabolic triangular elements with 10 nodes, which have three degrees of freedom on each nodes, are used for finite element analyses. Finite element model is created by using 3 mm average element size. Total number of elements is approximately 470000 and total number of nodes is approximately 66500. Finite element models, strain, displacement and stress values belong to test samples are given in Table 3.1.

Table 3.1 Finite element models for test samples

N1	
N2	
N3	
N4	
N5	

The displacement results belong to test samples, which are obtained after finite element analyses, are given below in Table 3.2.

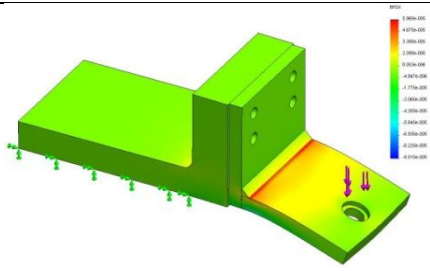
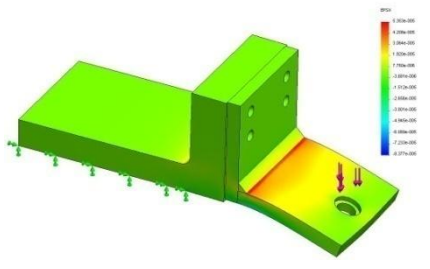
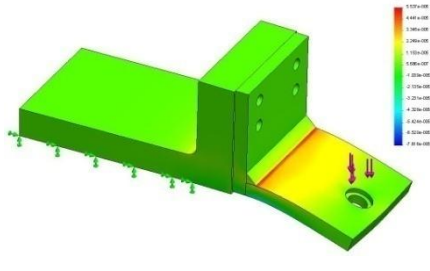
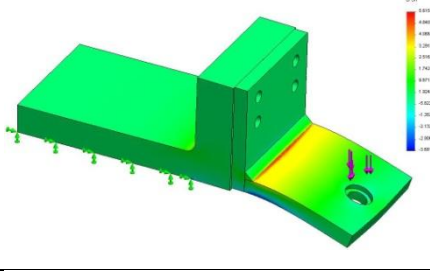
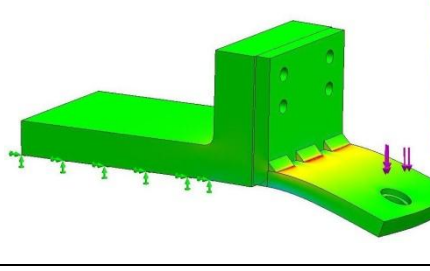
Table 3.2 Numeric displacement values belong to test samples

		Displacement
N1		(Test) Uy_max=-62.85 μm.
		(FE) Uy_max=-38.78 μm.
N2		(Test) Uy_max=-72.53 μm.
		(FE) Uy_max=-36.85 μm.
N3		(Test) Uy_max= -74.49 μm.
		(FE) Uy_max= -35.09 μm.
N4		(Test) Uy_max= -68.75 μm.
		(FE) Uy_max= -30.82 μm.
N5		(Test) Uy_max = -66.20 μm.
		(FE) Uy_max = -33.12 μm.

The strain results belong to test samples, which are obtained after finite element analyses, are given in Table 3.3. Values of S1 and S2 given in table below are

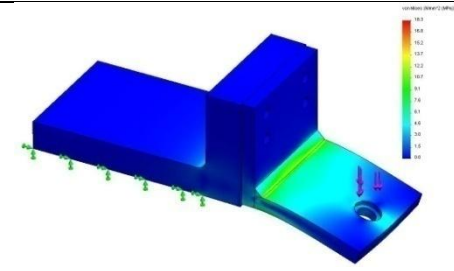
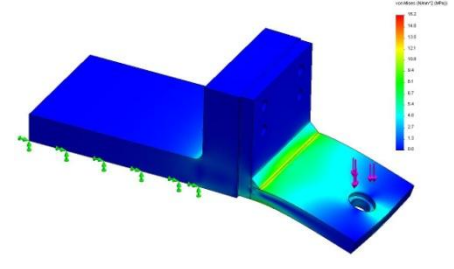
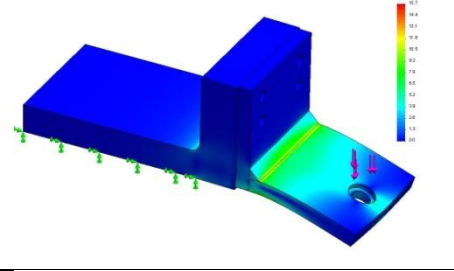
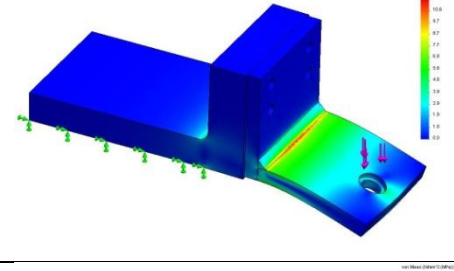
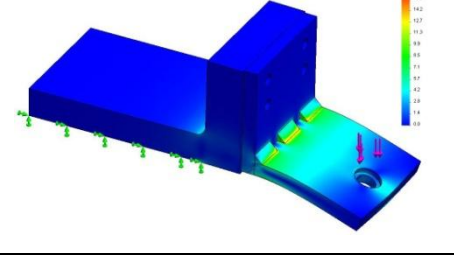
obtained by calculating the arithmetic mean of nodal strain values on surfaces covered by strain-gauges.

Table 3.3 Numeric Strain values belong to test samples

		S1	S2
N1		(Test) $S_{Xort}=3.65e-5$	(Test) $S_{Xort}=3.36e-5$
		(FE) $S_{Xort}=3.04e-5$	$S_{Xort}=3.64e-5$
N2		(Test) $S_{Xort}=3.04e-5$	(Test) $S_{Xort}=2.91e-5$
		(FE) $S_{Xort}=3.13e-5$	$S_{Xort}=3.61e-5$
N3		(Test) $S_{Xort}=3.03e-5$	(Test) $S_{Xort}=2.82e-5$
		(FE) $S_{Xort}=3.30e-5$	$S_{Xort}=3.61e-5$
N4		(Test) $S_{Xort}=3.54e-5$	(Test) $S_{Xort}=2.91e-5$
		(FE) $S_{Xort}=3.44e-5$	$S_{Xort}=3.16e-5$
N5		(Test) $S_{Xort}=3.98e-5$	(Test) $S_{Xort}=2.99e-5$
		(FE) $S_{Xort}=3.64e-5$	$S_{Xort}=3.49e-5$

The stress results belong to test samples, which are obtained after finite element analyses, are given in Table 3.4.

Table 3.4 Numeric stress values belong to test samples

		von Mises Stress (MPa)
N1		$\sigma_{eq} = 18.3$
N2		$\sigma_{eq} = 16.2$
N3		$\sigma_{eq} = 15.7$
N4		$\sigma_{eq} = 11.6$
N5		$\sigma_{eq} = 17.0$

In case numeric results and experimental results are compared, apparent difference between displacement values are observed. The difference can be explained with mounting the test samples to the plate beyond with bolts and different static behaviours of test plates related to uncontrolled tightening torques applied to the bolts with hand. On the other hand the accordance between numeric and experimental strain values attracts the attention. When the stress values are controlled, maximum stress value occurring on plate is lowered in case the welding seam size is increased. It is observed that the optimum stress condition is obtained when welding seams are applied small sized and double sided. Increase in stress values are observed when the welding seams are applied partially double-sided (method).

CHAPTER FOUR

NUMERIC ANALYSES AND EXPERIMENTAL STUDIES FOR PRESS BODY

In the scope of thesis experimental and numerical studies have been done for the automatically modeled press body. In experimental studies, four laser displacement sensor and strain measurement system, which are obtained in the scope of thesis, have been used. Keyence Laser displacement sensors, which have different measurement range and different sensitivities, are used. Two unit laser displacement controllers are used for four measurement unit. In experimental studies multiple channel strain measurements are followed through with system which has seven strain measurement modules. Schematic image belong to test rig and views from Dirinler C press strain and displacement measurement set is shown in Figure 4.1 and in Figure 4.2.

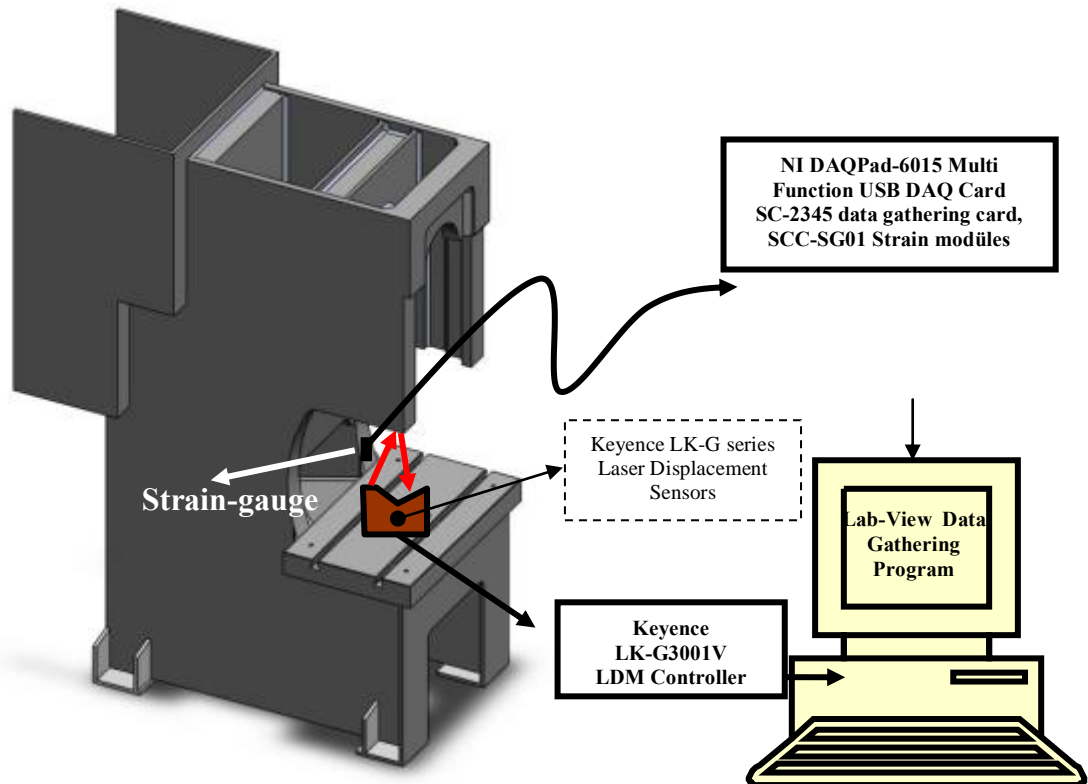


Figure 4. 1 Dirinler C press strain and displacement measuring set up (schematic)



Figure 4.2 Views from Dirinler C press strain and displacement measurement set

Within the scope of strain measurements, strain gauges are located on ten different points on the press body. The points where strain gauges are located are shown in Figure 4.3, Figure 4.4 and Figure 4.5.

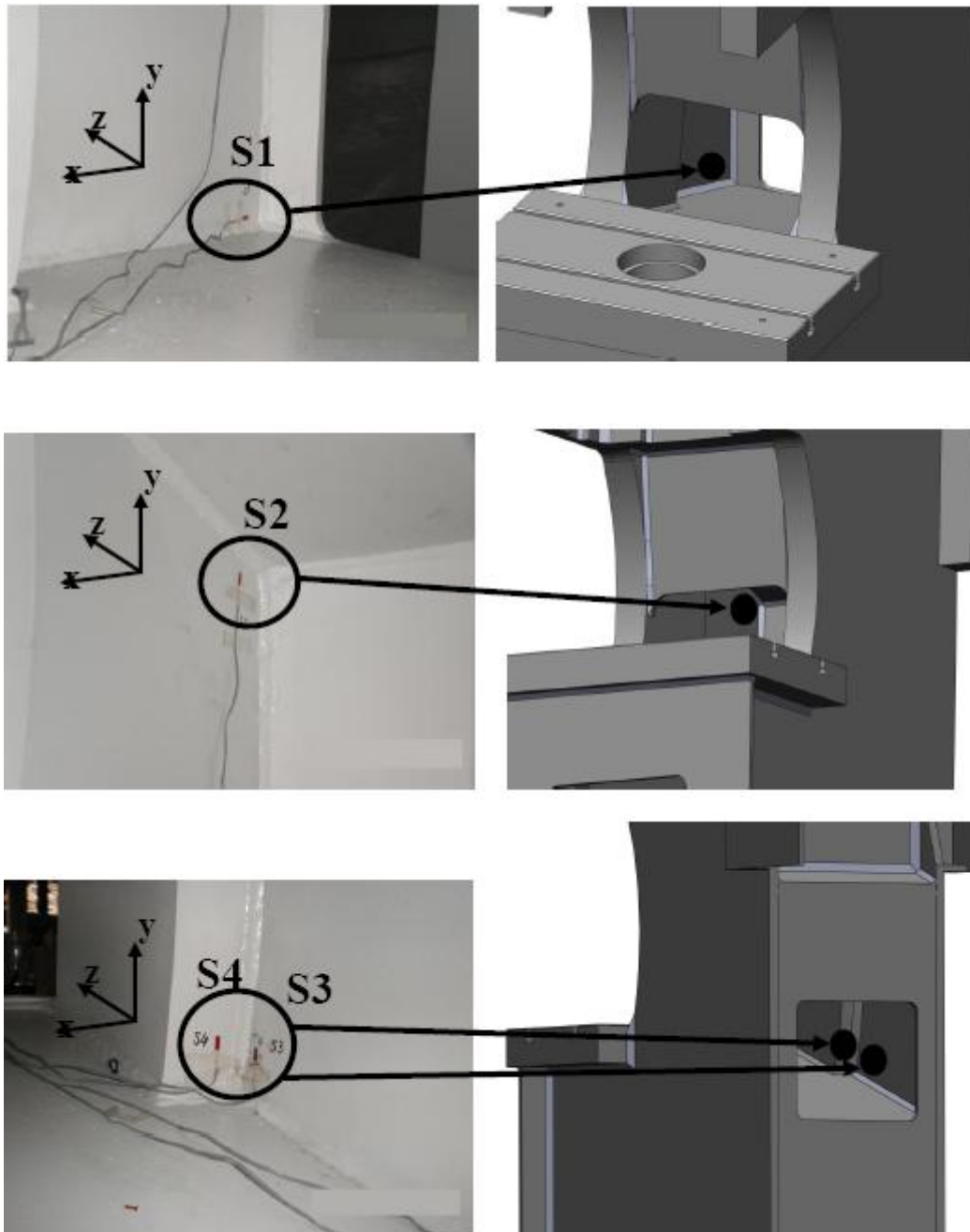


Figure 4.3 Strain measurement regions

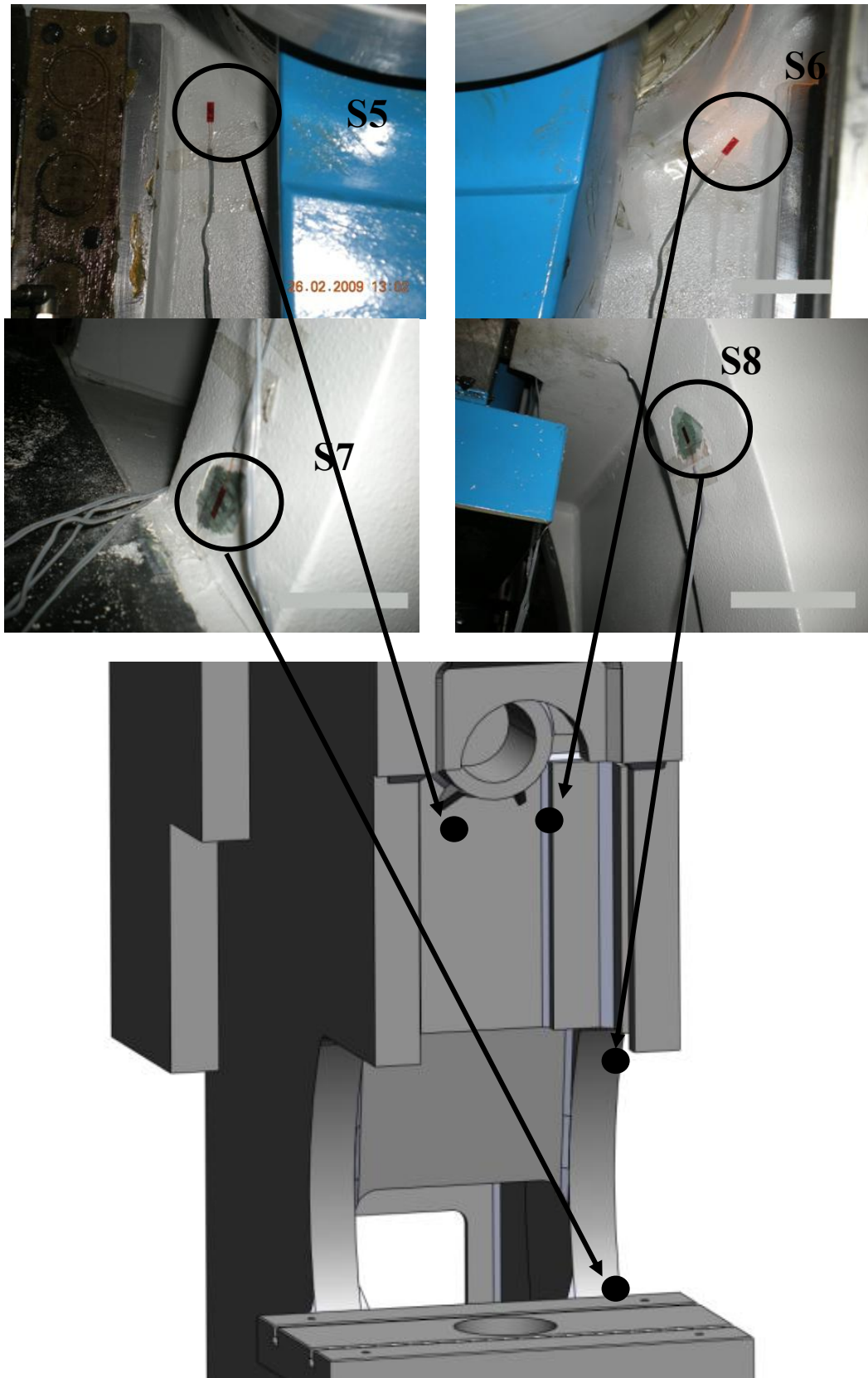


Figure 4.4 Strain measurement regions

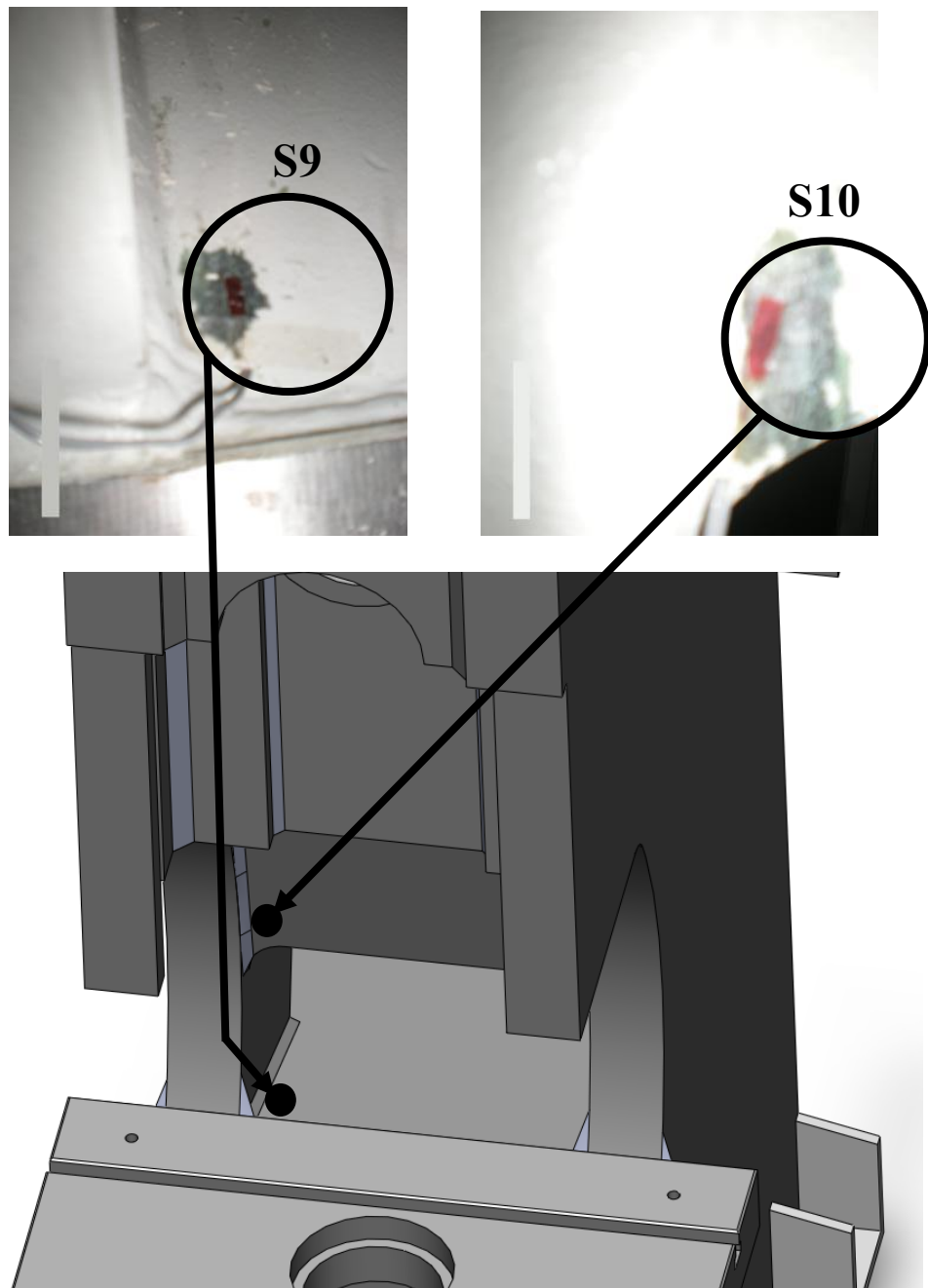


Figure 4.5 Strain measurement regions

4.1 Strain Measurement Results

During the loading, measurements have been carried out on C type press body with 80 tones capacity. There is not any indicator in order to show current load during the loading. The amount of loading is roughly determined by mechanic fuse in the measured press body.

Loading process is performed for proximity condition in which the mechanic fuse designed for 80 tones capacity is closest to fraction limit. Strain responses for the points shown above are shown in the figures from Figure 4.6 to Figure 4.15.

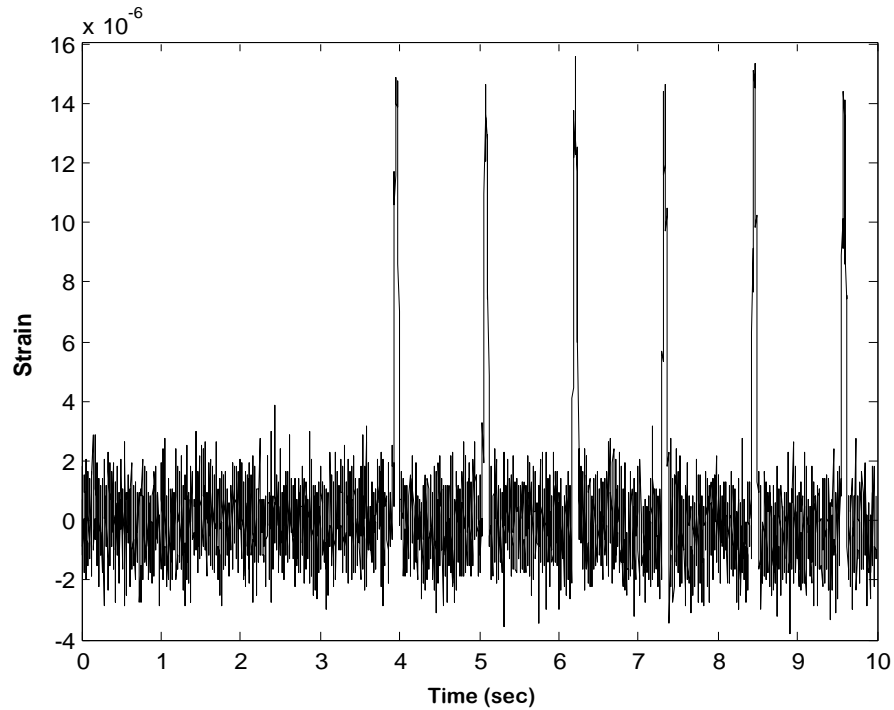


Figure 4.6 Strain response for the point S1

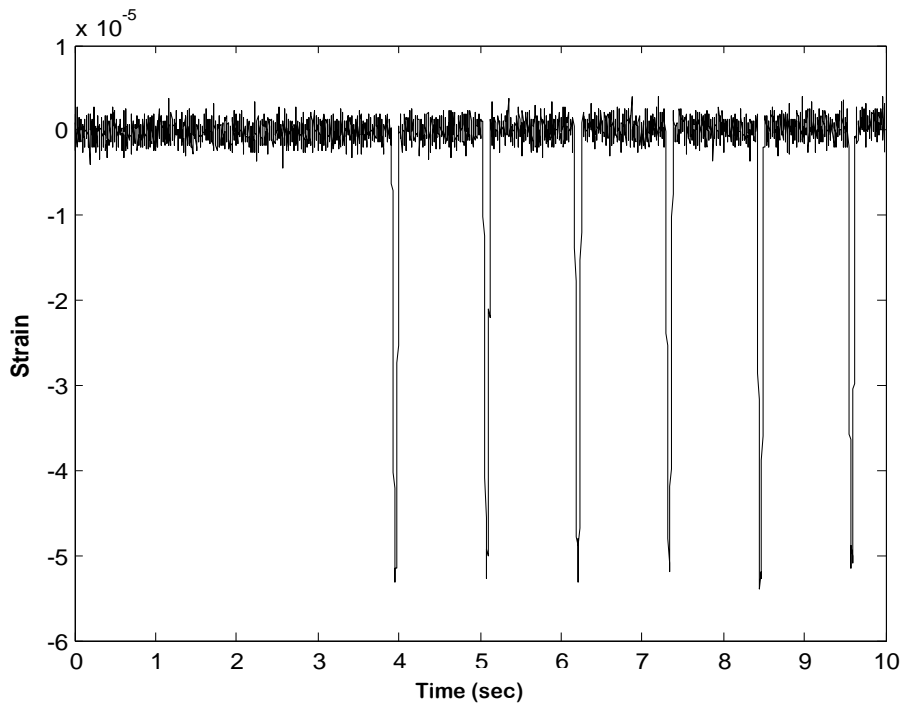


Figure 4.7 Strain response for the point S2

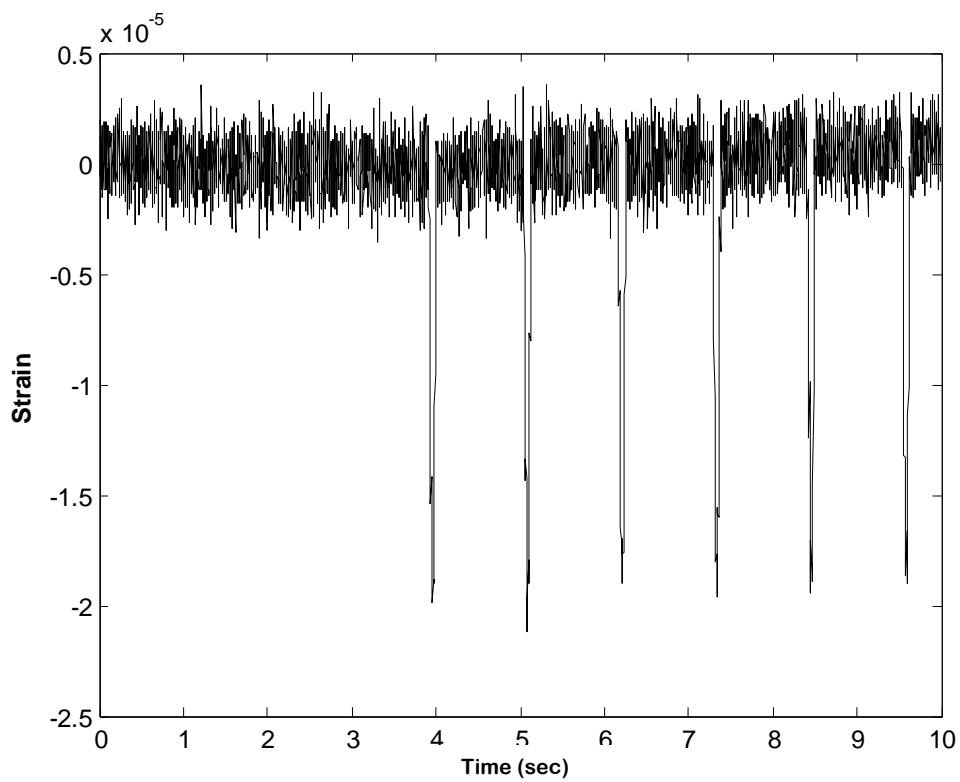


Figure 4.8 Strain response for the point S3

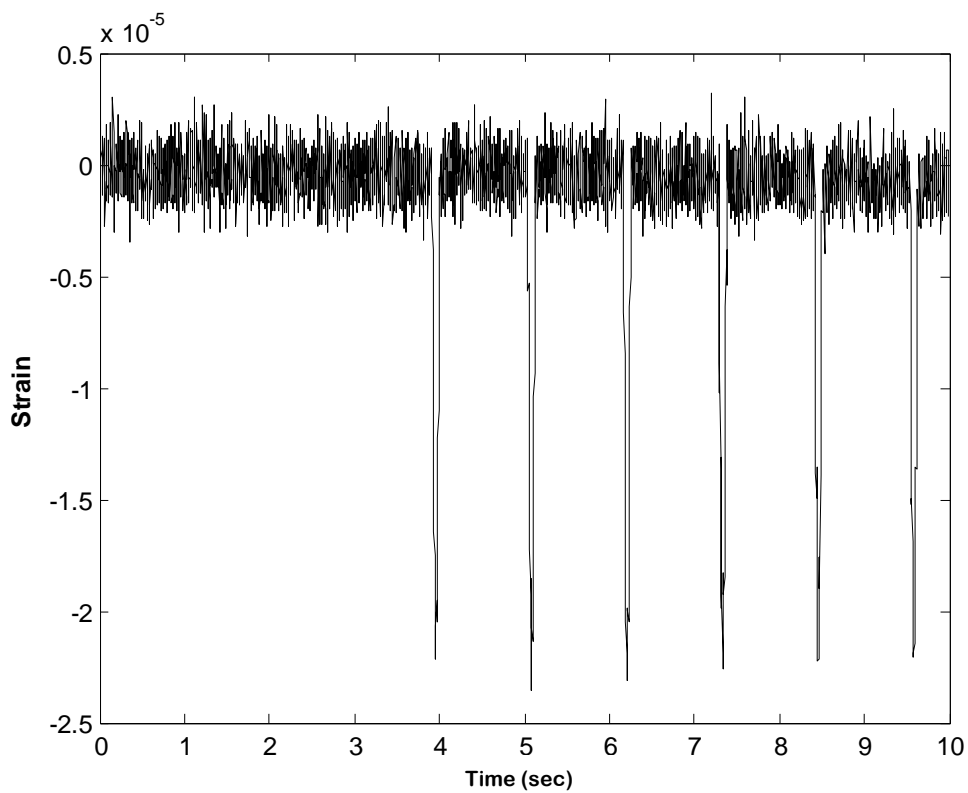


Figure 4.9 Strain response for the point S4

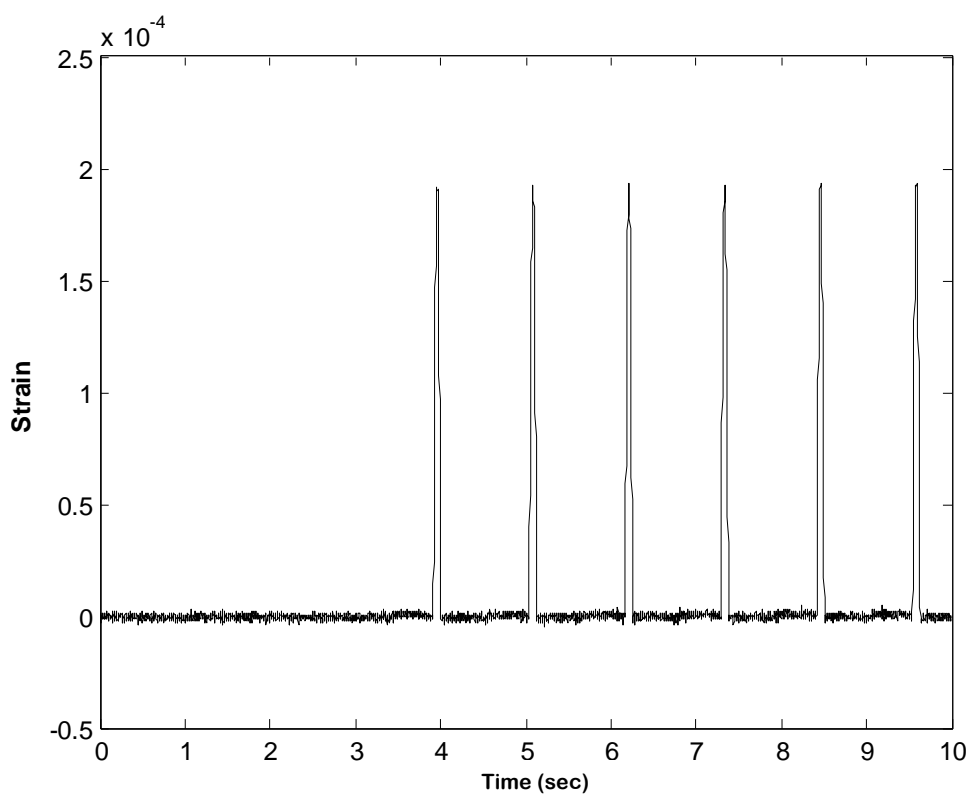


Figure 4.10 Strain response for the point S5

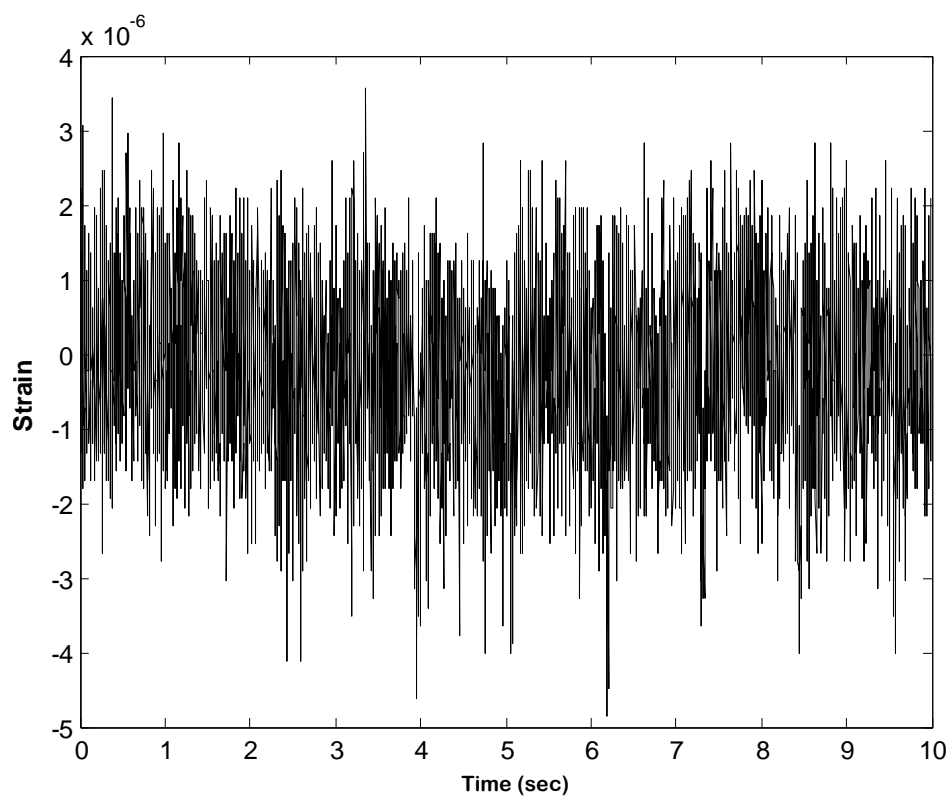


Figure 4.11 Strain response for the point S6

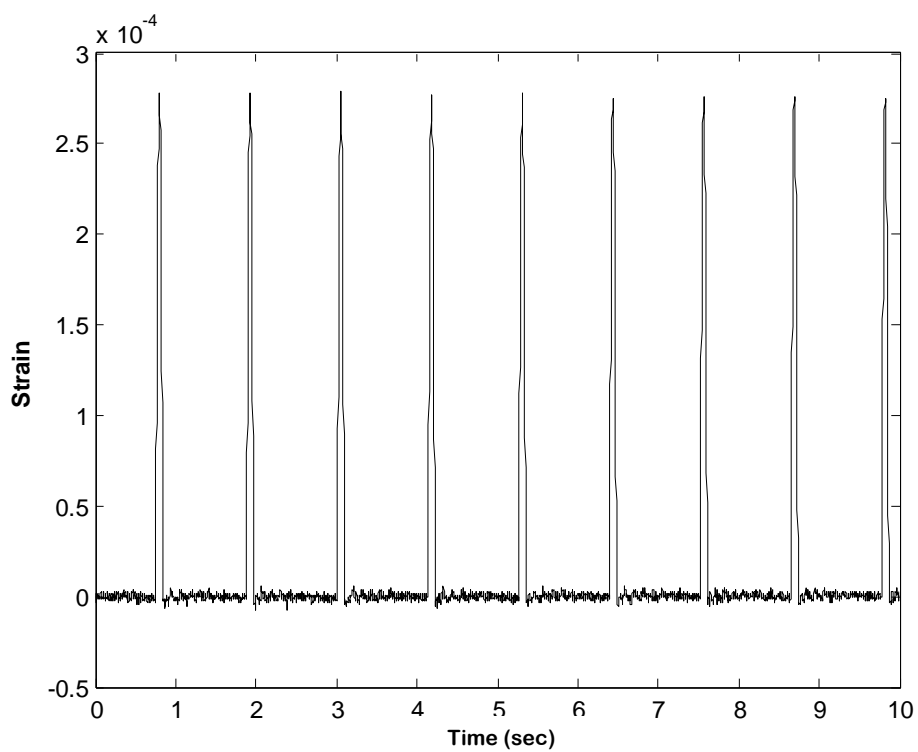


Figure 4.12 Strain response for the point S7

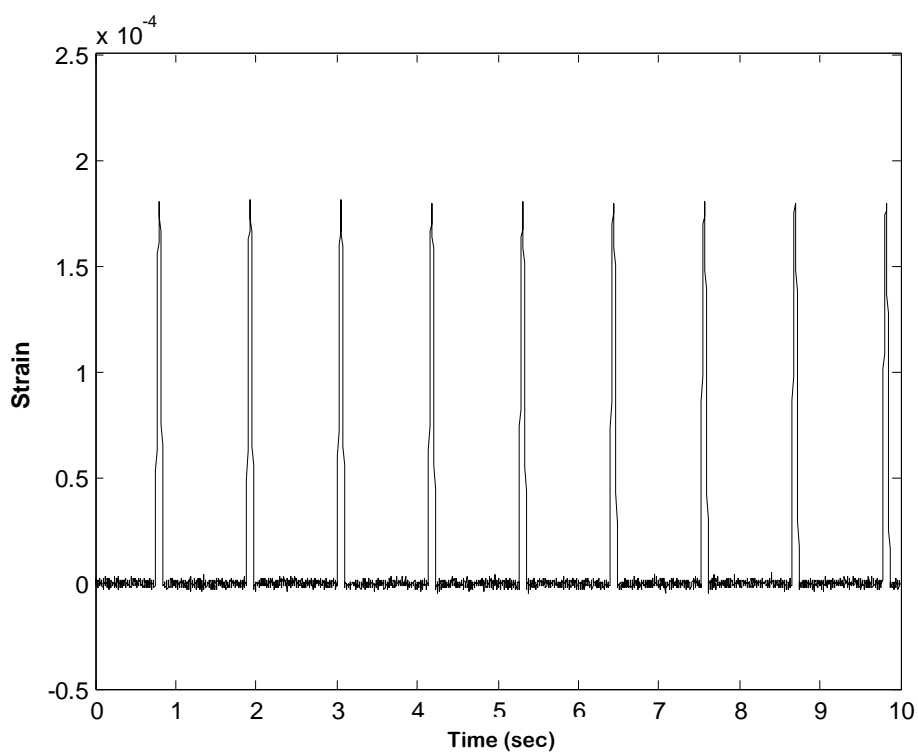


Figure 4.13 Strain response for the point S8

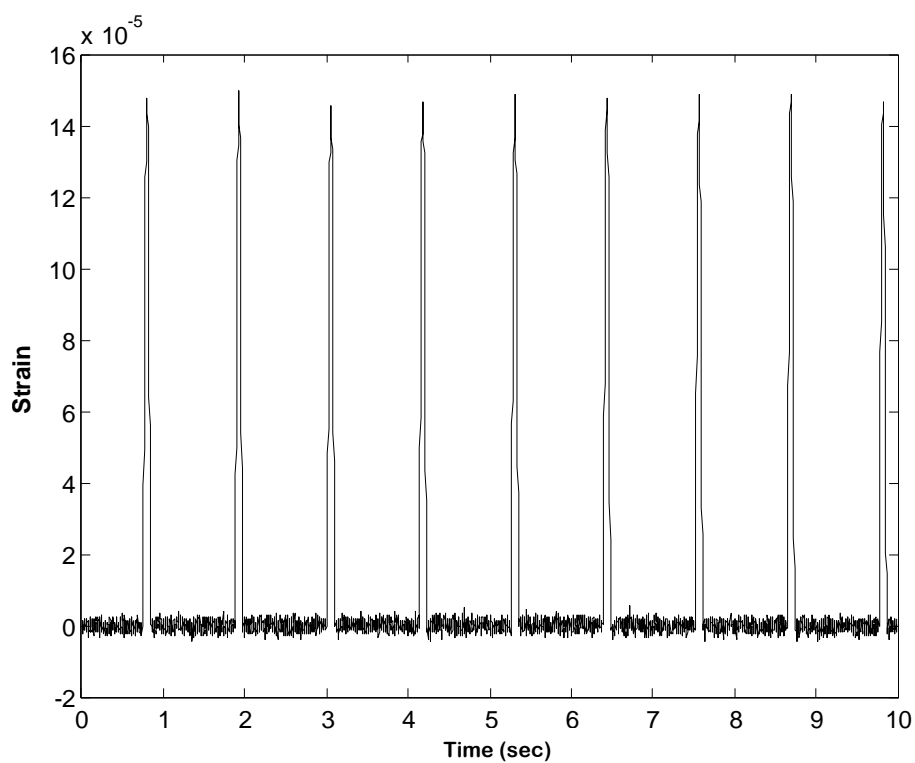


Figure 4.14 Strain response for the point S9

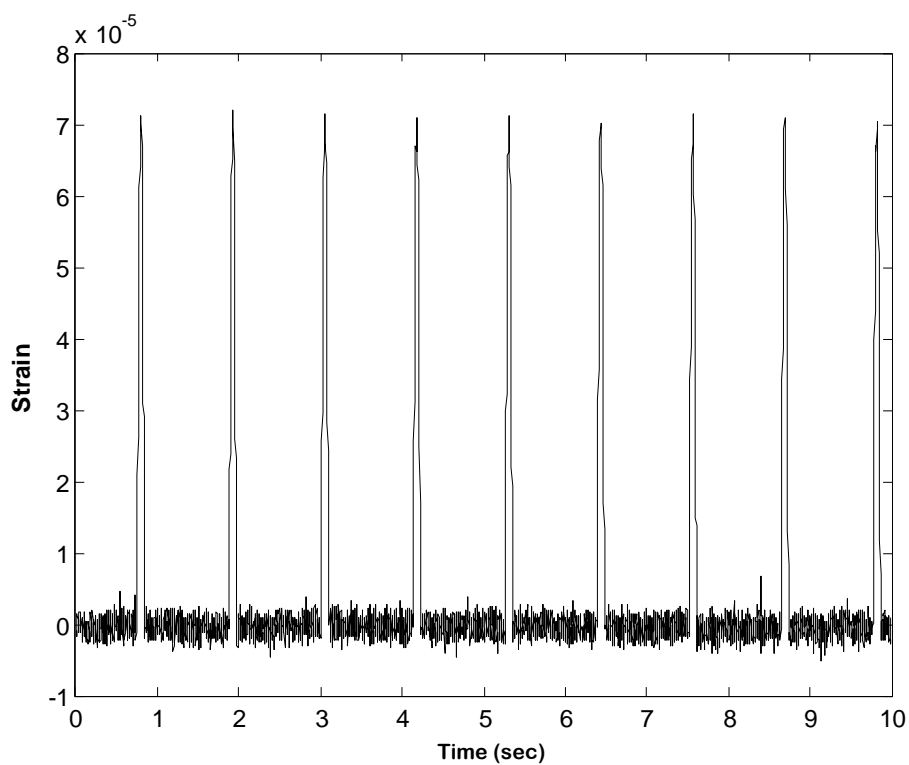


Figure 4.15 Strain response for the point S10

Strain values on the points S5, S7, S8 and S9 are higher than the other points as it is seen from strain measurement results. Strain direction for S1 and S9 is on X axis, for the other points strain direction is on Y axis.

4.2 Numeric Analyses

Static finite element analyses for C type press body are performed with CosmosWorks program. The finite element model belong to analysed C type press body is given in Figure 4.16. Parabolic triangular elements with 10 nodes, which have three degrees of freedom on each node, are used on press body finite element model. Average element and node numbers used in analyses are given in Figure 4.16.

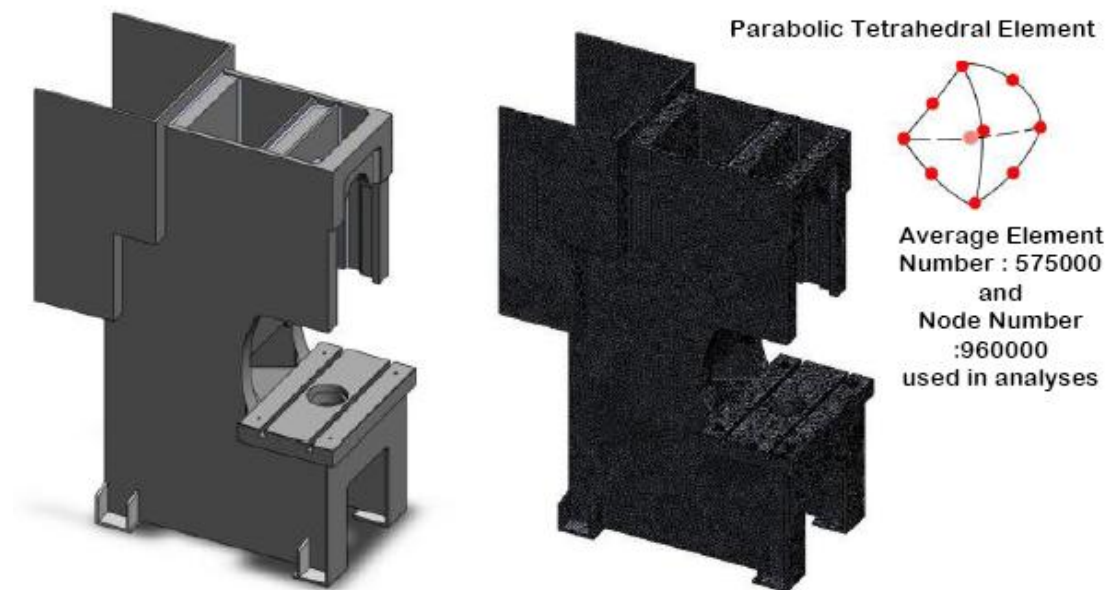


Figure 4.16 Solid and Finite element model for C type press body with 80 tones capacity

In static analyses press body is fixed from foots locating on bottom of press body. Press body is loaded with bearing load from main bearing and the rear main bearing as shown in Figure 4.17. Besides, the press body is vertically loaded on work table with 80 tones. Under mentioned condition static analyses are perform for press body. Bearing load on the model is shown in Figure 4.17 and Figure 4.18.

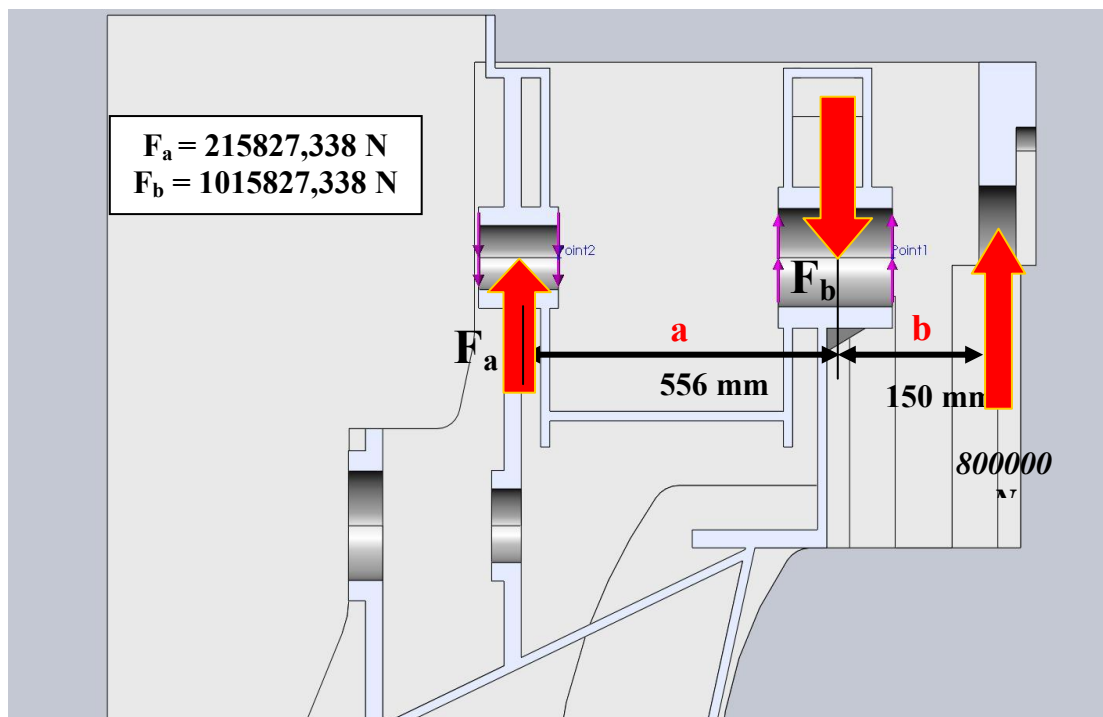


Figure 4.17 Press body bearing load

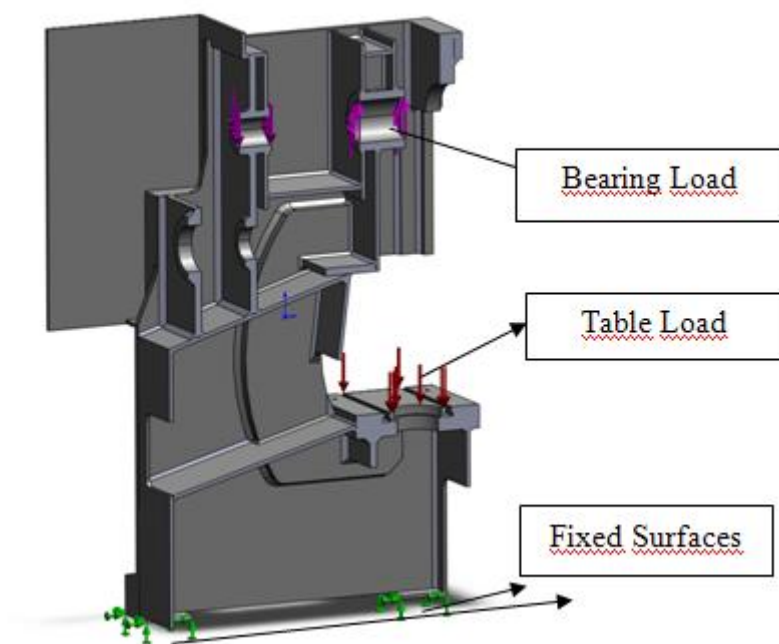


Figure 4.18 Press body boundary conditions (Half Sectioned Model)

Stress and Strain responses for 80 tones capacity press body are given in Figure 4.19 and Figure 4.20.

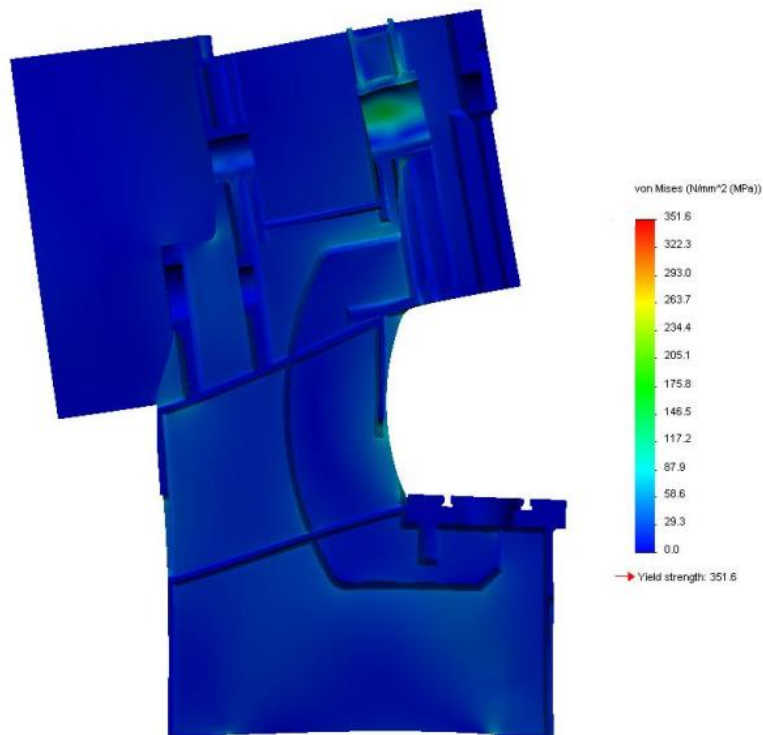


Figure 4.19 C type press body with 80 tones capacity equivalent stress response

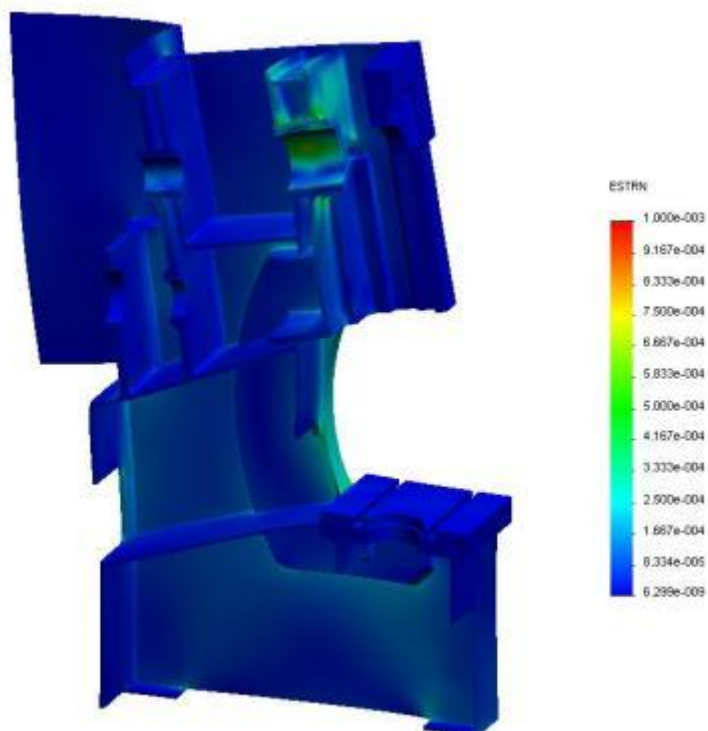


Figure 4.20 C type press body with 80 tones capacity equivalent strain response

Strain responses in X and Y Axis are given in Figure 4.21 and Figure 4.22.

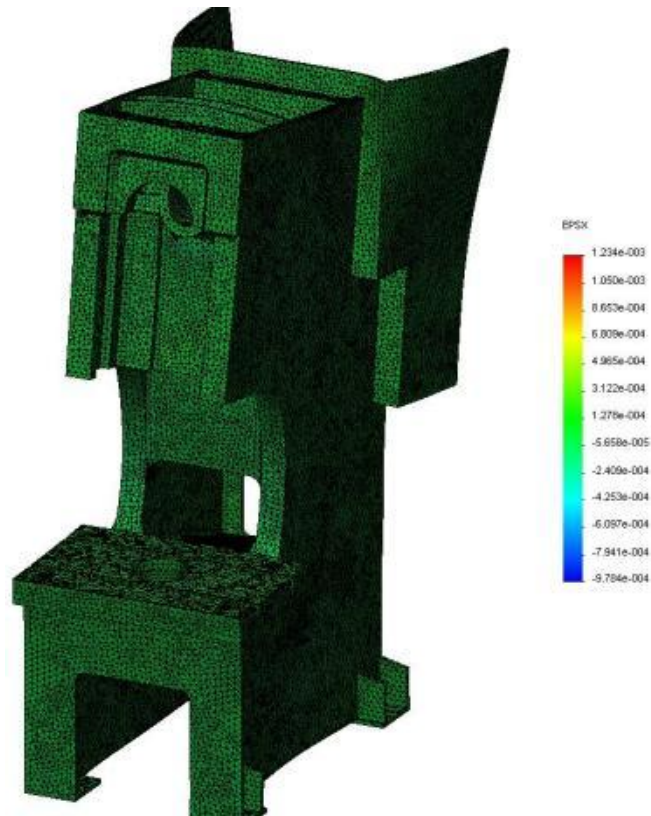


Figure 4.21 Press body X axis Strain response

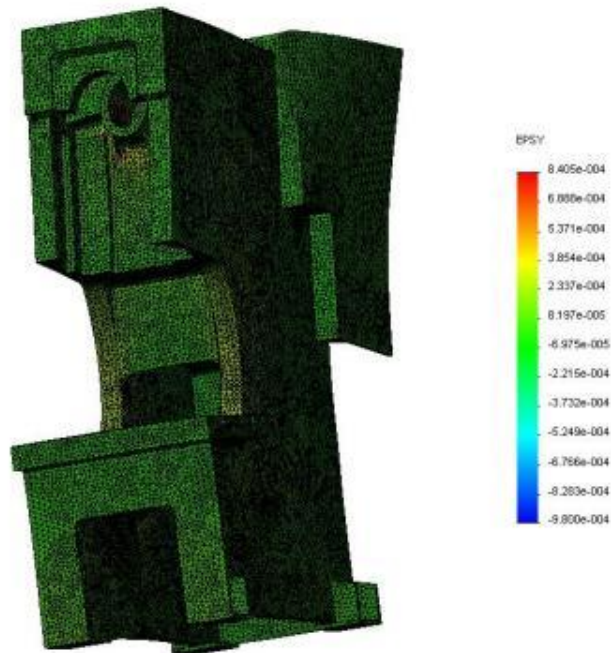


Figure 4.22 Press body Y axis Strain response

Numerical and experimental results, obtained from finite element analyses for press body, are compared in Table 4.1. As shown from the results, it is observed that the numeric results, obtained for complex structure as press body, are well qualified to give an idea about strain values on the structure. Especially the matches for points S7 and S8 are satisfactory. Strain values for all points except S3 are compatible as a whole.

Table 4.1. Experimental and numeric (FE) strain responses.

	Strain (Exp.-Max)	Strain (Numeric-Avg.)
S1 (S_x)	1.559e-5	2.729e-5
S2 (S_y)	-5.405e-5	-1.879e-4
S3 (S_y)	-2.122e-5	1.575e-5
S4 (S_y)	-2.354e-5	-5.855e-5
S5 (S_y)	1.936e-4	3.736e-4
S6 (S_y)	-	-
S7 (S_y)	2.788e-4	2.826e-4
S8 (S_y)	1.810e-4	2.079e-4
S9 (S_x)	1.495e-4	5.081e-5
S10 (S_y)	7.200e-5	1.393e-4

4.3 Press Displacement Response

Displacement responses are obtained by the help of laser displacement sensors, which are purchased in the scope of thesis. Displacement measurements for press body with 80 tones capacity are taken under normal loading conditions. Sampling frequency is used as 1 kHz in laser displacement measurements. Displacement measurement set up is shown in Figure 4.23 schematically. The displacement measurements are performed for two points A and B at which the maximum displacement responses are expected.

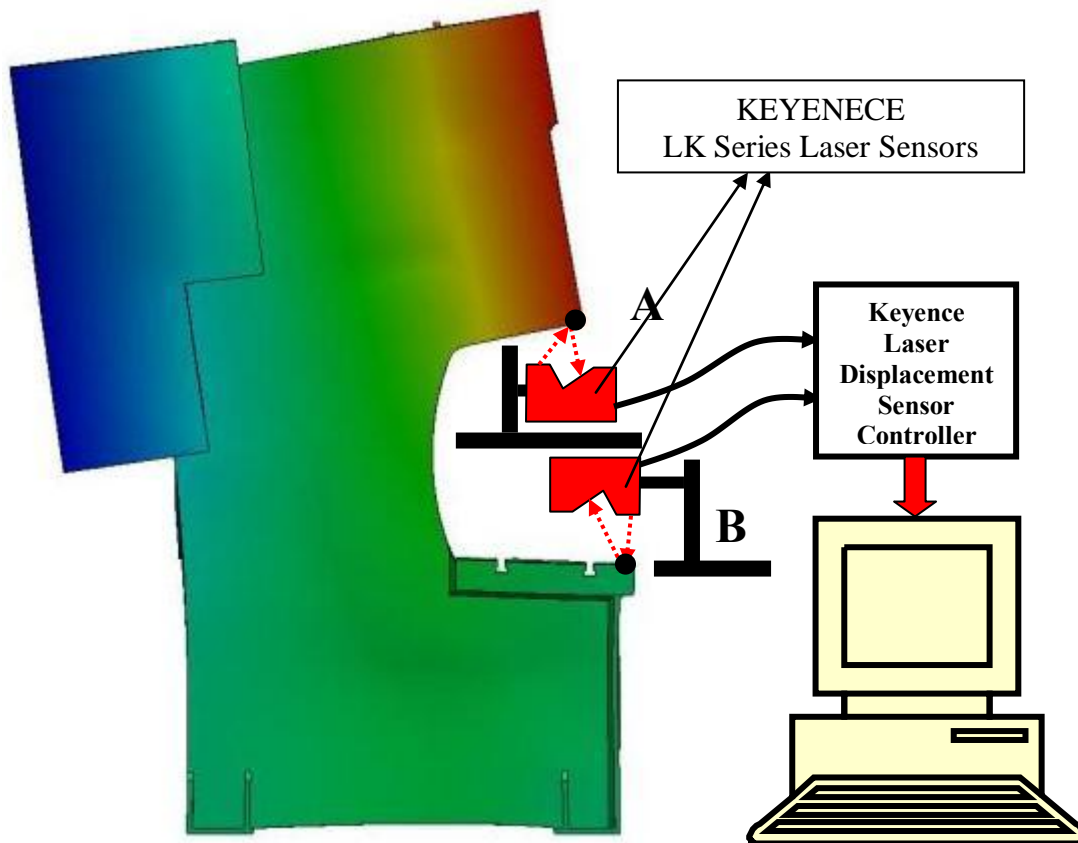


Figure 4.23 Displacement measurement system with laser sensors

Displacement response for point A when the press under load is given in Figure 4.24.

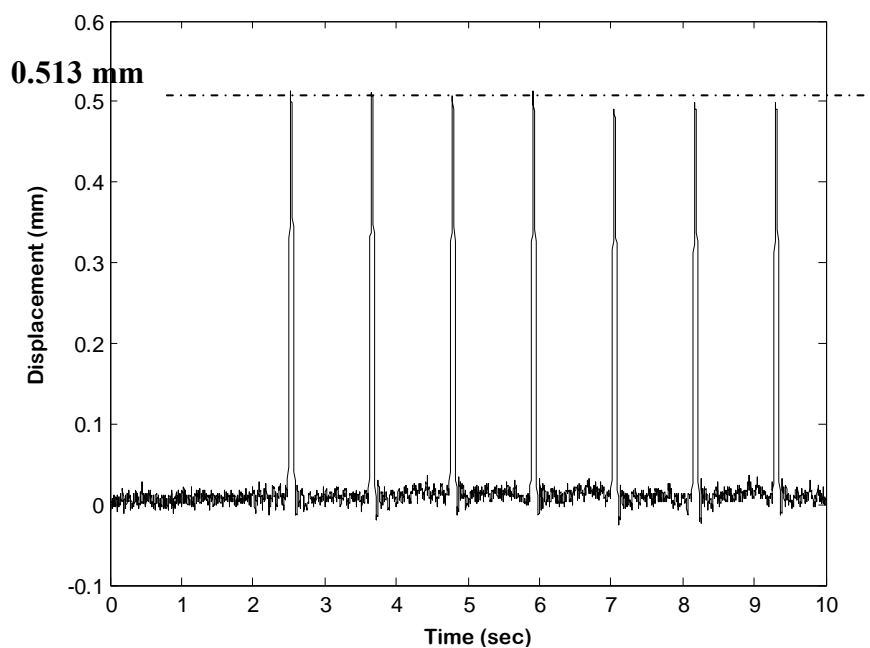


Figure 4.24 Displacement response for point A on Press body

Displacement response for point B is given in Figure 4.25. As it is seen from the response the amount of displacement for "Alt plaka" is 0.1328 mm. Frequency content which belong to displacement response on point B is given in Figure 4.26.

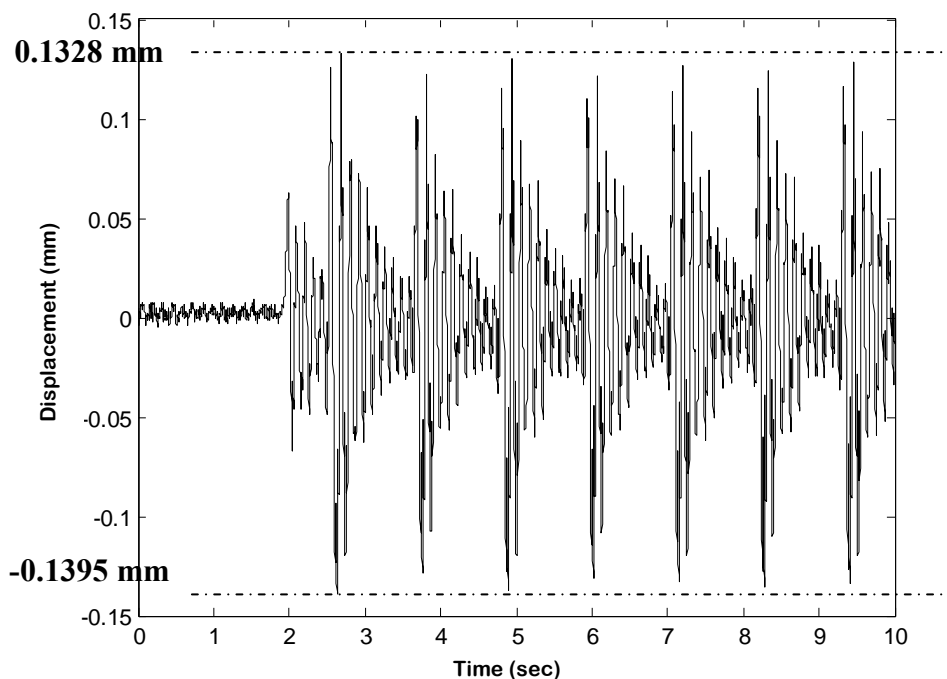


Figure 4.25 Displacement response for point B on Press body

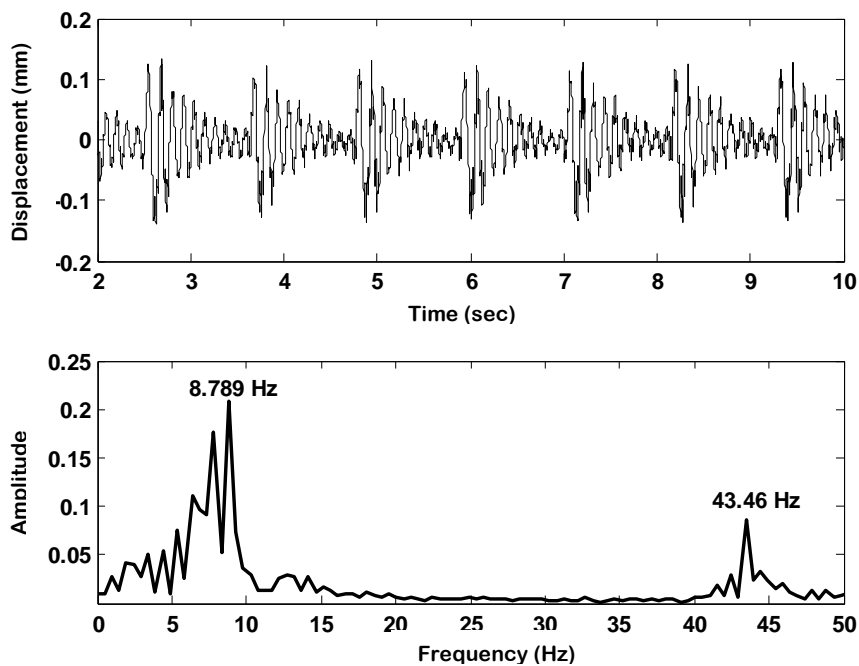


Figure 4.26 Displacement response and frequency content for point B

Two natural frequency value belong to structure are significant in Figure 4.26. These natural frequencies are frequencies which occur with effects of significant weights like crank, flywheel and the ram. Displacement response belong to press body is given in Figure 4.27.

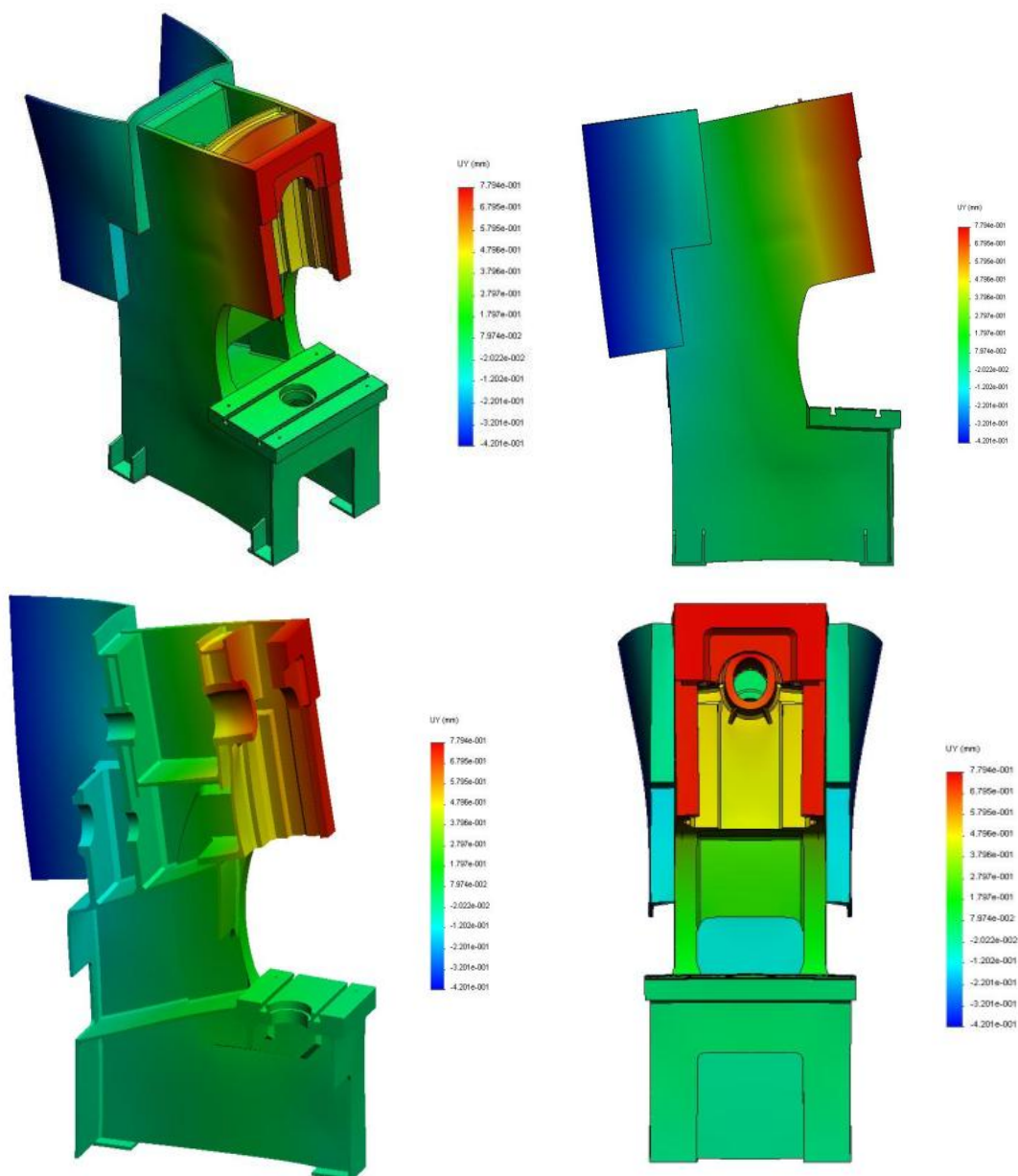


Figure 4.27 Displacement response of C type Press body with 80 tones capacity

The regions for which the displacement results are plotted are seen in Figure 4.28.

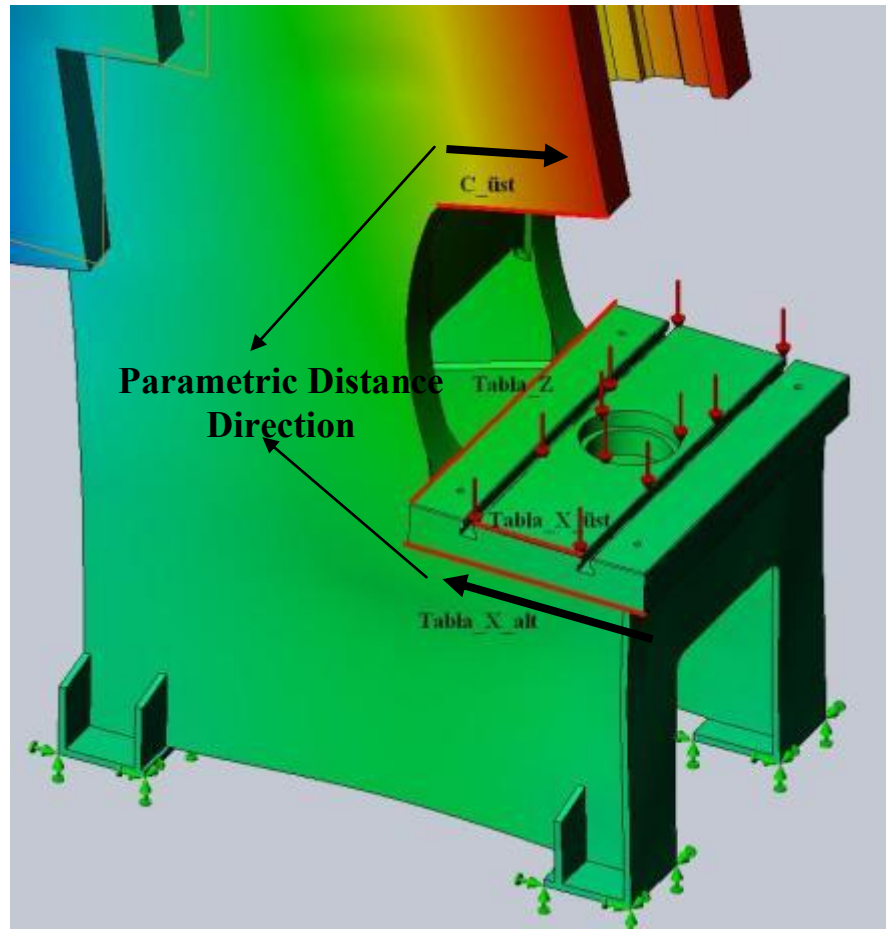


Figure 4.28 Edges used for press displacement analysis results

Displacement results for press "C_üst" edge are given in Figure 4.29. As it is seen from the figure, maximum numeric displacement value obtained on "C_üst" edge is 0.740 mm. Maximum displacement value on the same region obtained from the tests is 0.513 mm. Experimental and numeric results are quite well-matched. Besides, crank mass, flywheel mass and ram mass are not considered in numeric analyses. Because of these weights are in direction of gravity, displacements on point A are forced to decrease. Experimental displacement value on table point B is 0.104 mm and numeric displacement value which is obtained for the same point is 0.104 mm. As it seems experimental and numeric displacement values are quite similar.

Displacement responses and finite element analyses obtained on "C_üst" edge and "Tabla_X_alt" edge of press body, are given in Figure 4.30

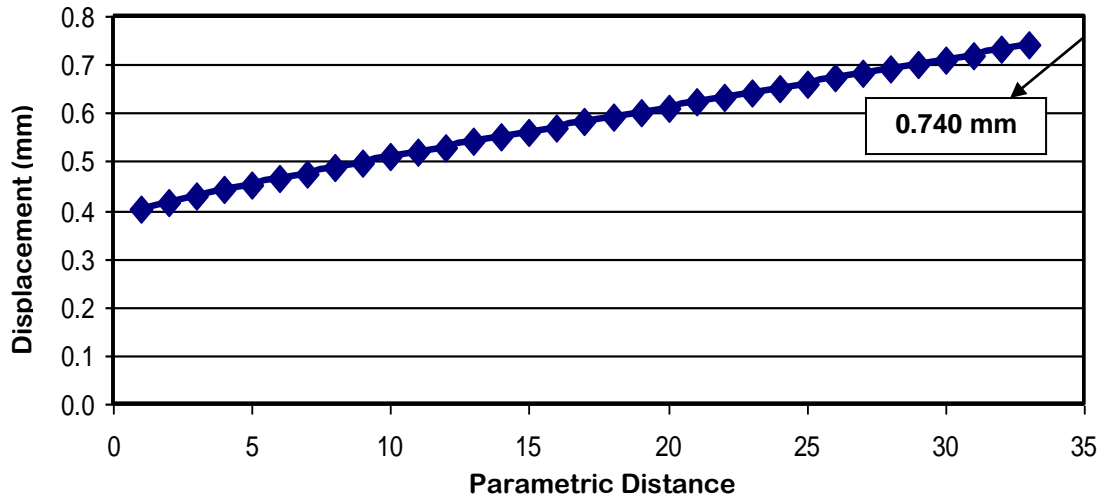


Figure 4.29 Numeric displacement response for the edge "C_üst"

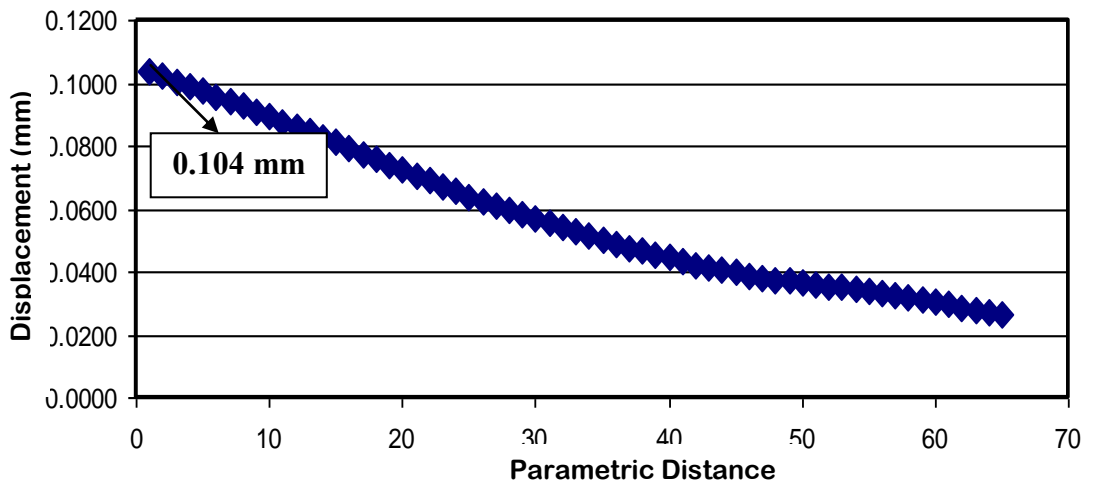


Figure 4.30 Numeric displacement response for the sub edge "Tabla_X"

4.4 Press Body Natural Frequency Analysis

Natural frequency value is an important parameter which expresses the stiffness of the structure. Natural frequency analysis has been done in order to analysis the effect of welding seam size on structural stiffness. Natural frequency values belong to free press body without welding, are given in Figure 4.31.

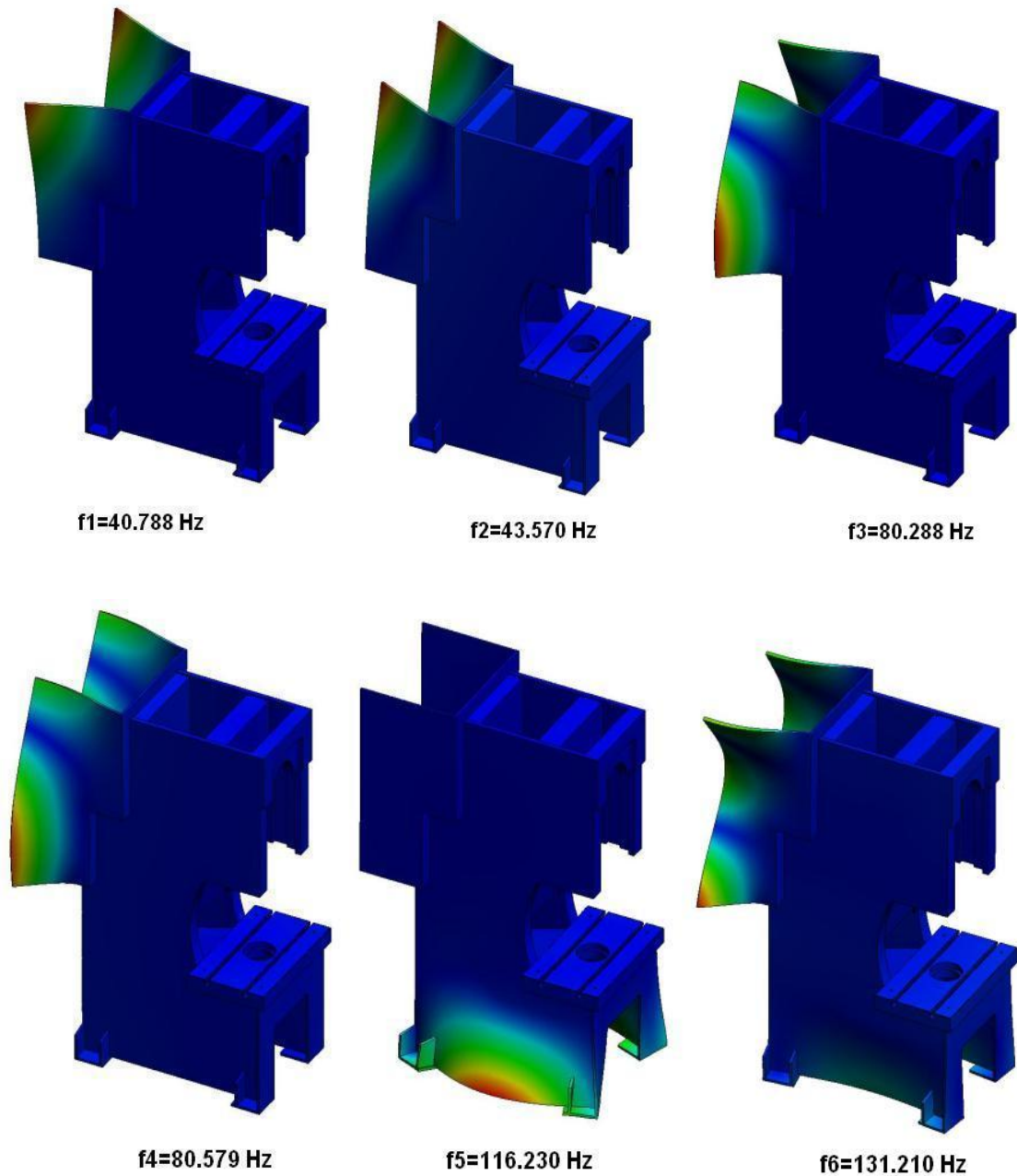


Figure 4.31 Natural frequency values belong to free press body without welding with 80 tones capacity (Free)

Natural frequency values for bottom fixed and weldless press body with 80 tones capacity are given in Figure 4.32. As it is understood from natural frequency values, boundary conditions have significant importance on natural frequency values.

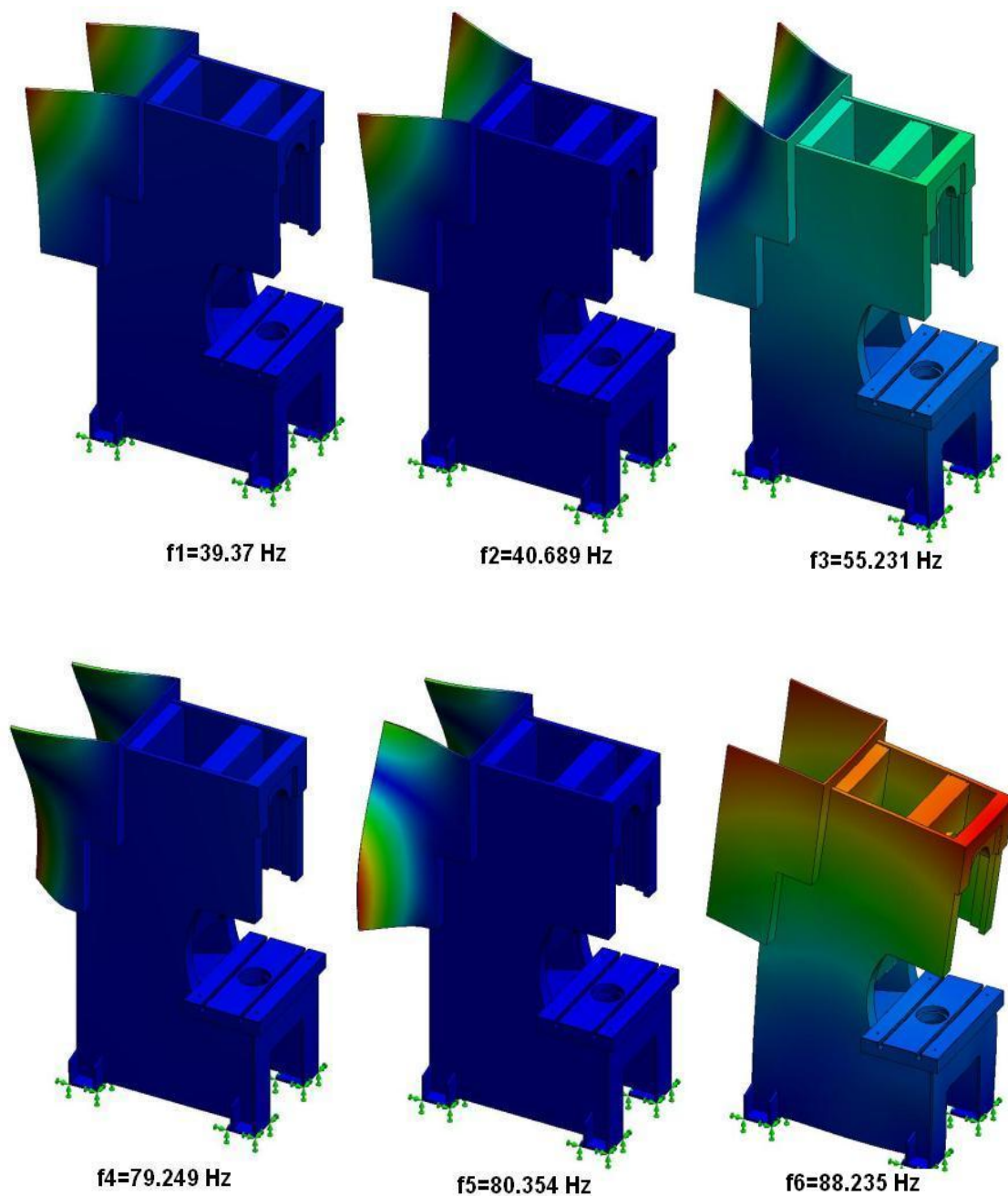


Figure 4.32 Natural frequency values belong to press body without welding with 80 tones capacity

Natural frequency analyses are performed for different welding size as well. By using 0.3, 0.5 and 0.7 values for welding seam size factor (X_Factor) mentioned in chapter 2, press models have been created and natural frequency values belong to structures are numerically determined as shown in Figure 4.33.

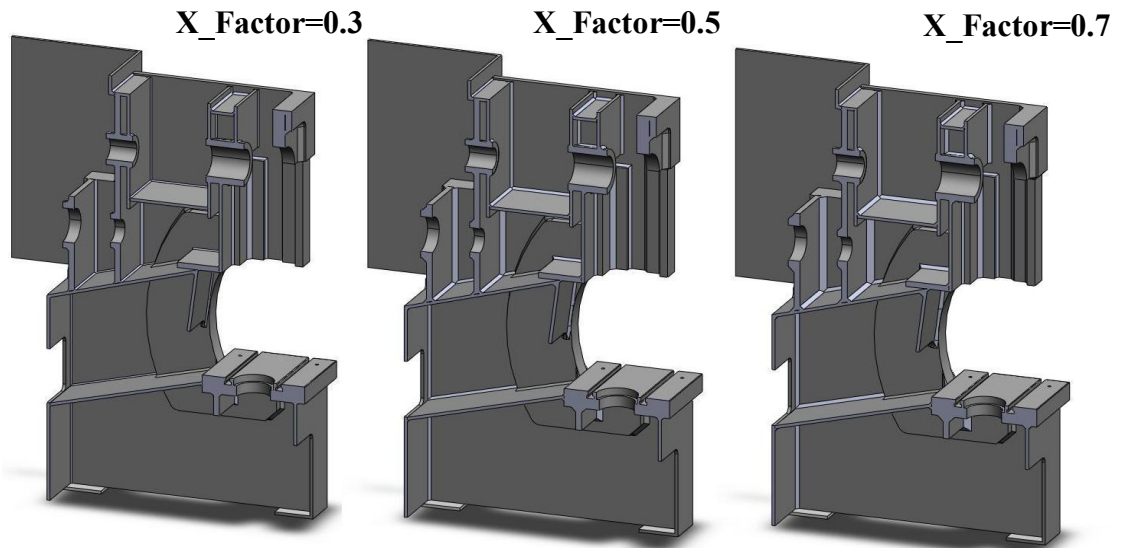


Figure 4.33 Press models having different welding seam sizes

Solid body and the finite element model which belong to welded press body created by considering the $X_Factor = 0.3$ are shown in Figure 4.34.

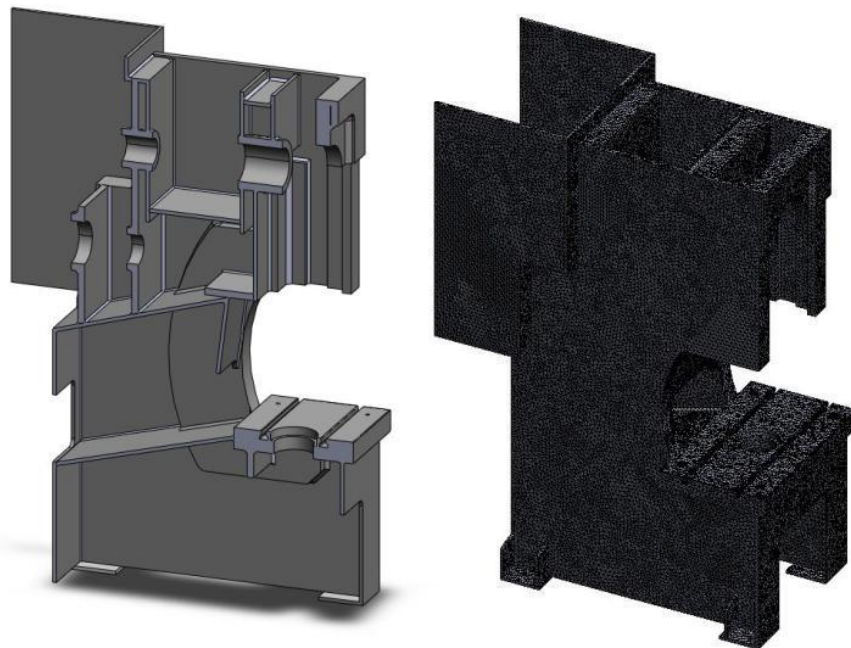


Figure 4.34 For $X_factor=0.3$ solid press model and finite elements model

First and third natural frequency values, which belong to free press body modeled with different welding seam sizes, are given in Figure 4.35 and 4.36. As is seen from the figure, alterations on welding seam size have significant effect on the natural frequency values. The structure is getting more rigid and frequency values increase by increasing the welding seam size.

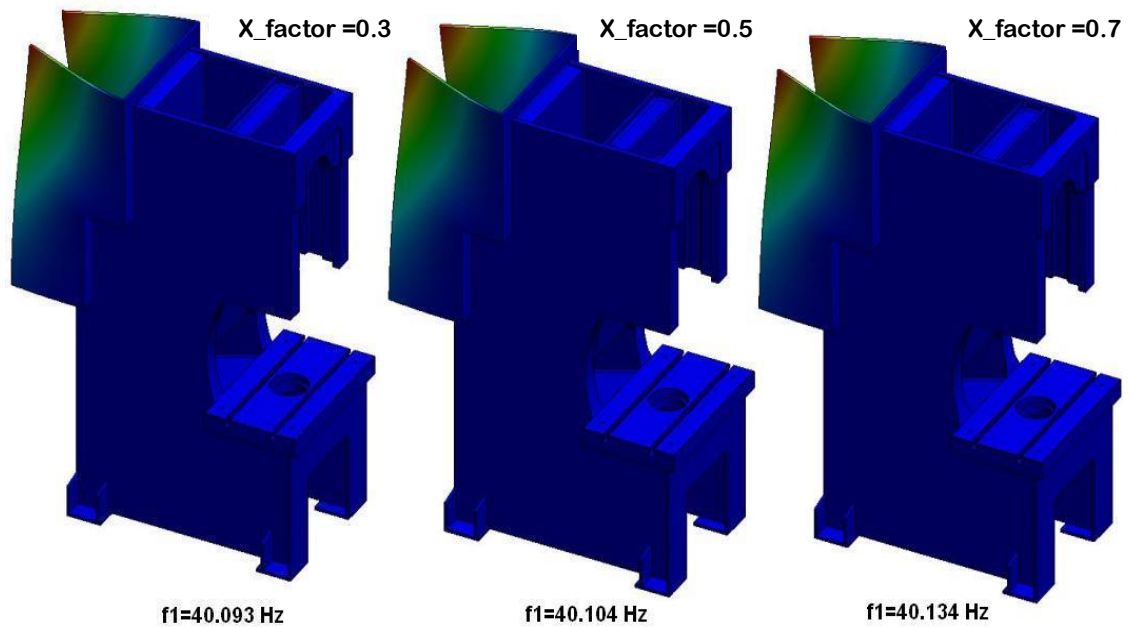


Figure 4.35 First natural frequency values for free model with different welding sizes

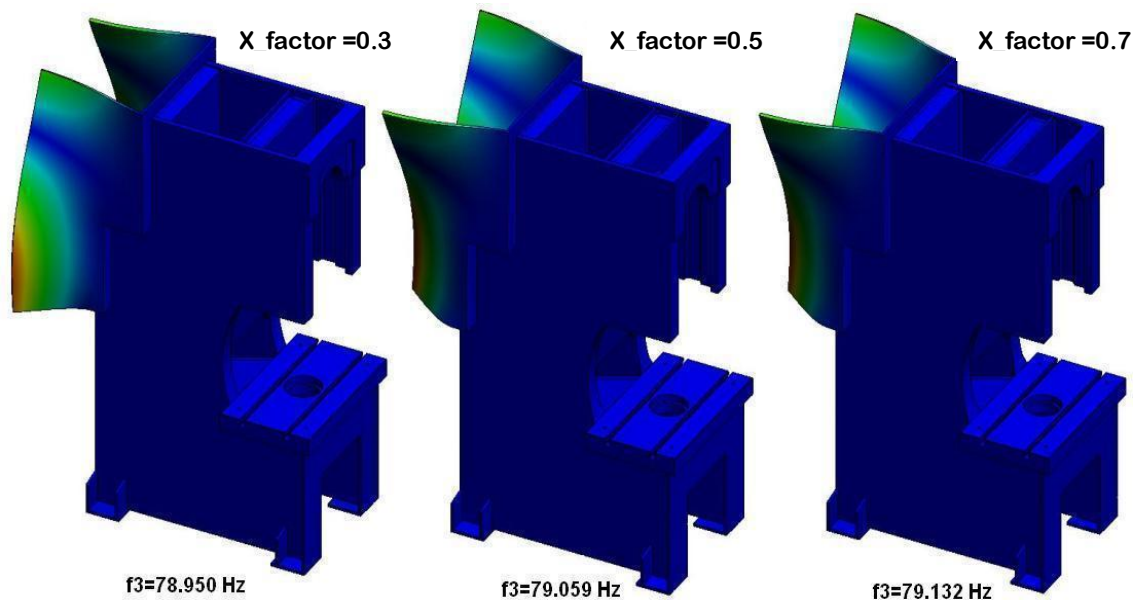


Figure 4.36 Third natural frequency values for free model with different welding sizes

First six natural frequency values belong to press body, are given in Table 4.2.

Table 4.2. Natural frequencies of free press body.

Natural Frequency (Hz)	X_factor		
	0.3	0.5	0.7
f1	40.093	40.104	40.134
f2	42.819	42.870	42.821
f3	78.950	79.059	79.132
f4	79.550	79.699	79.833
f5	116.55	118.710	118.400
f6	129.81	130.080	130.240

First and sixth natural frequency values, which belong to fixed press body modeled with different welding seam sizes, are given in Figure 4.37, Figure 4.38 and in Table 4.3. As can be seen from the figures, the natural frequencies of press body increase as the weld seam size increases.

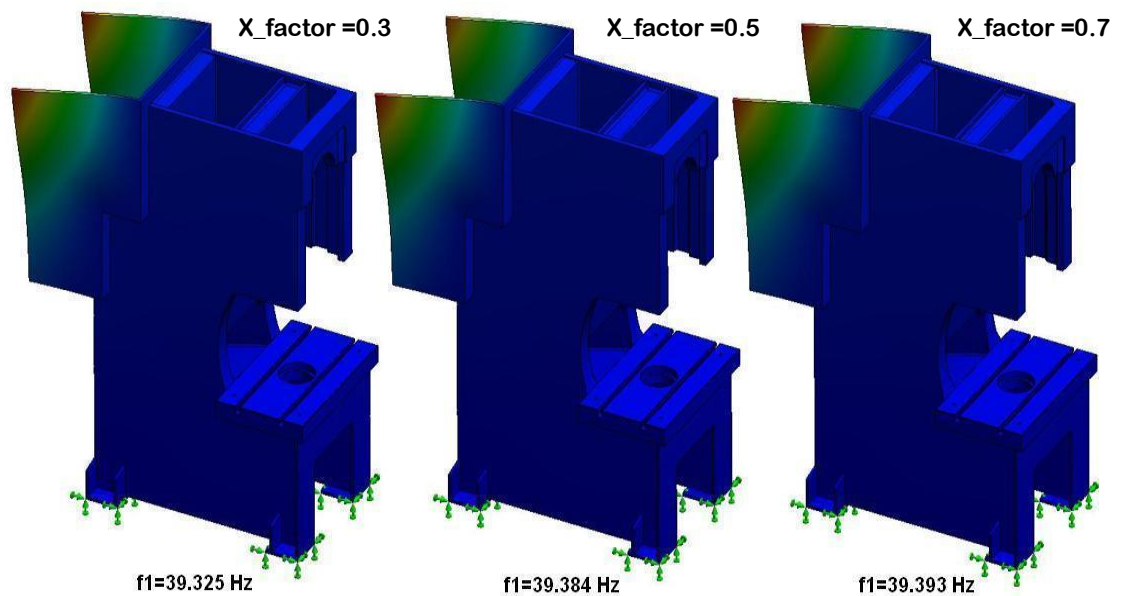


Figure 4.37 First natural frequency values for fixed body with different welding seam sizes

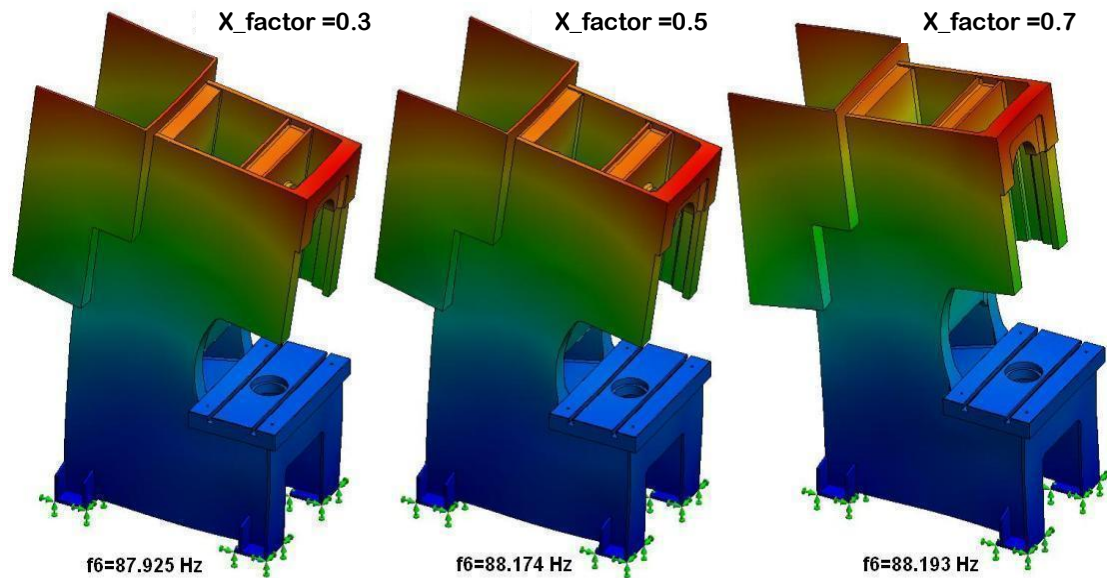


Figure 4.38 Sixth natural frequency values for fixed body with different welding seam sizes

Table 4.3. Natural frequencies of fixed press body.

Natural Frequency (Hz)	X factor		
	0.3	0.5	0.7
f1	39.325	39.384	39.393
f2	40.707	40.732	40.719
f3	54.299	54.604	54.651
f4	79.337	79.402	79.547
f5	80.459	80.476	80.609
f6	87.925	88.174	88.193

In the company, stress relieving treatment is executed by vibration method on press bodies in order to relief residual stresses occurring just after welding process. In order to execute this process, press body is placed on flexible chocks as shown in Figure 4.39 and press body is vibrated in different frequencies by vibromotor which is mountable on press table and has adjustable rotation frequencies. Body vibrations are measured by the help of an accelerometer from the region shown in figure simultaneously. Resonances belong to press body are detected by vibration measurement and press body is vibrated at this frequency in resonance condition in determined durations. Frequency response, which is obtained with the process

mentioned above for C type press body with 80 tones capacity is shown in Figure 4.40.

Resonance frequency values, which are shown in Figure 4.40, are in accordance with the frequency values given in Table 4.2.

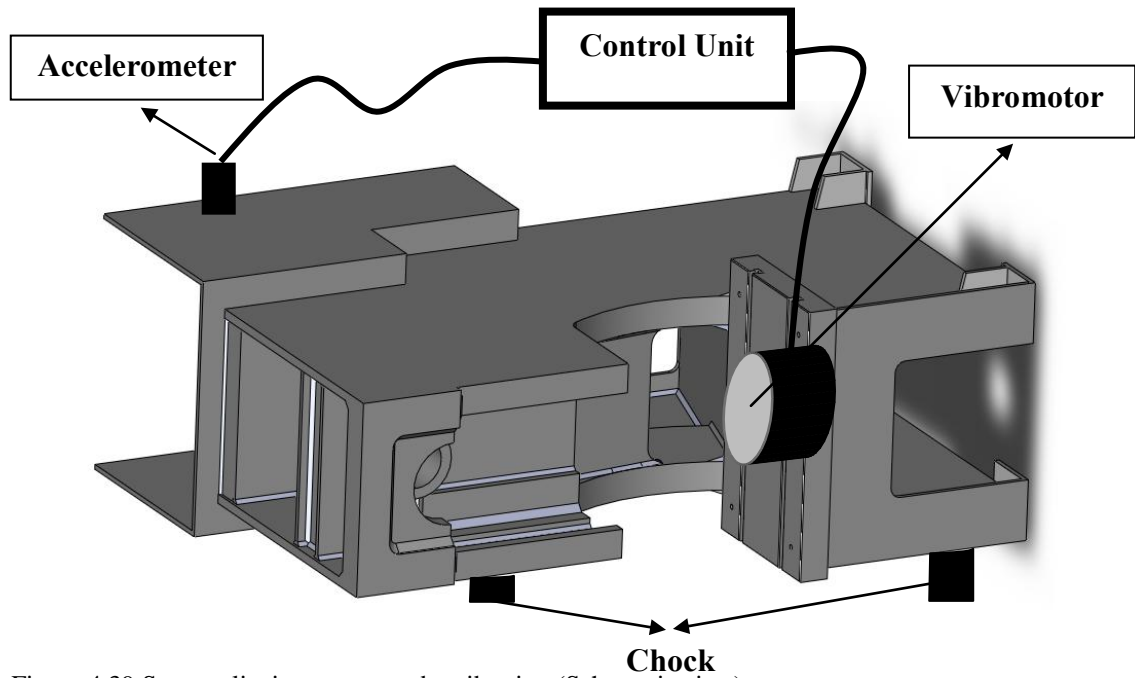


Figure 4.39 Stress relieving treatment by vibration (Schematic view)

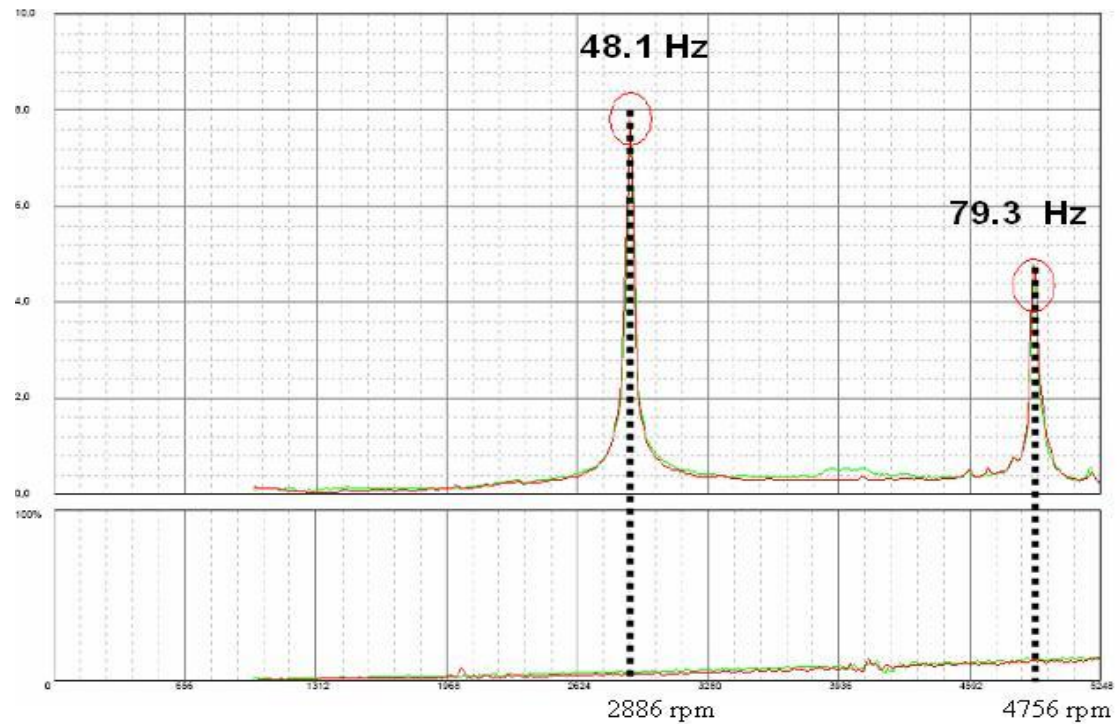


Figure 4.40 Report for Stress relieving treatment by vibration

4.5 The Effects of Welding Seam Size on Displacement Values of Press Body

In this section, effects of welding seam sizes on displacement values obtained from edges shown in Figure 4.28 are examined. Increments are observed on displacement values on the mentioned edge "C_üst" by using thin welding seams, as shown in Figure 4.41. The difference between welding factors 0.5 and 0.7 is not obvious.

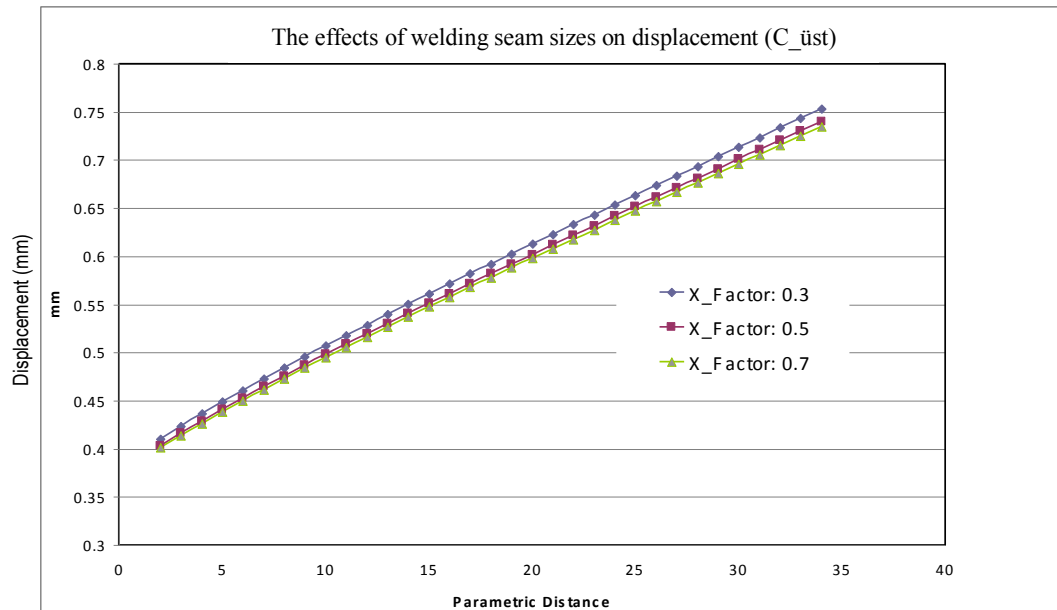


Figure 4.41 The effect of welding seam size on displacement of the edge "C_üst"

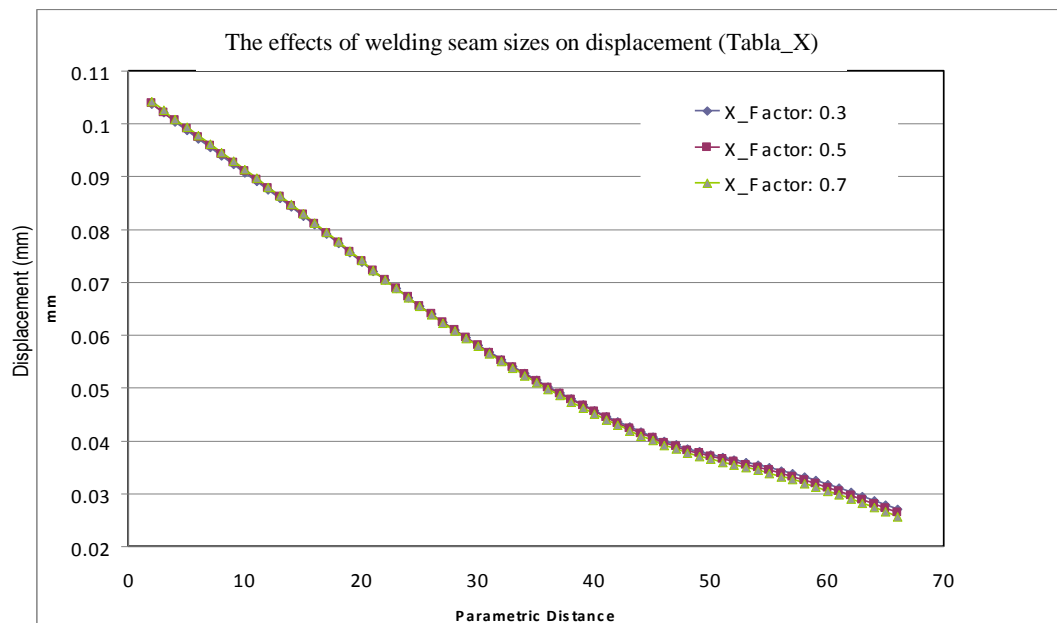


Figure 4.42 The effect of welding seam size on displacement of the edge "Tabla_X_alt"

As can be seen from the Figure 4.42, welding seam sizes have not a significant importance on displacement values of edge "Tabla_x_alt". However, welding seam sizes are effective on displacement values of the edge "Tabla_Z" as shown in Figure 4.43.

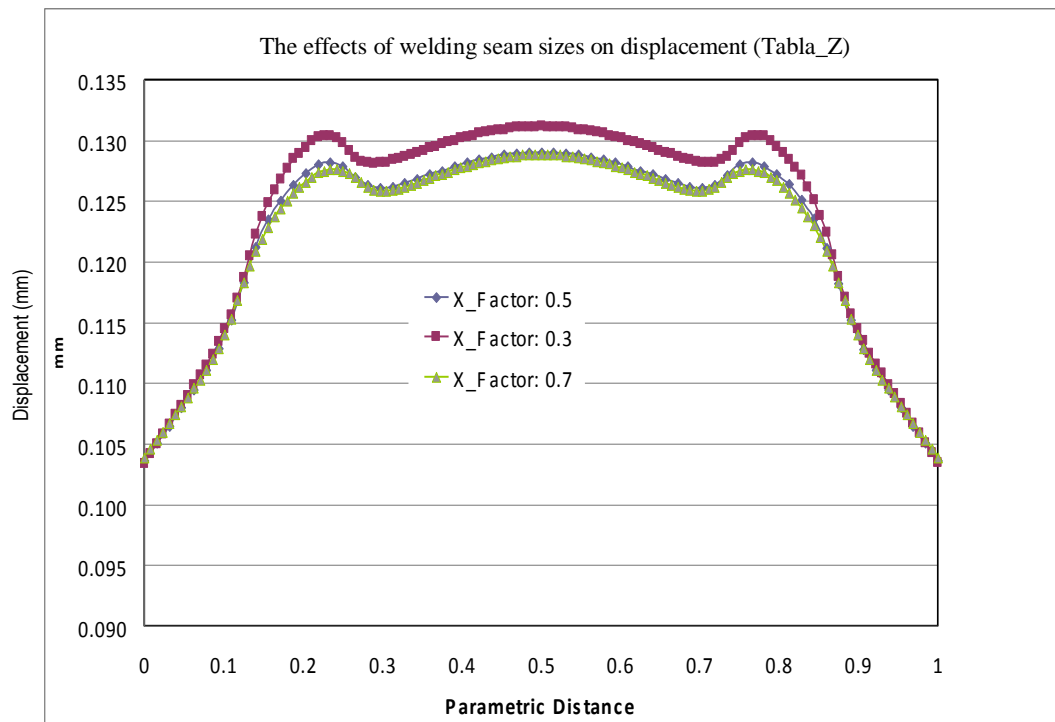


Figure 4.43 The effect of welding seam size on displacement of the edge"Tabla_Z"

4.6 The Effects of Welding Seam Size on Stress Values of Press Body

In this section, effects of welding seam size, on numerically determined equivalent von Mises stress values all along the welding seams on significant regions on press body, are studied. In numeric analyses 0.3, 0.5, and 0.7 X_Factor values are used for welding seams. X_factor=0.5 value corresponds the welding seam size which is used in present production. For the welding seam covering the "C_Saci" X_Factor definition is not used and the current welding seam size is given directly in mm. Welding seam regions, where the effect of welding seam size is studied, are shown in Figure 4.44.

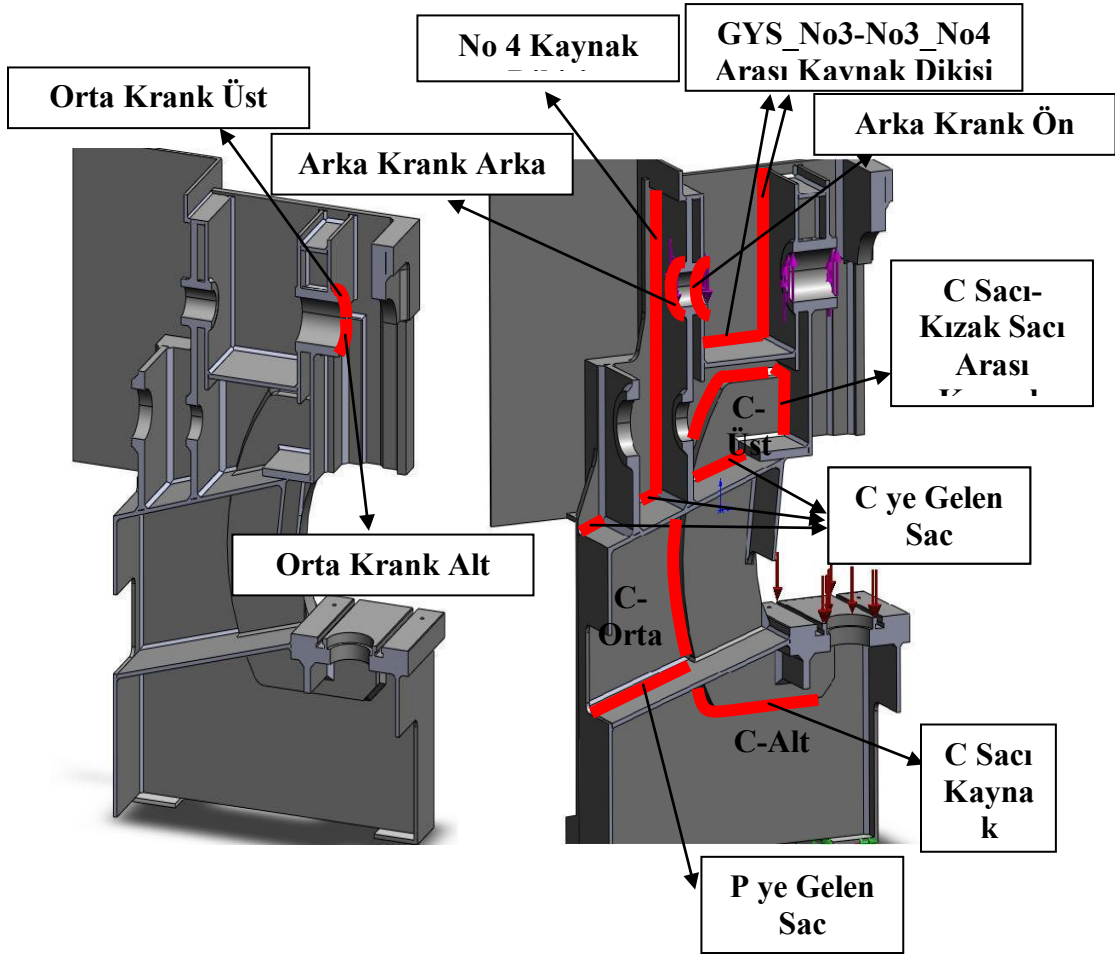


Figure 4.44 Analysed welding seam regions

Stress values for welding seam lines shown in Figure 4.44, which are determined with different welding sizes, are given between Figure 4.45 and Figure 4.56. When these results are studied, generally it is seen that in case welding seam sizes decrease, stress values increase. This condition is obviously observed in the line "GYS_No3-No3_No4". However, in some cases, parallel with sectionally welding seam sizes increment, decrement is observed in stress values.

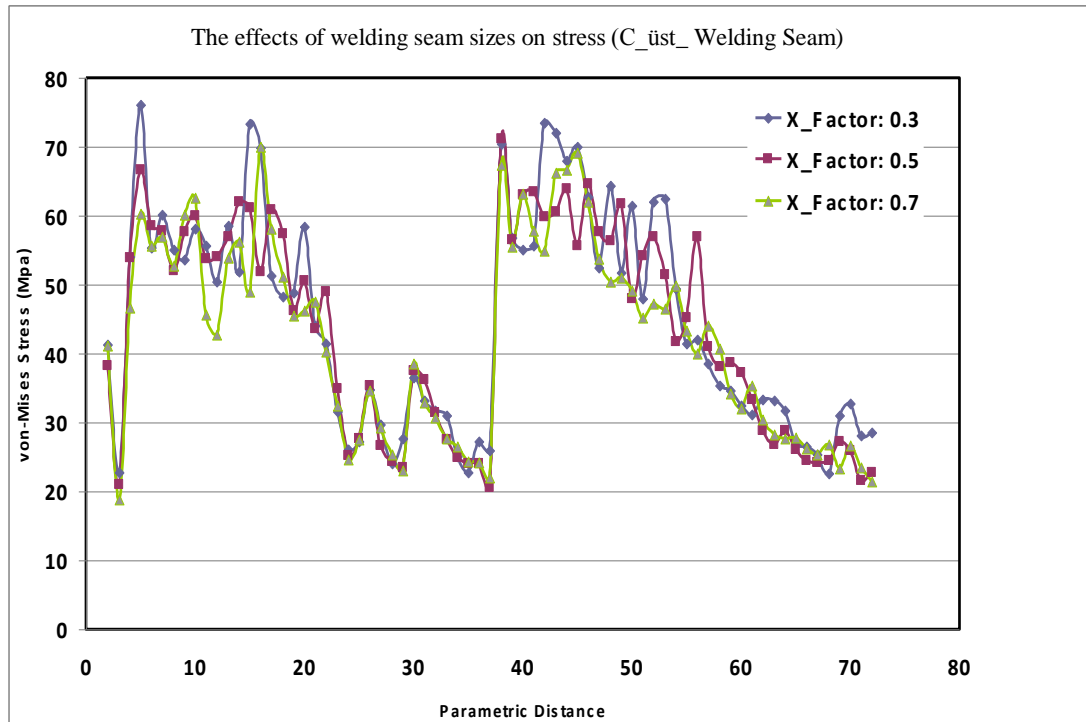


Figure 4.45 The effects of welding seam sizes on stress

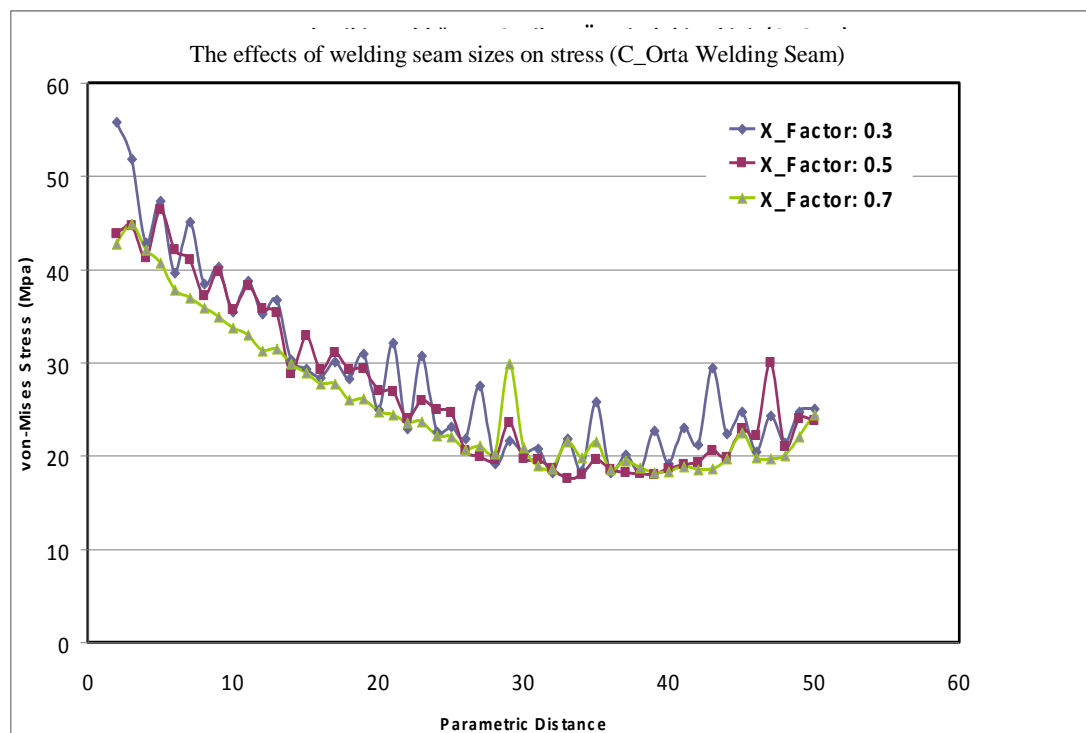


Figure 4.46 The effects of welding seam sizes on stress



Figure 4.47 The effects of welding seam sizes on stress

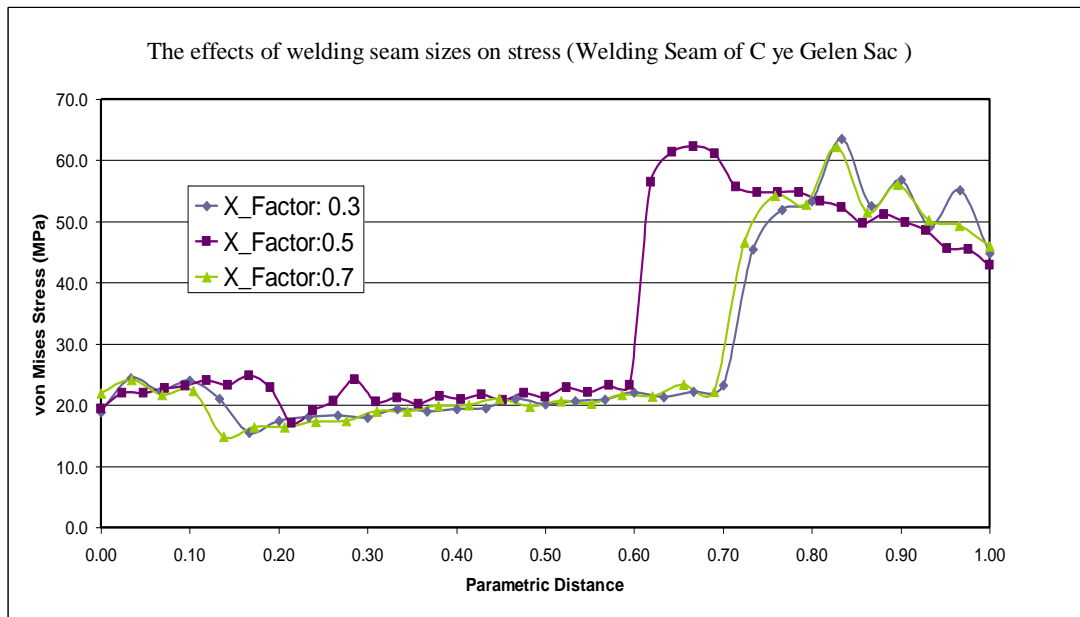


Figure 4.48 The effects of welding seam sizes on stress

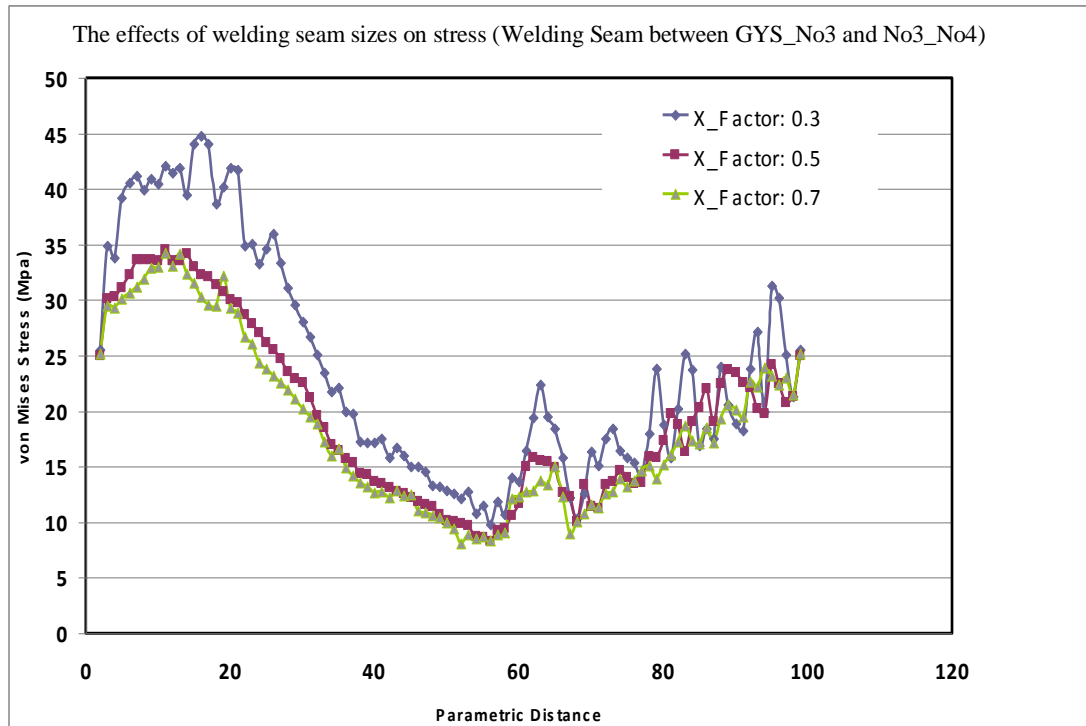


Figure 4.49 The effects of welding seam sizes on stress

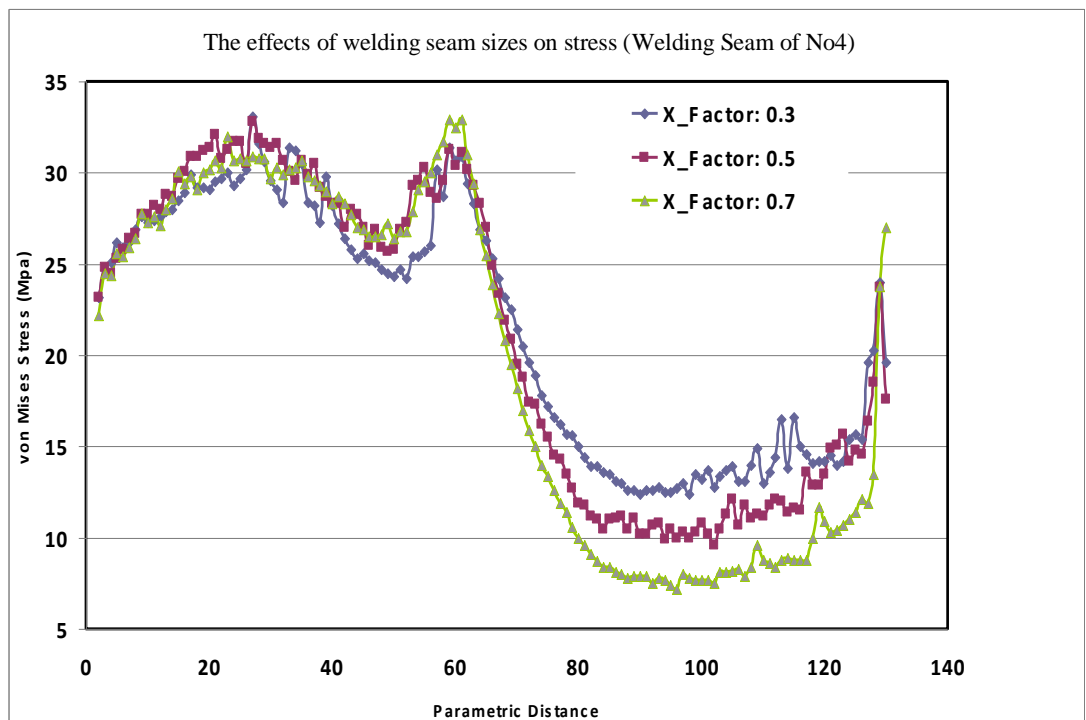


Figure 4.50 The effects of welding seam sizes on stress



Figure 4.51 The effects of welding seam sizes on stress

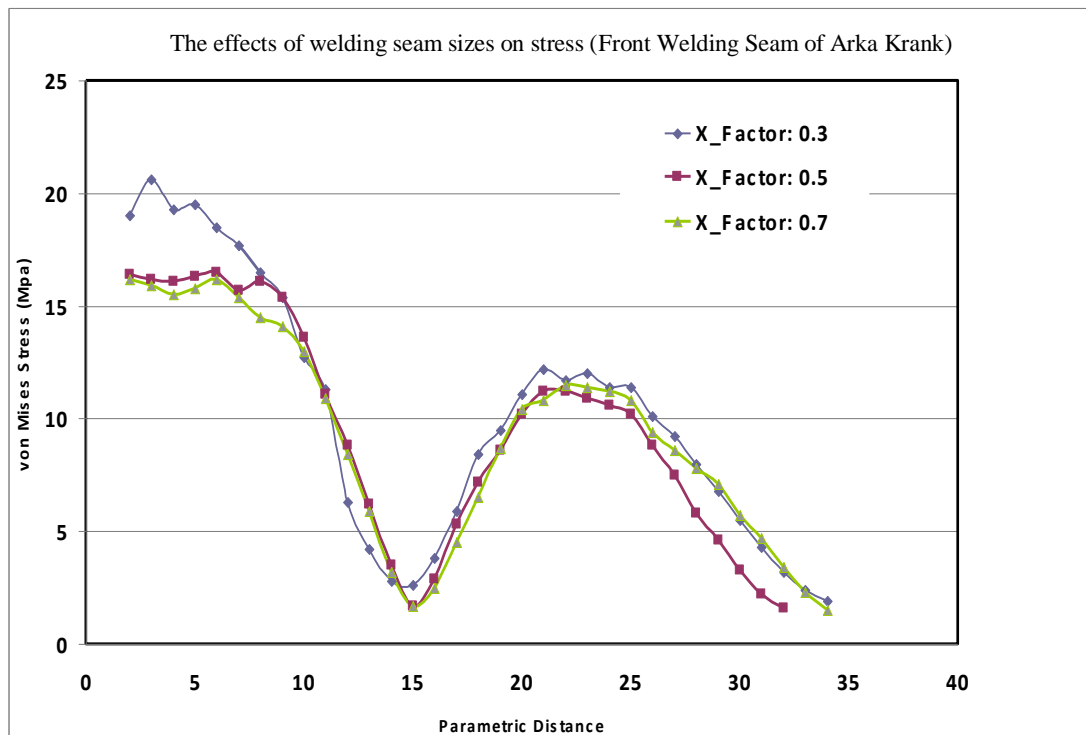


Figure 4.52 The effects of welding seam sizes on stress

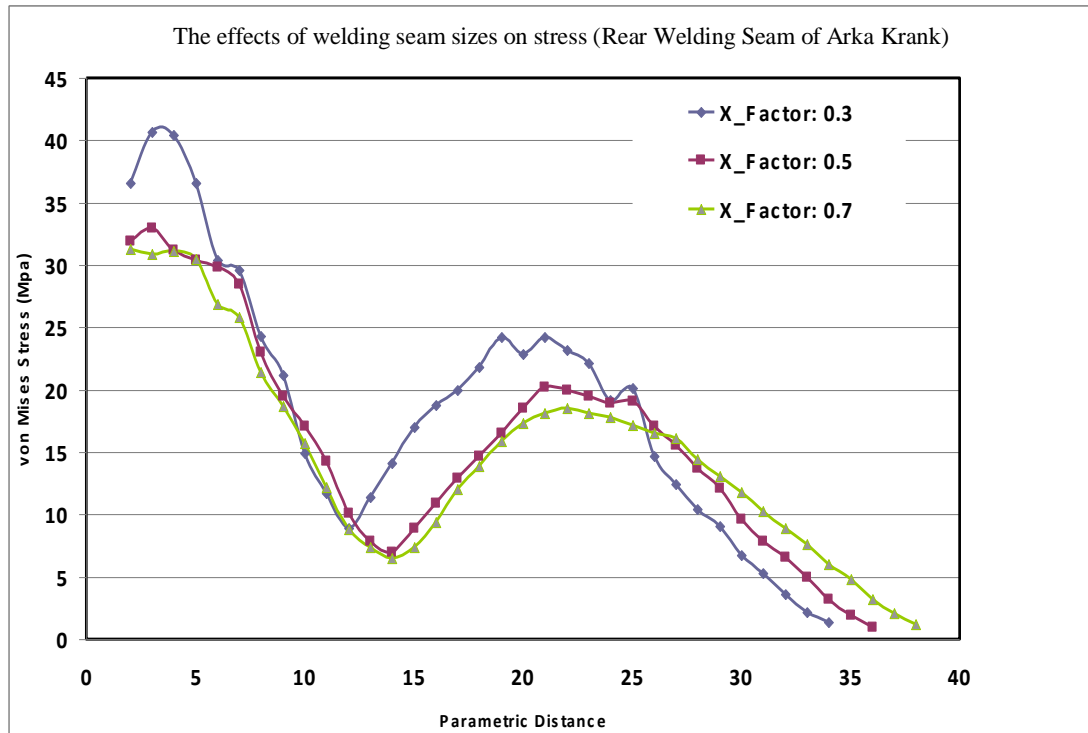


Figure 4.53 The effects of welding seam sizes on stress

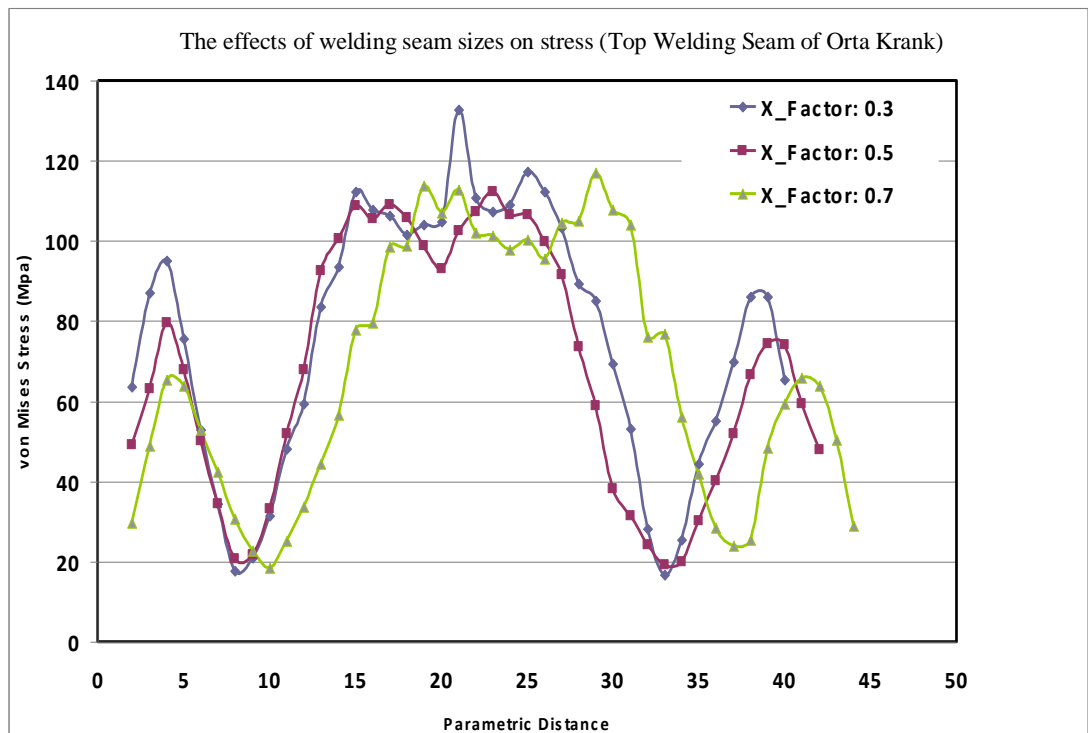


Figure 4.54 The effects of welding seam sizes on stress

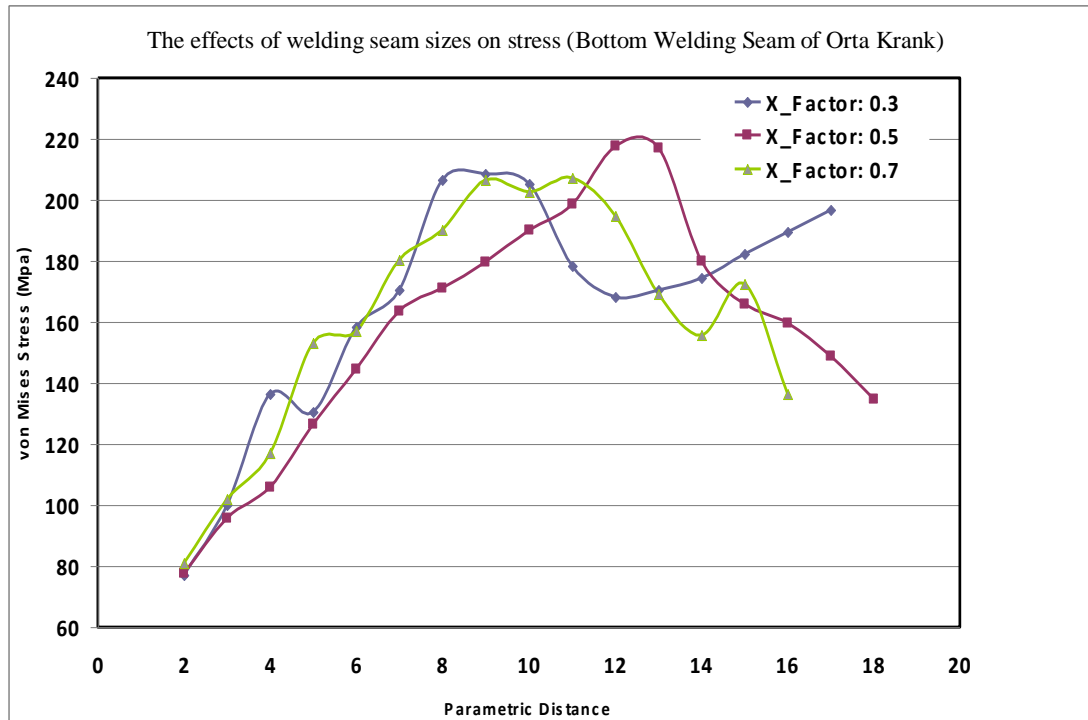


Figure 4.55 The effects of welding seam sizes on stress

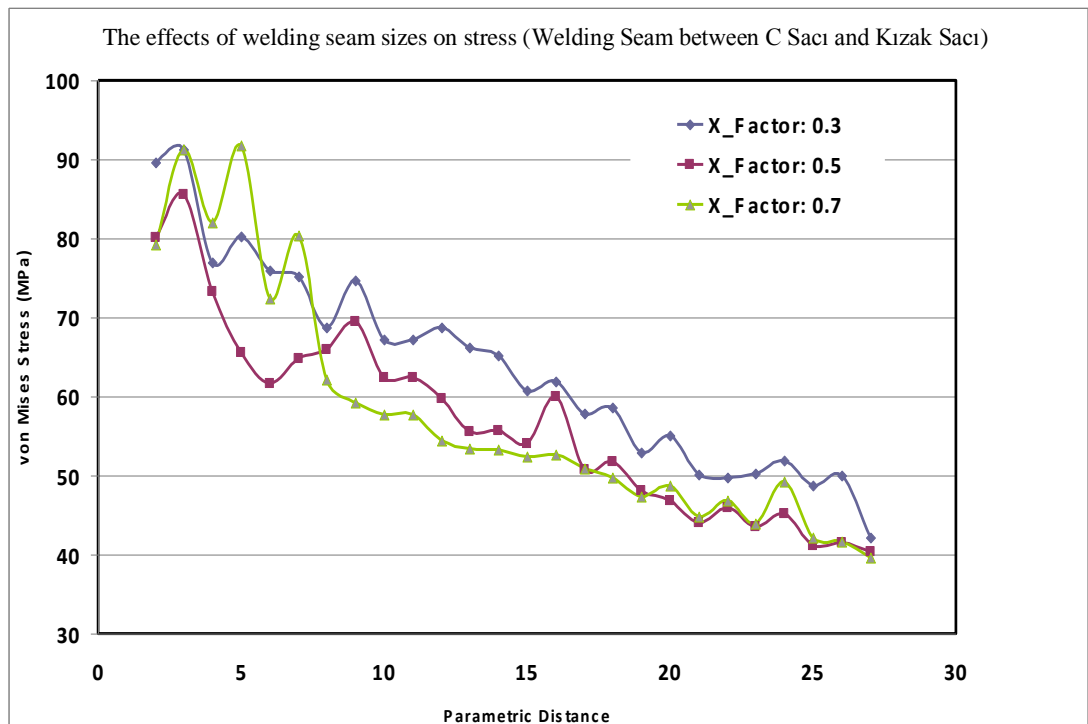


Figure 4.56 The effects of welding seam sizes on stress

4.7 The Effects of Welding Seam on Displacement Values of Different Combination

In this section the results of analyses performed for C type press body with 1500 mm ram distance and 630 mm distance between two "Govde Yan Sacı" are given. It is studied how weld seam sizes around "C Sacı", "C ye gelen sac" and "P ye gelen sac" affect the displacement values on the edges given in Figure 4.28.

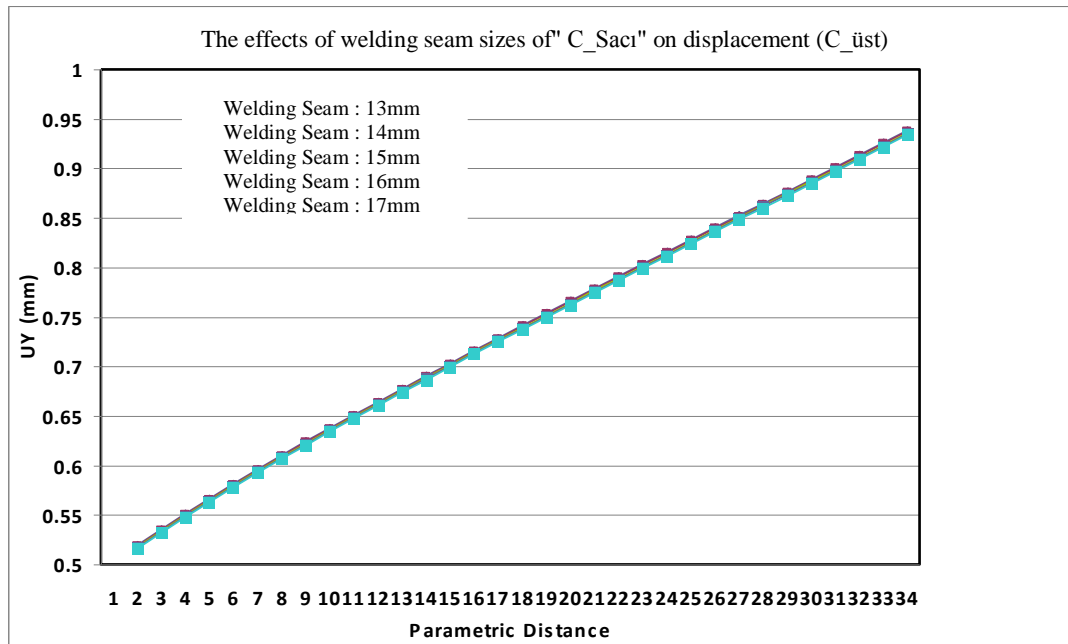


Figure 4.57 The effects of welding seam sizes on displacement



Figure 4.58 The effects of welding seam sizes on displacement

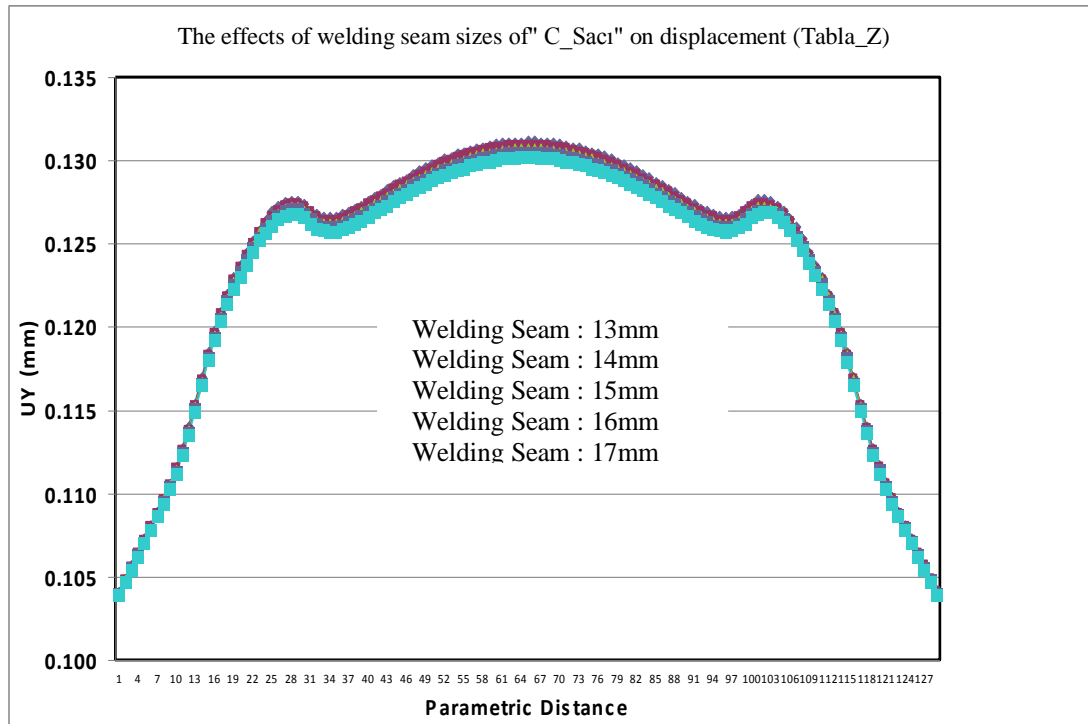


Figure 4.59 The effects of welding seam sizes on displacement

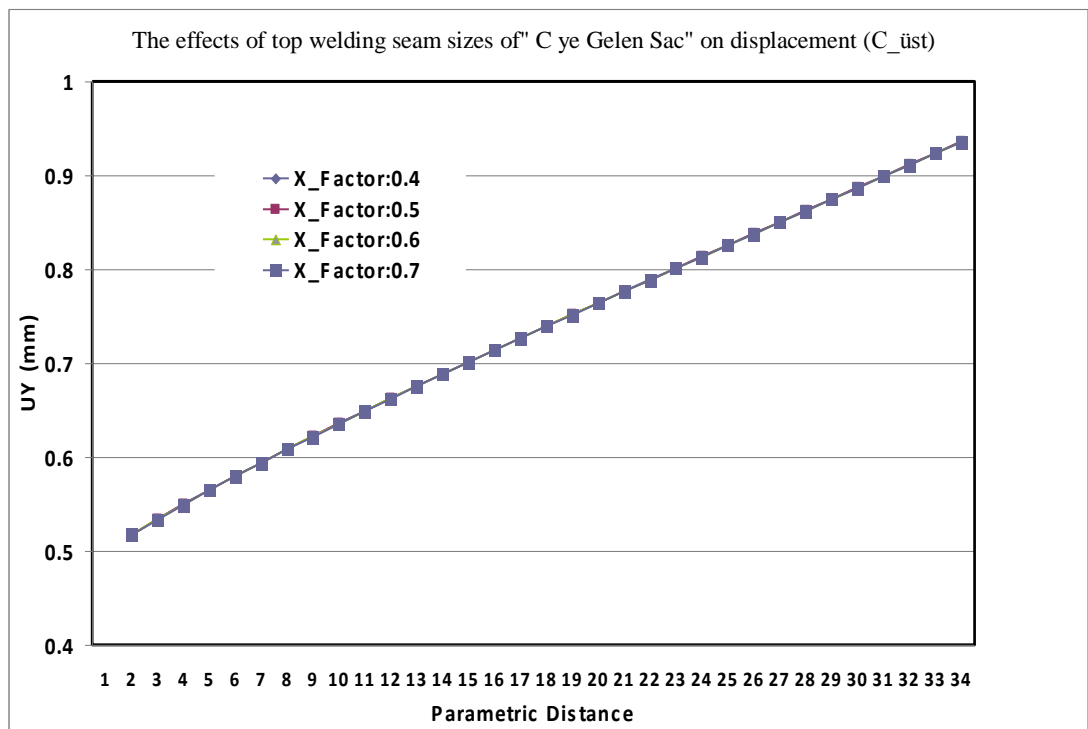


Figure 4.60 The effects of welding seam sizes on displacement

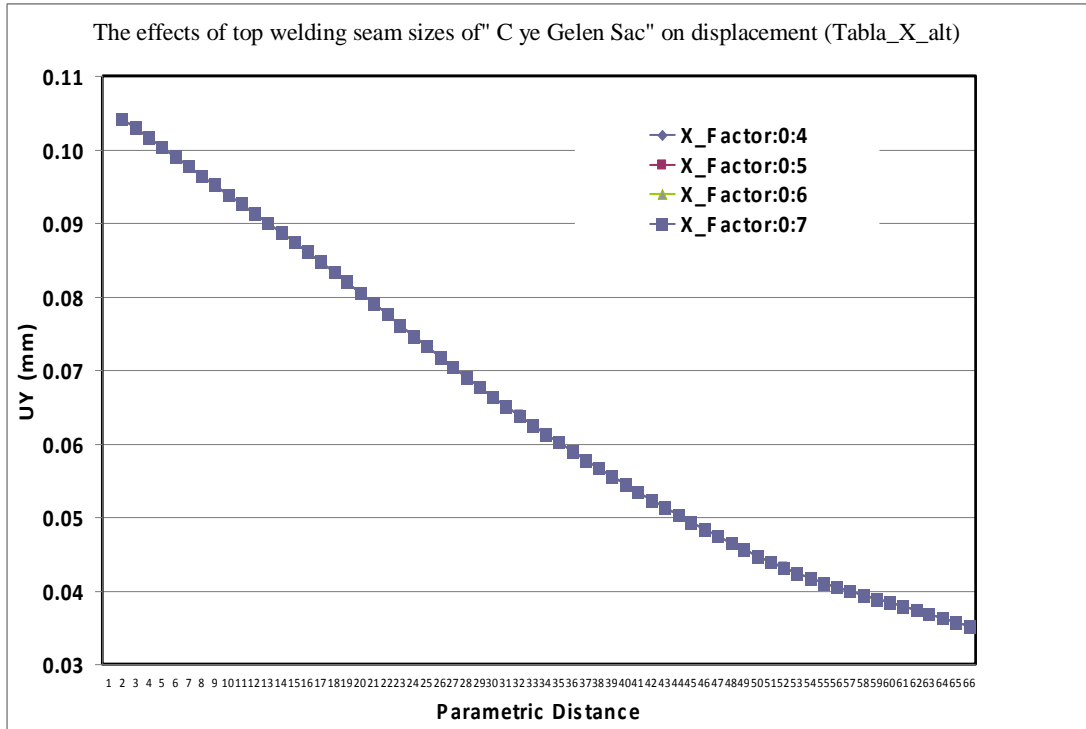


Figure 4.61 The effects of welding seam sizes on displacement

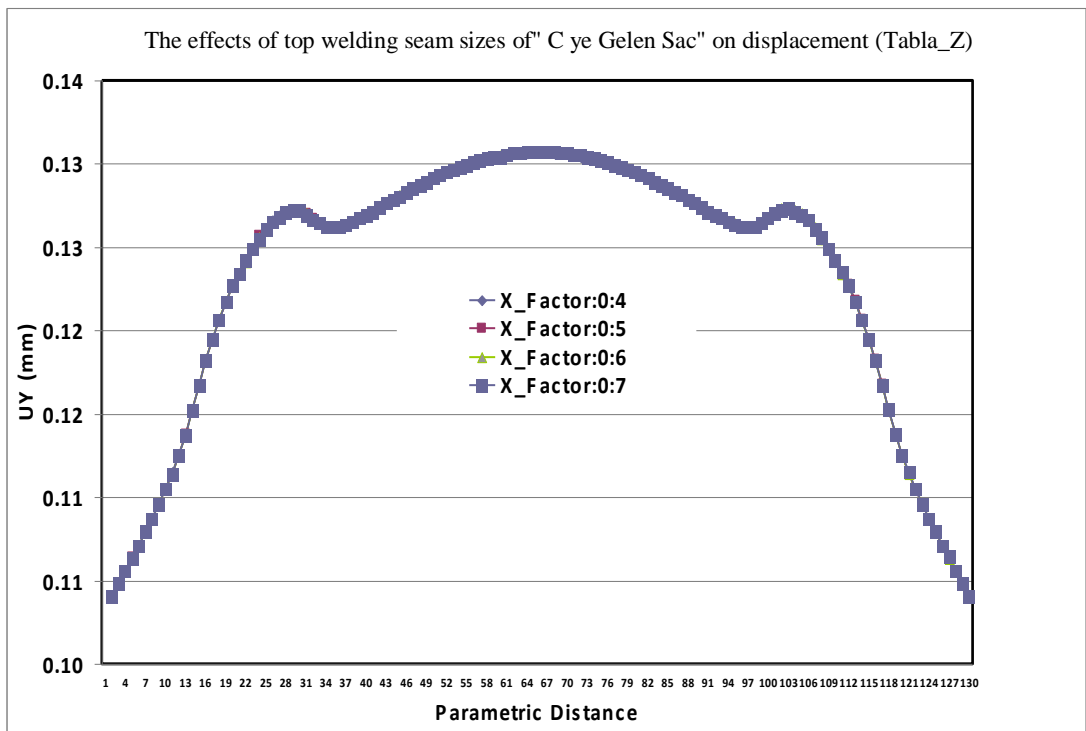


Figure 4.62 The effects of welding seam sizes on displacement

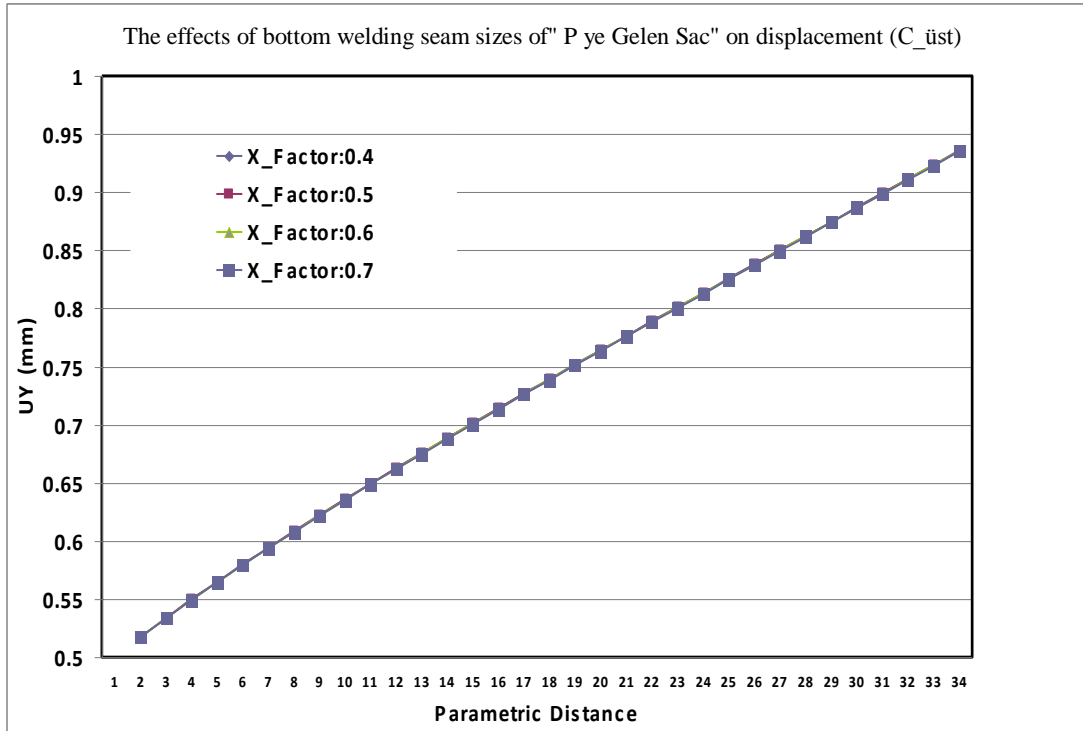


Figure 4.63 The effects of welding seam sizes on displacement

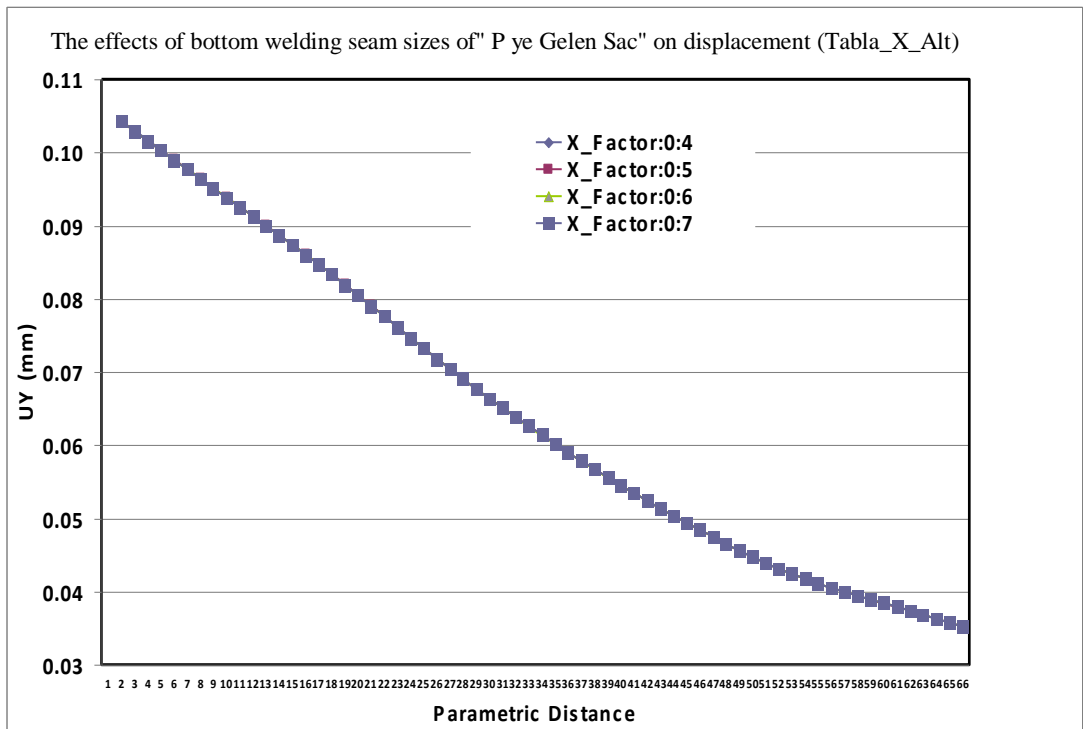


Figure 4.64 The effects of welding seam sizes on displacement

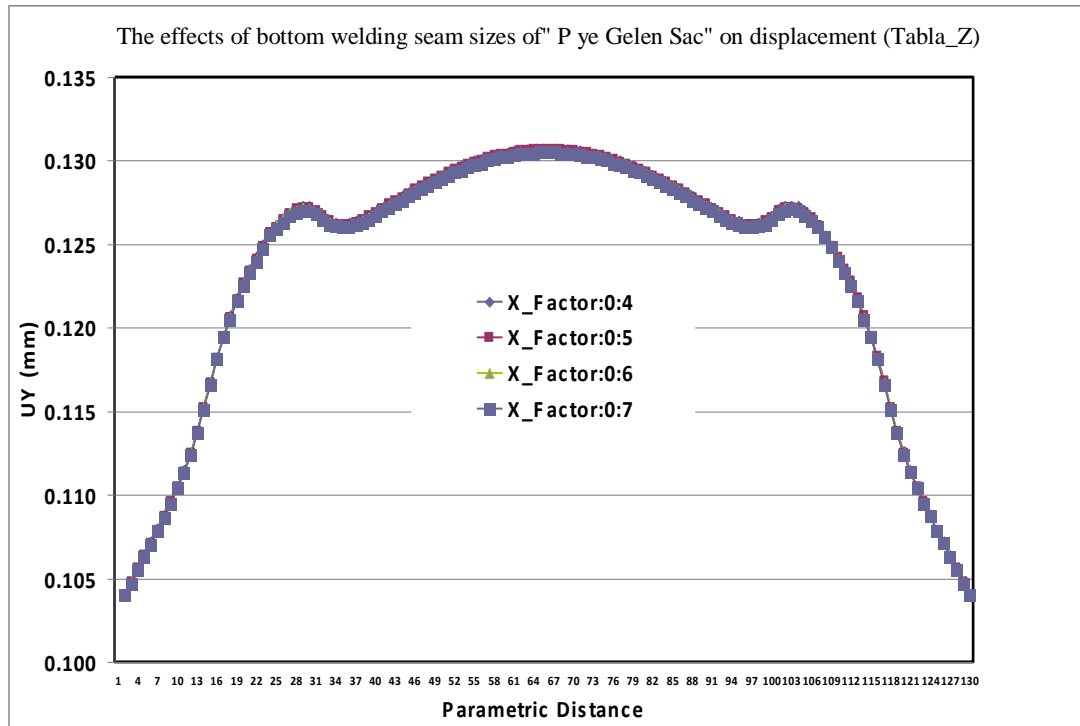


Figure 4.65 The effects of welding seam sizes on displacement

When the displacement responses are studied, it is observed that, alterations in considered welding seam sizes have not a significant importance on displacement values of examined regions on press.

4.8 Regional Effects of Welding Seam Size on Stress Values of Press Body

In this section, effects of welding seam sizes, shown in Figure 4.44, on the stress values along the welding lines are studied. While the related welding seam is changed the other welding seams surrounding the body remained in default size ($X_Factor = 0.5$). As seen from the numeric stress values given in Figures 4.66 and 4.77, by changing the regional welding seams, partially significant alterations on stress values on press body are observed. Generally, increase in welding seam size decreases the stress values on studying regions.

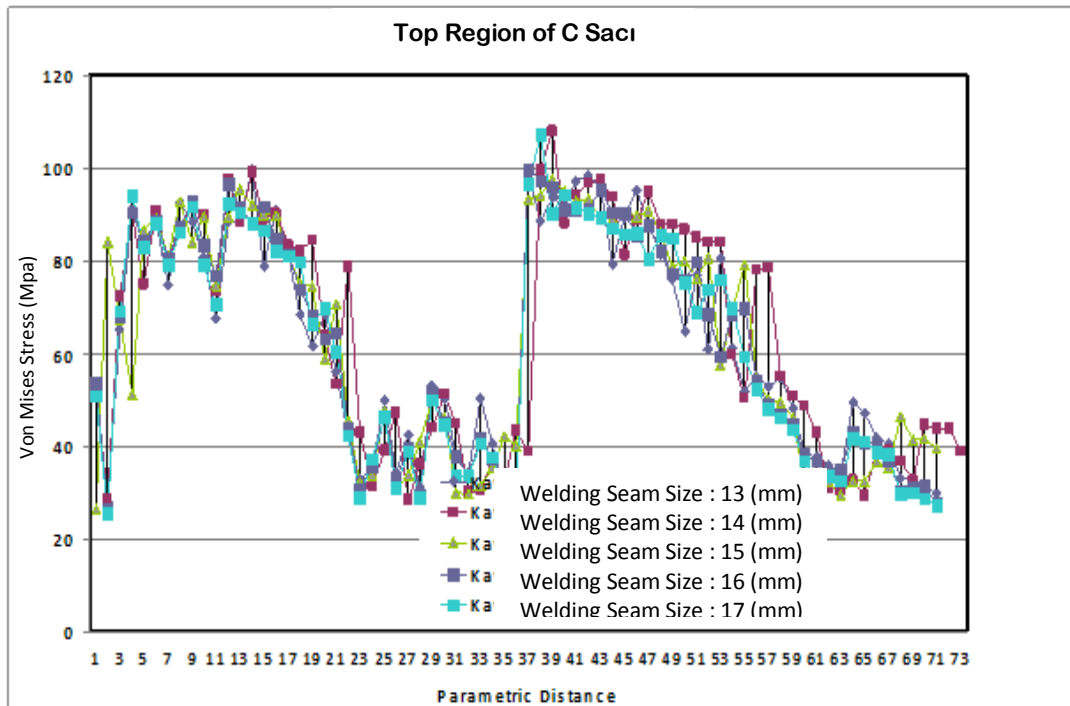


Figure 4.66 Regional effects of welding seam sizes on stress values

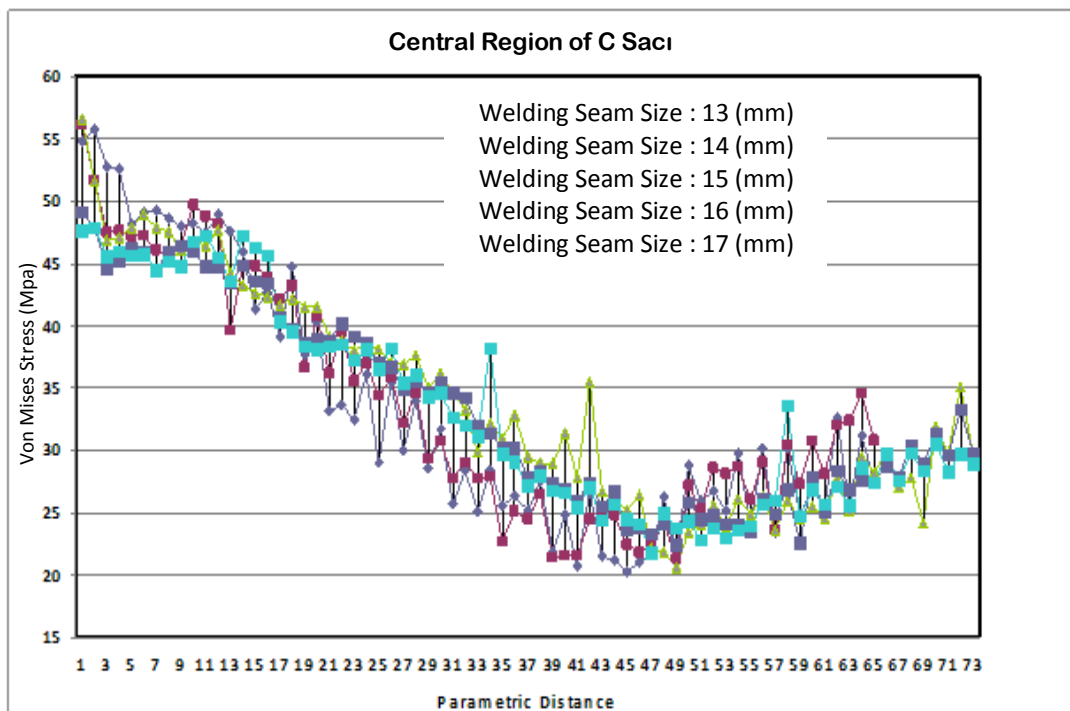


Figure 4.67 Regional effects of welding seam sizes on stress values

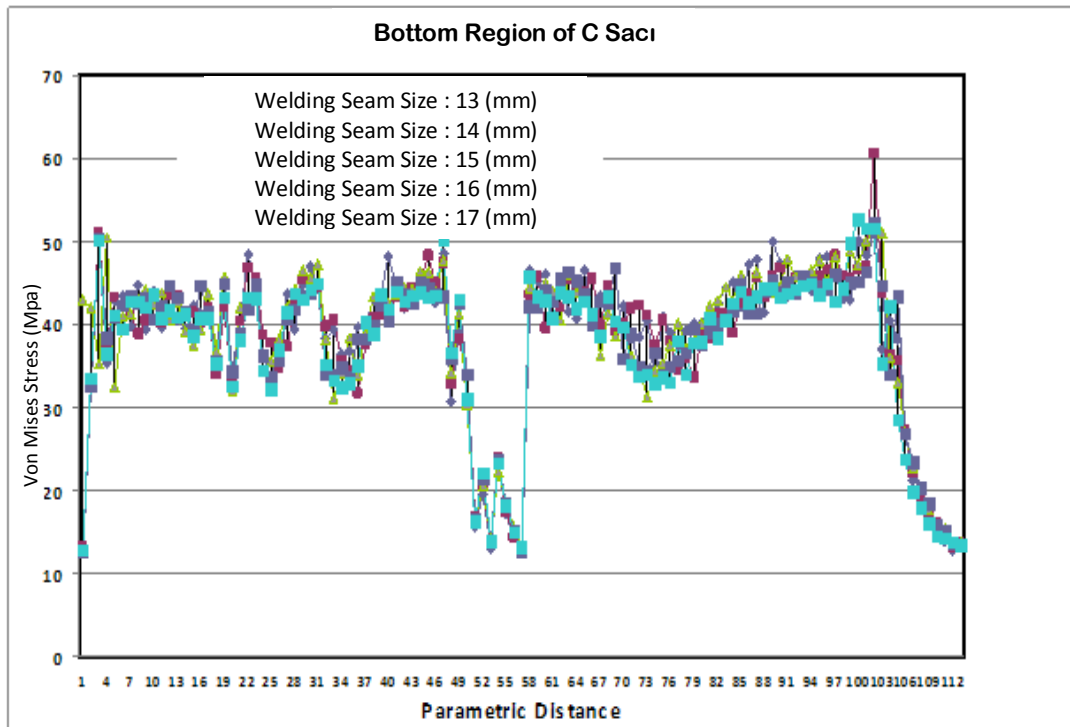


Figure 4.68 Regional effects of welding seam sizes on stress values

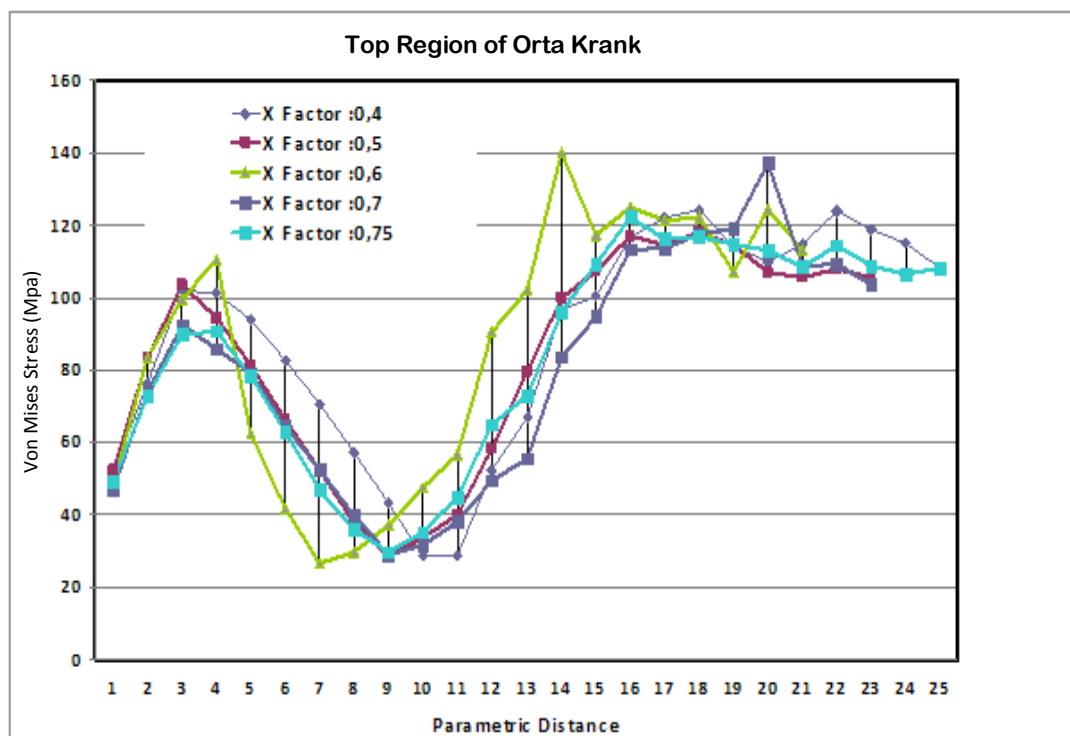


Figure 4.69 Regional effects of welding seam sizes on stress values

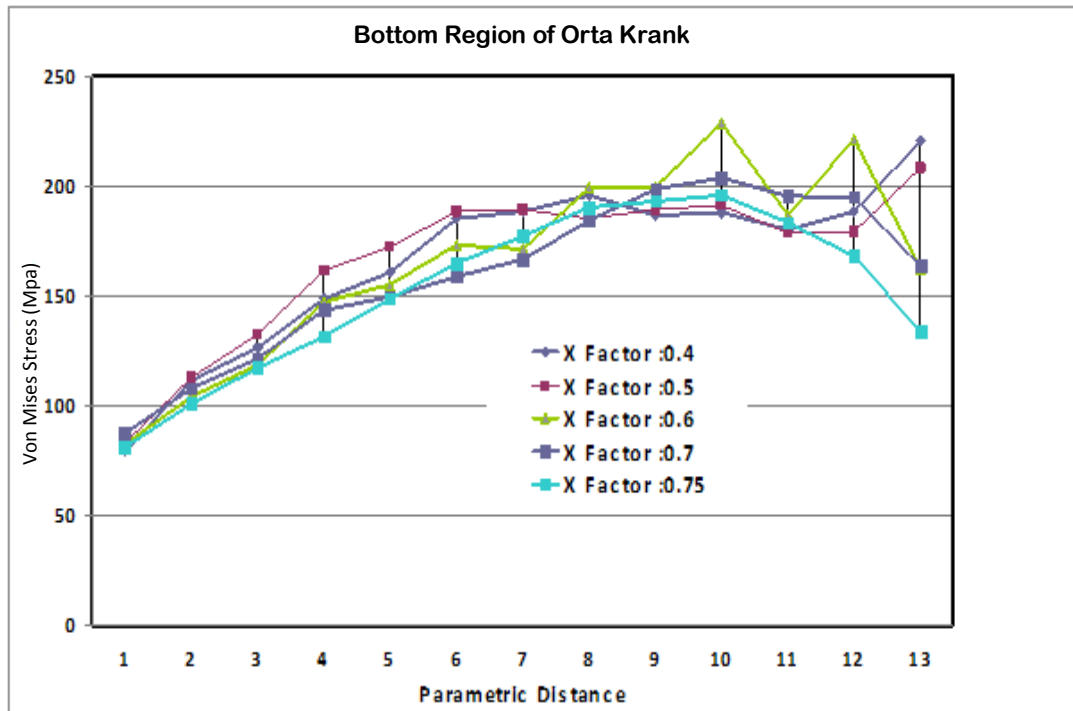


Figure 4.70 Regional effects of welding seam sizes on stress values

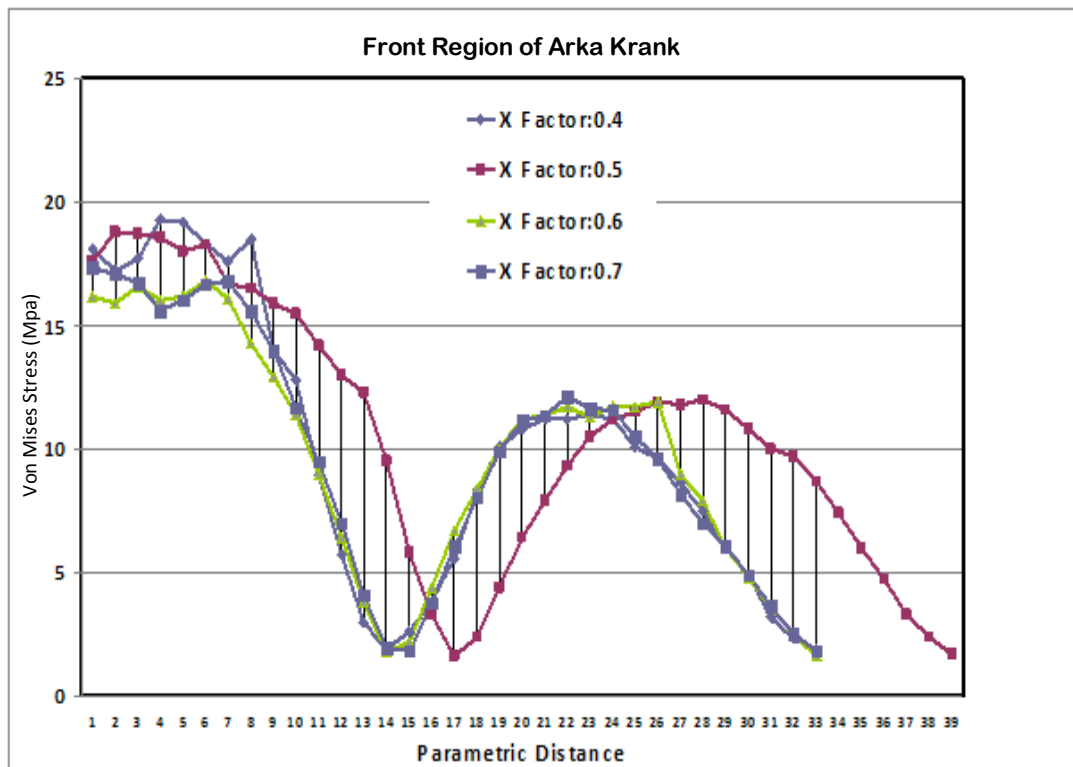


Figure 4.71 Regional effects of welding seam sizes on stress values

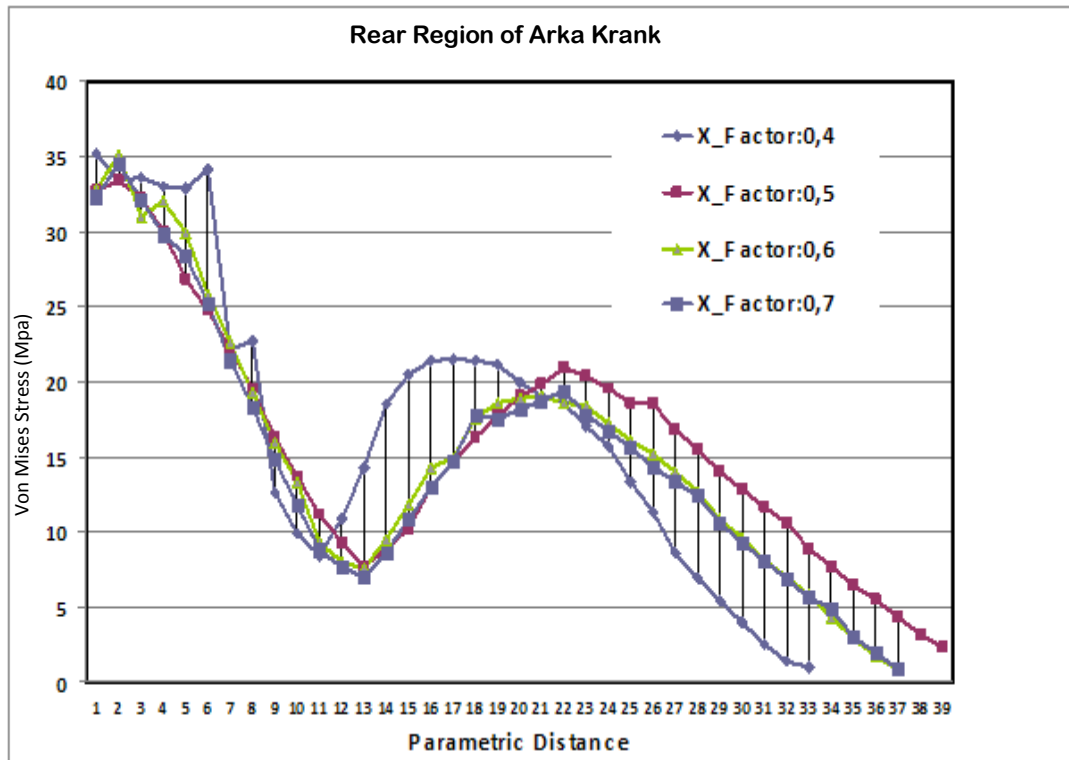


Figure 4.72 Regional effects of welding seam sizes on stress values

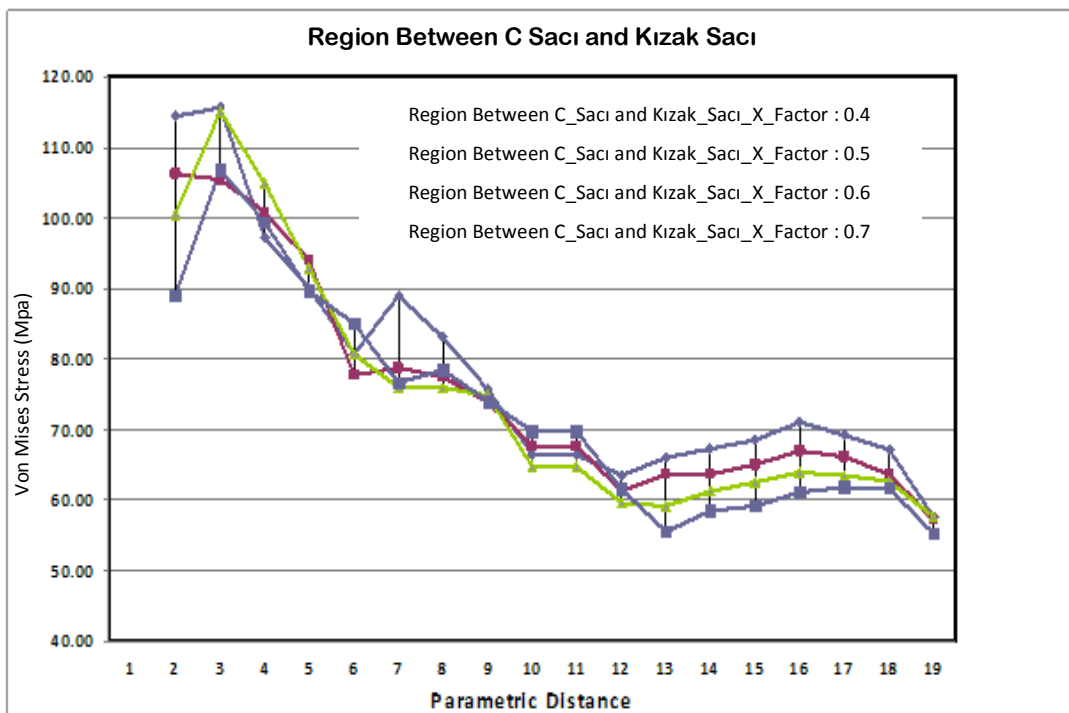


Figure 4.73 Regional effects of welding seam sizes on stress values

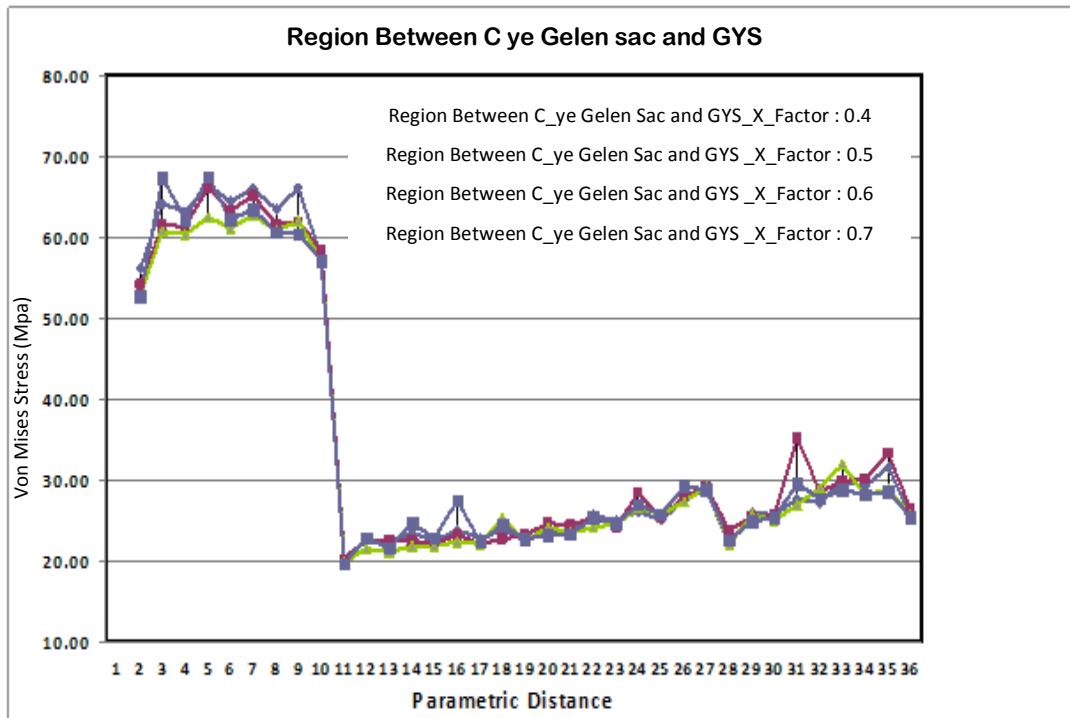


Figure 4.74 Regional effects of welding seam sizes on stress values

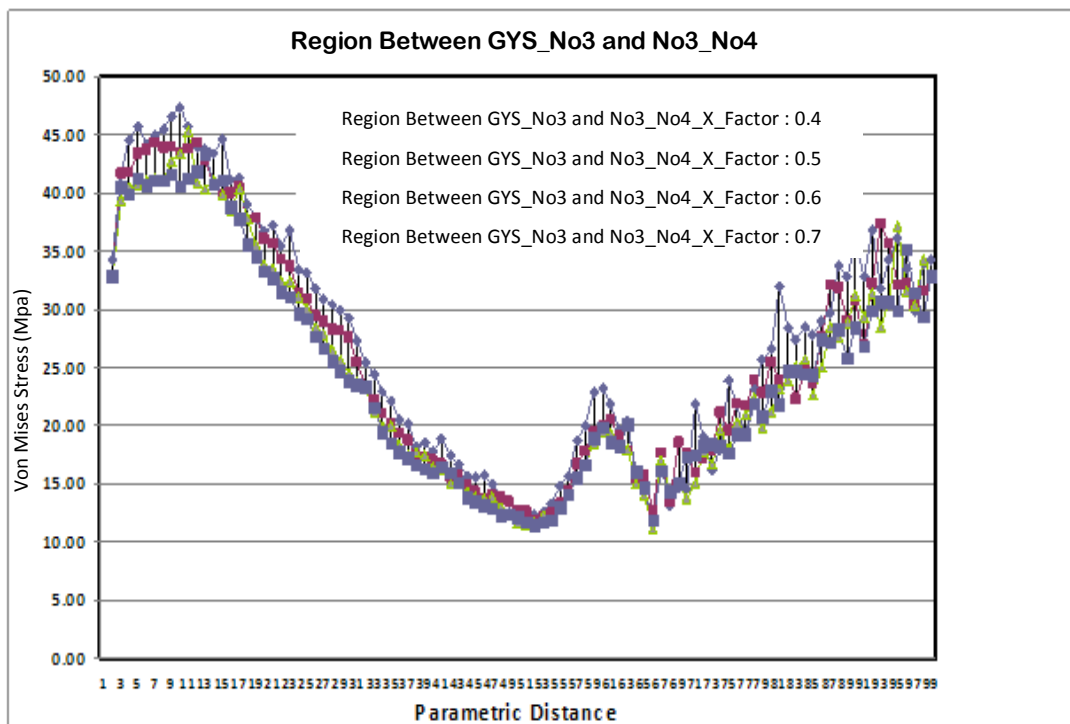


Figure 4.75 Regional effects of welding seam sizes on stress values

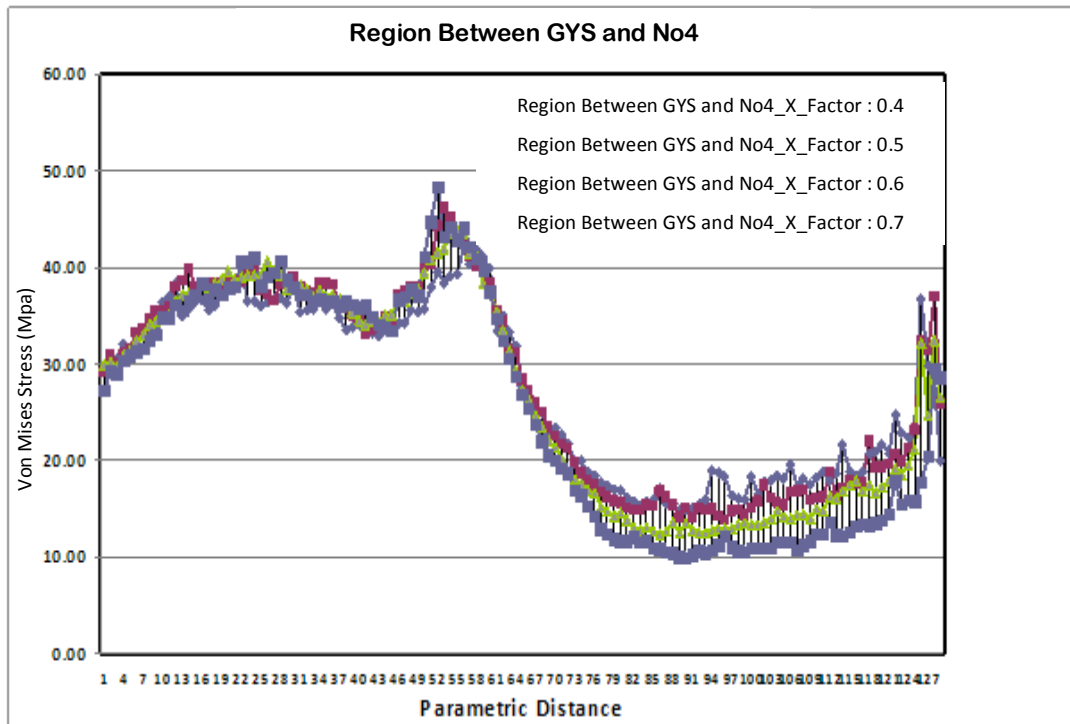


Figure 4.76 Regional effects of welding seam sizes on stress values

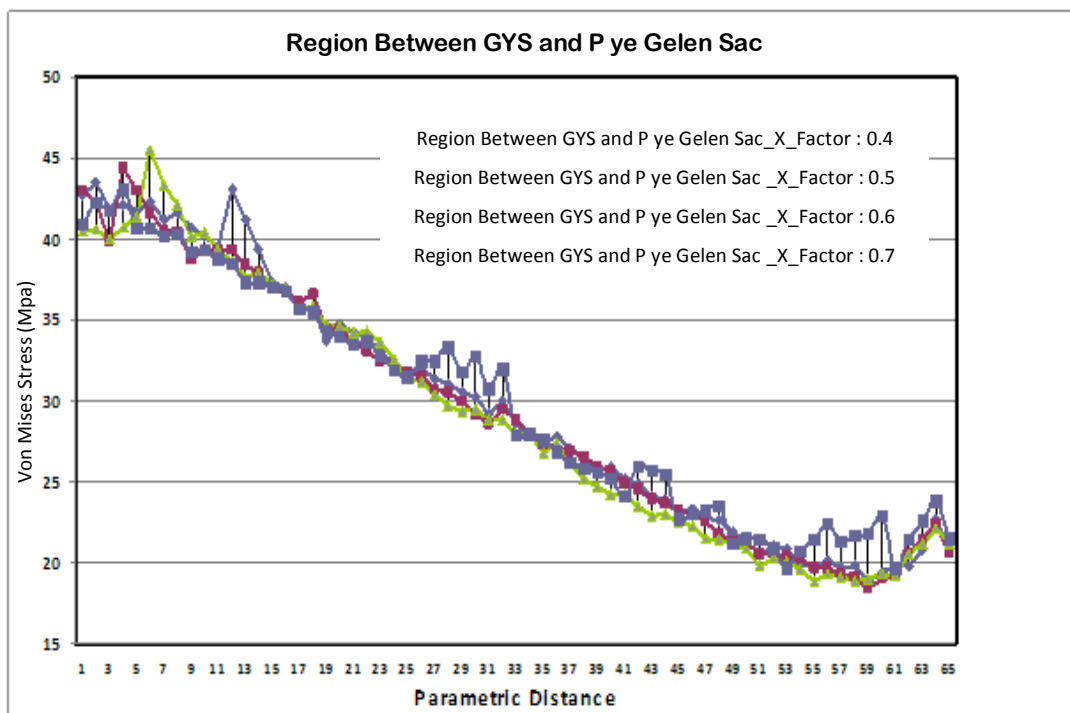


Figure 4.77 Regional effects of welding seam sizes on stress values

CHAPTER FIVE

CONCLUSIONS

Within the scope of this thesis, in accordance with the project layout an interface has been developed for modeling C Type eccentric press solid model and weld seams on it automatically. By using the developed interface, welded model of C Type eccentric press can be created automatically for the dimensions stated in the interface with SolidWorks 2007. Numerical analyses as part of the project are carried out via using CosmosWorks 2009 by considering the complex geometrical properties of the model and hardware requirements with regard to node number and degree of freedom of the finite element model.

Numerical analyses carried out within the scope of the project are the initial analyses performed by the project partner company concerning the properties of weld seams. The results of numerical analyses, performed for the C type press body on which weld seams are studied, are sufficient for R&D department studies of the project partner company. Furthermore, parametric modeling program developed in the scope of this project is sufficient for R&D department studies of the company as well and have pioneering study value for parametric modeling.

In consequence of the finite element analyses performed in this study for different weld seam dimensions, the following points of discussion and obtained results are summarized:

- Weld seam thickness has low effect on natural frequencies of the press body. Dynamic behaviour of the body is generally affected by the sheet thickness used in body manufacturing and press geometry.
- By increasing the sheet thickness over the body, displacement values on press table are decreased.
- Increase in the weld seam thickness decreases the stress values of weld seams around the crank bearing. The amount of decrease depends on the related weld seam.

- Regional variability of weld seam thicknesses have no considerable effect on displacement values on press table.

Within the scope of this project, strain and displacement measurements on real press machine are carried out and company employees are informed about the measurements. As mentioned in “Machine/Hardware Sharing Agreement” presenting in project contract appendix, strain measurement unit and two laser measurement sensors, one laser control unit with laser displacement system including utility equipment which are bought by project expense item, will be used within the company. Consequently, under favour of the project, the company acquired two significant measurement systems which can be used for R&D activities in the company.

For the purpose of using in scientific researches, two laser sensors, one controller and a hi-tech measurement system including utility equipment, are brought to university by means of this project.

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The Codes of Developed Software for 3-D Parametric Design of C Type Eccentric Press

PROGRAM CODES

Option Explicit

```
Private Sub Command1_MouseMove(Button As Integer, Shift As Integer, X As Single, Y As Single)
    ana_form.Label_yardim.Caption = "Bu Komut İle Değişiklik Yapılan Parametrenin Hangi Parçaları Etkilediği Görülür..."
End Sub
Private Sub Command4_Click()
    Montaj.Show
End Sub
```

```
Private Sub Command4_MouseMove(Button As Integer, Shift As Integer, X As Single, Y As Single)
    ana_form.Label_yardim.Caption = "Bu Komut İle Pres Gövdesine Ait Montaj İşlemi SolidWorks İle Gerçekleştirilir..."
End Sub
```

```
Private Sub Command5_Click()
    Dim cikis_cevap
    cikis_cevap = MsgBox("Program Tarafınızdan Sonlandırılıyor. Çıkmak İstedığınızden Emin misiniz (E/H)?", vbYesNo +
vbQuestion, "Program Sonlandırılıyor")
    If cikis_cevap = vbYes Then
        End
    ElseIf cikis_cevap = vbNo Then
        GoTo 10
    End If
    10
End Sub
```

```
Private Sub Command5_MouseMove(Button As Integer, Shift As Integer, X As Single, Y As Single)
    ana_form.Label_yardim.Caption = "Bu Komut İle Program Sona Erdirilir..."
End Sub
```

Private Sub Form_Load()

```
ana_form.List1.AddItem "Montajı Oluşturan Parça Listesi"
Show
parca_sayac = 0
parca_adedi = 17
```

```
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\MAKINA.JPG")
ana_form.Label2.Caption = ("Öncelikle parçaları oluşturabilmek ve Montaj ortamında kullanabilmek için aşağıda belirtilen yerlere kullanmak istediğiniz değerleri giriniz ve Başlat tuşuna basınız")
```

```
f10 = "D:\Dirinler_Makina A.Ş\GövDe\"
flsw = "C:\Program Files\SolidWorks\lang\english\Tutorial\"
flasmb = "Dirinler_Pres"
```

```
On_Parca.Visible = False
parca_modelleme.Visible = False
```

```
*****
" Tum Degiskenlerin Tanımlanması"*****
```

```
par_adet = 21
deg_adet = 36 ' Max değişken sayısı kaçsa o değer girilir
ReDim degiskenler(par_adet, deg_adet) As String
```

```
' GYS_SAG
degiskenler(1, 1) = "gys_1"
degiskenler(1, 2) = "gys_2"
degiskenler(1, 3) = "gys_3"
degiskenler(1, 4) = "gys_4"
degiskenler(1, 5) = "gys_5"
degiskenler(1, 6) = "gys_6"
degiskenler(1, 7) = "gys_7"
```

```

degiskenler(1, 8) = "gys_9"
degiskenler(1, 9) = "gys_10"
degiskenler(1, 10) = "gys_11"
degiskenler(1, 11) = "gys_12"
degiskenler(1, 12) = "gys_13"
degiskenler(1, 13) = "gys_14"
degiskenler(1, 14) = "gys_15"
degiskenler(1, 15) = "gys_16"
degiskenler(1, 16) = "gys_17"
degiskenler(1, 17) = "r1"
degiskenler(1, 18) = "r2"
degiskenler(1, 19) = "r3"
degiskenler(1, 20) = "r4"
degiskenler(1, 21) = "r5"
degiskenler(1, 22) = "r6"
degiskenler(1, 23) = "ap_parca_kalnlığı"
degiskenler(1, 24) = "eksen_1"
degiskenler(1, 24) = "parca_kalnlığı"

```

'GYS_SOL

```

degiskenler(2, 1) = "gys_1"
degiskenler(2, 2) = "gys_2"
degiskenler(2, 3) = "gys_3"
degiskenler(2, 4) = "gys_4"
degiskenler(2, 5) = "gys_5"
degiskenler(2, 6) = "gys_6"
degiskenler(2, 7) = "gys_7"
degiskenler(2, 8) = "gys_9"
degiskenler(2, 9) = "gys_10"
degiskenler(2, 10) = "gys_11"
degiskenler(2, 11) = "gys_12"
degiskenler(2, 12) = "gys_13"
degiskenler(2, 13) = "gys_14"
degiskenler(2, 14) = "gys_15"
degiskenler(2, 15) = "gys_16"
degiskenler(2, 16) = "gys_17"
degiskenler(2, 17) = "r1"
degiskenler(2, 18) = "r2"
degiskenler(2, 19) = "r3"
degiskenler(2, 20) = "r4"
degiskenler(2, 21) = "r5"
degiskenler(2, 22) = "r6"
degiskenler(2, 23) = "ap_parca_kalnlığı"
degiskenler(2, 24) = "eksen_1"
degiskenler(2, 24) = "parca_kalnlığı"

```

'ALT PLAKA

```

degiskenler(3, 1) = "ap_1"
degiskenler(3, 2) = "ap_3"
degiskenler(3, 3) = "ap_parca_kalnlığı"
degiskenler(3, 4) = "ap_4"
degiskenler(3, 5) = "ap_5"
degiskenler(3, 6) = "ap_6"
degiskenler(3, 7) = "ap_7"
degiskenler(3, 8) = "ap_8"
degiskenler(3, 9) = "ap_parca_genisligi"
degiskenler(3, 10) = "ap_r1"
degiskenler(3, 11) = "ap_r2"
degiskenler(3, 12) = "ap_buyuk_delik_derinligi"
degiskenler(3, 13) = "ap_vida_yaricapi"
degiskenler(3, 14) = "ap_vida_x_koord"
degiskenler(3, 15) = "ap_vida_y_koord"
degiskenler(3, 16) = "mesafe"
degiskenler(3, 17) = "cs_parca_kalnlığı"

```

'C SACI

```

degiskenler(4, 1) = "gys_6"
degiskenler(4, 2) = "gys_5"
degiskenler(4, 3) = "gys_1"
degiskenler(4, 4) = "gys_3"
degiskenler(4, 5) = "r1"
degiskenler(4, 6) = "gys_4"
degiskenler(4, 7) = "r2"

```


degiskenler(4, 8) = "cs_8"
 degiskenler(4, 9) = "gys_2"
 degiskenler(4, 10) = "cs_10"
 degiskenler(4, 11) = "cs_2"
 degiskenler(4, 12) = "cs_1"
 degiskenler(4, 13) = "cs_r1"
 degiskenler(4, 14) = "cs_3"
 degiskenler(4, 15) = "cs_11"
 degiskenler(4, 16) = "cs_12"
 degiskenler(4, 17) = "cs_r6"
 degiskenler(4, 18) = "cs_r2"
 degiskenler(4, 19) = "cs_r3"
 degiskenler(4, 20) = "cs_r5"
 degiskenler(4, 21) = "cs_parca_kalnlığı"
 degiskenler(4, 22) = "mesafe"
 degiskenler(4, 23) = "ct_parca_boyu"
 degiskenler(4, 24) = "ct_parca_kalnlığı"

'AYAK SACI

degiskenler(5, 1) = "as_parca_kalnlığı"
 degiskenler(5, 2) = "as_1"
 degiskenler(5, 3) = "as_2"
 degiskenler(5, 4) = "as_3"
 degiskenler(5, 5) = "as_4"
 degiskenler(5, 6) = "as_5"
 degiskenler(5, 7) = "as_parca_genisligi"
 degiskenler(5, 8) = "as_r"
 degiskenler(5, 9) = "gys_6"
 degiskenler(5, 10) = "gys_5"
 degiskenler(5, 11) = "gys_1"
 degiskenler(5, 12) = "gys_3"
 degiskenler(5, 13) = "r1"
 degiskenler(5, 14) = "gys_4"
 degiskenler(5, 15) = "r2"
 degiskenler(5, 16) = "cs_8"
 degiskenler(5, 17) = "gys_2"
 degiskenler(5, 18) = "cs_10"
 degiskenler(5, 19) = "cs_3"
 degiskenler(5, 20) = "cs_2"
 degiskenler(5, 21) = "cs_1"
 degiskenler(5, 22) = "cs_r1"
 degiskenler(5, 23) = "cygs_KM"
 degiskenler(5, 24) = "cygs_parca_kalnlığı"
 degiskenler(5, 25) = "mesafe"
 degiskenler(5, 26) = "ks_parca_kalnlığı"
 degiskenler(5, 27) = "ct_parca_boyu"
 degiskenler(5, 28) = "ms_parca_kalnlığı"
 degiskenler(5, 29) = "C_Muhafaza_Saci_KM"

'ÖN PANO

degiskenler(6, 1) = "gys_2"
 degiskenler(6, 2) = "mesafe"
 degiskenler(6, 3) = "parca_kalnlığı"
 degiskenler(6, 4) = "ön_p_1"
 degiskenler(6, 5) = "ön_p_2"
 degiskenler(6, 6) = "ön_p_r"
 degiskenler(6, 7) = "ön_p_parca_kalnlığı"

P ye GELEN SAC

degiskenler(7, 1) = "gys_4"
 degiskenler(7, 2) = "as_3"
 degiskenler(7, 3) = "gys_1"
 degiskenler(7, 4) = "gys_3"
 degiskenler(7, 5) = "r1"
 degiskenler(7, 6) = "as_parca_kalnlığı"
 degiskenler(7, 7) = "mesafe"
 degiskenler(7, 8) = "cs_2"
 degiskenler(7, 9) = "cs_1"
 degiskenler(7, 10) = "cs_r1"
 degiskenler(7, 11) = "cs_8"
 degiskenler(7, 12) = "gys_2"
 degiskenler(7, 13) = "cs_10"
 degiskenler(7, 14) = "cs_parca_kalnlığı"

'KIZAK SACI

degiskenler(8, 1) = "gys_9"
degiskenler(8, 2) = "öyms_parca_kalnlığı"
degiskenler(8, 3) = "X_1"
degiskenler(8, 4) = "X_3"
degiskenler(8, 5) = "X_5"
degiskenler(8, 6) = "no_4s_parca_kalnlığı"
degiskenler(8, 7) = "X_6"
degiskenler(8, 8) = "no_5s_parca_kalnlığı"
degiskenler(8, 9) = "X_7"
degiskenler(8, 10) = "gys_1"
degiskenler(8, 11) = "gys_3"
degiskenler(8, 12) = "r1"
degiskenler(8, 13) = "cs_8"
degiskenler(8, 14) = "gys_2"
degiskenler(8, 15) = "cs_10"
degiskenler(8, 16) = "cs_parca_kalnlığı"
degiskenler(8, 17) = "cs_1"
degiskenler(8, 18) = "gys_5"
degiskenler(8, 19) = "eksen_1"
degiskenler(8, 20) = "ap_parca_kalnlığı"
degiskenler(8, 21) = "mesafe"
degiskenler(8, 22) = "oy_r2"
degiskenler(8, 23) = "ks_parca_kalnlığı"
degiskenler(8, 24) = "gys_10"
degiskenler(8, 25) = "üst_girinti"
degiskenler(8, 26) = "No_1_Parca_kalnlığı"
degiskenler(8, 27) = "No_2_Parca_kalnlığı"
degiskenler(8, 28) = "No_3_Parca_kalnlığı"
degiskenler(8, 29) = "uzaklık"
degiskenler(8, 30) = "ay_r2"
degiskenler(8, 31) = "No1_Parca_kalnlığı"

'ÖN YATAK

degiskenler(9, 1) = "gys_10"
degiskenler(9, 2) = "mesafe"
degiskenler(9, 3) = "gys_2"
degiskenler(9, 4) = "ap_parca_kalnlığı"
degiskenler(9, 5) = "eksen_1"
degiskenler(9, 6) = "ön_yatak_r"
degiskenler(9, 7) = "ön_yatak_parca_kalnlığı"
degiskenler(9, 8) = "eksen_3"

'ÖN YATAK MESAFE SACI

degiskenler(10, 1) = "gys_10"
degiskenler(10, 2) = "mesafe"
degiskenler(10, 3) = "gys_2"
degiskenler(10, 4) = "ap_parca_kalnlığı"
degiskenler(10, 5) = "eksen_1"
degiskenler(10, 6) = "öyms_1"
degiskenler(10, 7) = "öyms_2"
degiskenler(10, 8) = "öyms_r"
degiskenler(10, 9) = "öyms_parca_kalnlığı"
degiskenler(10, 10) = "gys_5"
degiskenler(10, 11) = "eksen_3"

'İKİ C ARASI MUHAFAZA SACI

degiskenler(11, 1) = "ms_parca_kalnlığı"
degiskenler(11, 2) = "ms_1"
degiskenler(11, 3) = "gys_1"
degiskenler(11, 4) = "gys_3"
degiskenler(11, 5) = "r1"
degiskenler(11, 6) = "cs_8"
degiskenler(11, 7) = "gys_2"
degiskenler(11, 8) = "cs_10"
degiskenler(11, 9) = "gys_6"
degiskenler(11, 10) = "gys_5"
degiskenler(11, 11) = "gys_4"
degiskenler(11, 12) = "r2"
degiskenler(11, 13) = "cs_2"
degiskenler(11, 14) = "cs_3"
degiskenler(11, 15) = "ks_parca_kalnlığı"

```

degiskenler(11, 16) = "ct_parca_boyu"
degiskenler(11, 17) = "cs_parca_kalnlığı"
degiskenler(11, 18) = "mesafe"
degiskenler(11, 19) = "ms_3"
degiskenler(11, 20) = "ms_r"
degiskenler(11, 21) = "C_Muhafaza_Sacı_KM"
'C YE GELEN SAC
degiskenler(12, 1) = "gys_6"
degiskenler(12, 2) = "gys_5"
degiskenler(12, 3) = "gys_1"
degiskenler(12, 4) = "gys_3"
degiskenler(12, 5) = "r1"
degiskenler(12, 6) = "gys_4"
degiskenler(12, 7) = "r2"
degiskenler(12, 8) = "cs_8"
degiskenler(12, 9) = "gys_2"
degiskenler(12, 10) = "cs_10"
degiskenler(12, 11) = "cs_3"
degiskenler(12, 12) = "cs_2"
degiskenler(12, 13) = "cs_1"
degiskenler(12, 14) = "cs_r1"
degiskenler(12, 15) = "cygs_KM"
degiskenler(12, 16) = "cygs_parca_kalnlığı"
degiskenler(12, 17) = "mesafe"
degiskenler(12, 18) = "ks_parca_kalnlığı"
degiskenler(12, 19) = "ct_parca_boyu"
degiskenler(12, 20) = "ms_parca_kalnlığı"
degiskenler(12, 21) = "as_1"
degiskenler(12, 22) = "as_2"
degiskenler(12, 23) = "cygs_ms_boyu"
degiskenler(12, 24) = "cs_parca_kalnlığı"
degiskenler(12, 25) = "as_parca_kalnlığı"
degiskenler(12, 26) = "C_Muhafaza_Sacı_KM"

```

'NO 5 SACI

```

degiskenler(13, 1) = "as_parca_kalnlığı"
degiskenler(13, 2) = "as_1"
degiskenler(13, 3) = "as_2"
degiskenler(13, 4) = "no_4s_parca_kalnlığı"
degiskenler(13, 5) = "X_7"
degiskenler(13, 6) = "cygs_parca_kalnlığı"
degiskenler(13, 7) = "gys_14"
degiskenler(13, 8) = "mesafe"
degiskenler(13, 9) = "eksen_1"
degiskenler(13, 10) = "gys_2"
degiskenler(13, 11) = "ap_parca_kalnlığı"
degiskenler(13, 12) = "eksen_2"
degiskenler(13, 13) = "ayds_r1"
degiskenler(13, 14) = "no_5s_parca_kalnlığı"
degiskenler(13, 15) = "gys_6"
degiskenler(13, 16) = "gys_5"
degiskenler(13, 17) = "gys_1"
degiskenler(13, 18) = "gys_3"
degiskenler(13, 19) = "r1"
degiskenler(13, 20) = "gys_4"
degiskenler(13, 21) = "r2"
degiskenler(13, 22) = "cs_8"
degiskenler(13, 23) = "cs_10"
degiskenler(13, 24) = "cs_2"
degiskenler(13, 25) = "cs_1"
degiskenler(13, 26) = "cs_r1"
degiskenler(13, 27) = "cs_3"
degiskenler(13, 28) = "ks_parca_kalnlığı"
degiskenler(13, 29) = "ct_parca_boyu"
degiskenler(13, 30) = "ms_parca_kalnlığı"
degiskenler(13, 31) = "cygs_KM"
degiskenler(13, 32) = "cygs_ms_boyu"
degiskenler(13, 33) = "C_Muhafaza_Sacı_KM"

```

'YAN KAPAK

```

degiskenler(14, 1) = "yk_1"
degiskenler(14, 2) = "yk_2"
degiskenler(14, 3) = "yk_3"

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degiskenler(14, 4) = "yk_4"
degiskenler(14, 5) = "yk_parca_kalnlığı"
degiskenler(14, 6) = "gys_13"
degiskenler(14, 7) = "mesafe"
degiskenler(14, 8) = "X_6"
degiskenler(14, 9) = "X_7"
degiskenler(14, 10) = "no_5s_parca_kalnlığı"
degiskenler(14, 11) = "gys_11"
degiskenler(14, 12) = "girinti"
degiskenler(14, 13) = "No1_parca_kalnlığı"
degiskenler(14, 14) = "parca_kalnlığı"
degiskenler(14, 15) = "gys_10"
degiskenler(14, 16) = "gys_14"
degiskenler(14, 17) = "üst_girinti"
' KRANK ORTA YATAK
degiskenler(15, 1) = "oy_r1"
degiskenler(15, 2) = "oy_r2"
degiskenler(15, 3) = "oy_boy"
' KRANK ARKA YATAK
degiskenler(16, 1) = "ay_r1"
degiskenler(16, 2) = "ay_r2"
degiskenler(16, 3) = "ay_boy"

' ARKA YATAK DESTEK SACI
degiskenler(17, 1) = "ayds_r1"
degiskenler(17, 2) = "ayds_r2"
degiskenler(17, 3) = "ayds_boy"

' ÖN YATAK DESTEK SACI
degiskenler(18, 1) = "öyds_r1"
degiskenler(18, 2) = "öyds_r2"
degiskenler(18, 3) = "öyds_boy"
NO 4 SACI
degiskenler(19, 1) = "as_parca_kalnlığı"
degiskenler(19, 2) = "as_1"
degiskenler(19, 3) = "as_2"
degiskenler(19, 4) = "no_4s_parca_kalnlığı"
degiskenler(19, 5) = "X_7"
degiskenler(19, 6) = "cygs_parca_kalnlığı"
degiskenler(19, 7) = "X_6"
degiskenler(19, 8) = "no_5s_parca_kalnlığı"
degiskenler(19, 9) = "gys_10"
degiskenler(19, 10) = "üst_girinti"
degiskenler(19, 11) = "mesafe"
degiskenler(19, 12) = "eksen_1"
degiskenler(19, 13) = "gys_2"
degiskenler(19, 14) = "ap_parca_kalnlığı"
degiskenler(19, 15) = "gys_6"
degiskenler(19, 16) = "gys_5"
degiskenler(19, 17) = "gys_1"
degiskenler(19, 18) = "gys_3"
degiskenler(19, 19) = "r1"
degiskenler(19, 20) = "gys_4"
degiskenler(19, 21) = "r2"
degiskenler(19, 22) = "cs_8"
degiskenler(19, 23) = "cs_10"
degiskenler(19, 24) = "cs_2"
degiskenler(19, 25) = "cs_1"
degiskenler(19, 26) = "cs_r1"
degiskenler(19, 27) = "cs_3"
degiskenler(19, 28) = "ks_parca_kalnlığı"
degiskenler(19, 29) = "ct_parca_boyu"
degiskenler(19, 30) = "ms_parca_kalnlığı"
degiskenler(19, 31) = "cygs_KM"
degiskenler(19, 32) = "cygs_ms_boyu"
degiskenler(19, 33) = "ay_r2"
degiskenler(19, 34) = "öyds_r1"
degiskenler(19, 35) = "eksen_2"
degiskenler(19, 36) = "C_Muhafaza_Sacı_KM"

' BURC DESTEĞİ
' BURC FEDERİ
degiskenler(20, 1) = "bf_1"

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degiskenler(20, 2) = "bf_2"
degiskenler(20, 3) = "bf_3"
degiskenler(20, 4) = "bf_4"
degiskenler(20, 5) = "bf_parca_kalınligı"
' BURC TAKVİYESİ
degiskenler(21, 1) = "bt_1"
degiskenler(21, 2) = "bt_2"
degiskenler(21, 3) = "bt_3"
degiskenler(21, 4) = "bt_parca_kalınligı"

' KAYNAK DİKİSİ DEFAULT DEGERLERİ

Open "D:\Dirinler_Makina A.Ş\GövDe\Kaynak_Dikis_Degerleri.txt" For Input As 1

Input #1, OKUD
Input #1, OKAD
Input #1, AKOD
Input #1, AKAD
Input #1, N4SD
Input #1, CGSD
Input #1, PGSD
Input #1, GYSN3N4AD
Input #1, CSKSAD
Close #1

Kaynak_Dikis_Boyutlari.Text1.Text = Str(OKUD)
Kaynak_Dikis_Boyutlari.Text2.Text = Str(OKAD)
Kaynak_Dikis_Boyutlari.Text3.Text = Str(AKOD)
Kaynak_Dikis_Boyutlari.Text4.Text = Str(AKAD)
Kaynak_Dikis_Boyutlari.Text5.Text = Str(N4SD)
Kaynak_Dikis_Boyutlari.Text6.Text = Str(CGSD)
Kaynak_Dikis_Boyutlari.Text7.Text = Str(PGSD)
Kaynak_Dikis_Boyutlari.Text8.Text = Str(GYSN3N4AD)
Kaynak_Dikis_Boyutlari.Text9.Text = Str(CSKSAD)

End Sub

Private Sub Form_MouseMove(Button As Integer, Shift As Integer, X As Single, Y As Single)
ana_form.Label_yardim.Caption = ""
End Sub

Private Sub gys_sag_Click()
gys_sag1.Show
End Sub

Private Sub gys_sol_Click()
gys_sol1.Show
'ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\Parametrik_Parcalar\gys_solx.jpg")
End Sub

Private Sub Altplk_Click()
Alt_Plaka.Show
End Sub

Private Sub mnu_ayaksacı_Click()
Ayak_Sacı.Show

End Sub

Private Sub mnu_ayds_Click()
Yataklar.Show
Yataklar.Frame1.Enabled = False
Yataklar.Frame2.Enabled = False
Yataklar.Frame3.Enabled = False
Yataklar.Frame4.Enabled = True
End Sub

Private Sub mnu_burcfederi_Click()
Burc_Federi_ve_Takviyesi.Show
Burc_Federi_ve_Takviyesi.Frame1.Enabled = False
Burc_Federi_ve_Takviyesi.Frame3.Enabled = True

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```
End Sub

Private Sub mnu_burctakviyesi_Click()
Burc_Federi_ve_Takviyesi.Show
Burc_Federi_ve_Takviyesi.Frame1.Enabled = True
Burc_Federi_ve_Takviyesi.Frame3.Enabled = False
End Sub

Private Sub mnu_csacı_Click()
C_sacı.Show
End Sub

Private Sub mnu_cyegelen_Click()
C_ye_gelen_sac.Show

End Sub

Private Sub mnu_kaynak_Click()
C_Sacı_Kaynak_Dikisi.Show
End Sub

Private Sub mnu_kaynak_dikisleri_Click()
Kaynak_Dikis_Boyutlari.Show
End Sub

Private Sub mnu_kızaksacı_Click()
Kızak_Sacı.Show
End Sub

Private Sub mnu_krankarkayatak_Click()
Yataklar.Show
Yataklar.Frame1.Enabled = False
Yataklar.Frame2.Enabled = False
Yataklar.Frame3.Enabled = True
Yataklar.Frame4.Enabled = False
End Sub

Private Sub mnu_krankortayatak_Click()
Yataklar.Show
Yataklar.Frame1.Enabled = True
Yataklar.Frame2.Enabled = False
Yataklar.Frame3.Enabled = False
Yataklar.Frame4.Enabled = False

End Sub

Private Sub mnu_muhafzasacı_Click()
Muhafaza_Sacı.Show

End Sub

Private Sub mnu_no4sacı_Click()
No_4_Sacı.Show

End Sub

Private Sub mnu_no5_Click()
No_5_Sacı.Show
End Sub

Private Sub mnu_önpano_Click()
Ön_Pano.Show

End Sub

Private Sub mnu_önyatak_Click()
Ön_Yatak.Show

End Sub

Private Sub mnu_önyatakmesafesacı_Click()
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Ön_Yatak_Mesafe_Saci.Show

End Sub

Private Sub mnu_öyds_Click()
Yataklar.Show
Yataklar.Frame1.Enabled = False
Yataklar.Frame2.Enabled = True
Yataklar.Frame3.Enabled = False
Yataklar.Frame4.Enabled = False
End Sub

Private Sub mnu_pyegelen_Click()
P_ye_gelen_sac.Show

End Sub

Private Sub mnu_yankapak_Click()
Yan_Kapak.Show

End Sub

Private Sub On_Parca_Click()
Dim xx As Long

parca_sayac = parca_sayac - 1

If parca_sayac > 0 And parca_sayac < parca_adedi Then
Son_Parca.Caption = "Sonraki Parça"

ElseIf parca_sayac > parca_adedi Then
xx = MsgBox("Başka Parça Yoktur", vbExclamation, "Parça Sayısı Hatası")
End If

Select Case parca_sayac

Case Is = 1
Son_Parca.Visible = True
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_GYS_SOL.JPG")
On_Parca.Visible = False
parca_etiketi.Caption = "*** GÖVDE YAN SACI (SAG) ***"
Case Is = 2
Son_Parca.Visible = True
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_GYS_SOL.JPG")
On_Parca.Visible = True
parca_etiketi.Caption = "*** GÖVDE YAN SACI (SOL) ***"
Case Is = 3
Son_Parca.Visible = True
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ALT_PLAKA.JPG")
On_Parca.Visible = True
parca_etiketi.Caption = "*** ALT PLAKA ***"
Case Is = 4
Son_Parca.Visible = True
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_C_SACI.JPG")
On_Parca.Visible = True
parca_etiketi.Caption = "*** C SACI ***"
Case Is = 5
Son_Parca.Visible = True
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_AYAK_SACI.JPG")
On_Parca.Visible = True
parca_etiketi.Caption = "*** AYAK SACI ***"
Case Is = 6
Son_Parca.Visible = True
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ON_PANO.JPG")
On_Parca.Visible = True
parca_etiketi.Caption = "*** ÖN PANO ***"
Case Is = 7
Son_Parca.Visible = True
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_P_YE_GELEN_SAC.JPG")
On_Parca.Visible = True
parca_etiketi.Caption = "*** P'YE GELEN SAC ***"

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Case Is = 8
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_KIZAK_SACI.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** KIZAK SACI ***"
Case Is = 9
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ÖN_YATAK.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** ÖN YATAK ***"
Case Is = 10
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ÖN_YATAK_MESAFE_SACI.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** ÖN YATAK MESAFE SACI ***"
Case Is = 11
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_MUHAFAZA_SACI.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** İKI C ARASI MUHAFAZA SACI ***"
Case Is = 12
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_C_YE_GELEN_SAC.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** C YE GELEN SAC ***"
Case Is = 13
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_NO_5S.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** NO 5 SACI ***"
Case Is = 14
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_YAN_KAPAK.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** YAN KAPAK ***"
Case Is = 15
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_YATAKLAR.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** YATAKLAR ***"
Case Is = 16
  Son_Parca.Visible = True
  ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_NO_4S.JPG")
  On_Parca.Visible = True
  parca_etiketi.Caption = "*** NO 4 SACI ***"

End Select
End Sub

Private Sub On_Parca_MouseMove(Button As Integer, Shift As Integer, X As Single, Y As Single)
  ana_form.Label_yardim.Caption = "Bu Komut İle Bir Önceki Parçaya Dönülür..."
End Sub

Private Sub parca_modelleme_Click()

  Select Case parca_sayac

  Case Is = 1
    gys_sag1.Show
  Case Is = 2
    gys_sol1.Show
  Case Is = 3
    Alt_Plaka.Show
  Case Is = 4
    C_saci.Show
  Case Is = 5
    Ayak_Saci.Show
  Case Is = 6
    Ön_Pano.Show
  Case Is = 7
    P_ye_gelen_sac.Show
  Case Is = 8
    Kizak_Saci.Show

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Case Is = 9
  Ön_Yatak.Show
Case Is = 10
  Ön_Yatak_Mesafe_Saci.Show
Case Is = 11
  Muhafaza_Saci.Show
Case Is = 12
  C_ye_gelen_sac.Show
Case Is = 13
  No_5_Saci.Show
  Case Is = 14
  Yan_Kapak.Show
Case Is = 15
  Yataklar.Show
Case Is = 16
  No_4_Saci.Show
Case Is = 17
  Burc_Federasyonu_Takviyesi.Show

End Select

End Sub

Private Sub parca_modelleme_MouseMove(Button As Integer, Shift As Integer, X As Single, Y As Single)
  ana_form.Label_yardim.Caption = "Bu Komut Resimdeki Parça SolidWorks İle Modellenir..."
End Sub

Private Sub Son_Parca_Click()
  Dim xx As Long

  parca_sayac = parca_sayac + 1

  If parca_sayac > 0 And parca_sayac < parca_adedi Then
    ana_form.Label2.Caption = ""
  End If

  Select Case parca_sayac

  Case Is = 1
    Son_Parca.Visible = True
    ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_GYS_SOL.JPG")
    parca_modelleme.Visible = True
    parca_etiketi.Caption = "*** GÖVDE YAN SACI (SAG) ***"
  Case Is = 2
    Son_Parca.Visible = True
    ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_GYS_SOL.JPG")
    On_Parca.Visible = True
    parca_modelleme.Visible = True
    parca_etiketi.Caption = "*** GÖVDE YAN SACI (SOL) ***"
  Case Is = 3
    Son_Parca.Visible = True
    ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ALT_PLAKA.JPG")
    On_Parca.Visible = True
    parca_modelleme.Visible = True
    parca_etiketi.Caption = "*** ALT PLAKA ***"
  Case Is = 4
    Son_Parca.Visible = True
    ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_C_SACI.JPG")
    On_Parca.Visible = True
    parca_modelleme.Visible = True
    parca_etiketi.Caption = "*** C SACI ***"
  Case Is = 5
    Son_Parca.Visible = True
    ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_AYAK_SACI.JPG")
    On_Parca.Visible = True
    parca_modelleme.Visible = True
    parca_etiketi.Caption = "*** AYAK SACI ***"
  Case Is = 6
    Son_Parca.Visible = True
    ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ON_PANO.JPG")
    On_Parca.Visible = True

```

```

parca_modelleme.Visible = True
parca_etiketi.Caption = "*** ÖN PANO ***"
Case Is = 7
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_P_YE_GELEN_SAC.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** P'YE GELEN SAC ***"
Son_Parca.Visible = True
Case Is = 8
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_KIZAK_SACI.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** KIZAK SACI ***"
Son_Parca.Visible = True
Case Is = 9
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ÖN_YATAK.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** öN YATAK ***"
Son_Parca.Visible = True
Case Is = 10
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_ÖN_YATAK_MESAFE_SACI.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** ÖN YATAK MESAFE SACI ***"
Son_Parca.Visible = True
Case Is = 11
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_MUHAFAZA_SACI.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** İKİ C ARASI MUHAFAZA SACI ***"
Son_Parca.Visible = True
Case Is = 12
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_C_YE_GELEN_SAC.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** C YE GELEN SAC ***"
Son_Parca.Visible = True
Case Is = 13
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_NO_5S.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** NO 5 SACI ***"
Son_Parca.Visible = True
Case Is = 14
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_YAN_KAPAK.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** YAN KAPAK ***"
Son_Parca.Visible = True
Case Is = 15
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_YATAKLAR.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** YATAKLAR ***"
Son_Parca.Visible = True
Case Is = 16
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_NO_4S.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** NO 4 SACI ***"
Son_Parca.Visible = True
Case Is = 17
ana_form.Picture1.Picture = LoadPicture("D:\Dirinler_Makina A.Ş\GövDe\A_BURC_FT.JPG")
On_Parca.Visible = True
parca_modelleme.Visible = True
parca_etiketi.Caption = "*** BURC FEDERİ VE TAKVİYESİ ***"
Son_Parca.Visible = False

```

End Select

End Sub

```
Private Sub Son_Parca_MouseMove(Button As Integer, Shift As Integer, X As Single, Y As Single)
    ana_form.Label_yardim.Caption = "Bu Komut İle Bir Sonraki Parçaya Geçilir..."
End Sub
```

Option Explicit

```
Private Sub Combo1_Click()
    parca_listesi.Visible = False
    parca_listesi.Clear
```

End Sub

```
Rem Public ap_1, ap_parca_kalınligı, ap_3, ap_4, ap_5, ap_6, ap_7, ap_8, ap_parca_genisligi, ap_r1, ap_r2,
ap_buyuk_delik_derinligi, ap_vida_yaricapi, ap_vida_x_koord, ap_vida_y_koord As Double
```

```
Private Sub Command1_Click()
```

Call VeriOkuma

```
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc
```

Call snap

```
Dim msg As String
'ap_parca_genisligi' ni asagıda oldugu gibi diger parametrelere baglı ahale getirelim mi ?
'ap_parca_genisligi = mesafe + parca_kalınligı * 2 + 0.105 * 2
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.CreateLine2 0, 0, 0, ap_1, 0, 0
part.ViewZoomtofit2
part.CreateLine2 ap_1, 0, 0, ap_1, ap_parca_kalınligı, 0
part.CreateLine2 ap_1, ap_parca_kalınligı, 0, (ap_1 - ap_3), ap_parca_kalınligı, 0
part.CreateLine2 (ap_1 - ap_3), ap_parca_kalınligı, 0, (ap_1 - ap_3), (ap_parca_kalınligı - ap_4), 0
part.CreateLine2 (ap_1 - ap_3), (ap_parca_kalınligı - ap_4), 0, ((ap_1 - ap_3) + ((ap_6 - ap_7) / 2)), (ap_parca_kalınligı - ap_4),
0
part.CreateLine2 ((ap_1 - ap_3) + ((ap_6 - ap_7) / 2)), (ap_parca_kalınligı - ap_4), 0, ((ap_1 - ap_3) + ((ap_6 - ap_7) / 2)),
(ap_parca_kalınligı - ap_5 - ap_4), 0
part.CreateLine2 ((ap_1 - ap_3) + ((ap_6 - ap_7) / 2)), (ap_parca_kalınligı - ap_5 - ap_4), 0, (((ap_1 - ap_3) + ((ap_6 - ap_7) /
2)) - ap_6), (ap_parca_kalınligı - ap_5 - ap_4), 0
part.CreateLine2 (((ap_1 - ap_3) + ((ap_6 - ap_7) / 2)) - ap_6), (ap_parca_kalınligı - ap_5 - ap_4), 0, (((ap_1 - ap_3) + ((ap_6 -
ap_7) / 2)) - ap_6), (ap_parca_kalınligı - ap_4), 0
part.CreateLine2 (((ap_1 - ap_3) + ((ap_6 - ap_7) / 2)) - ap_6), (ap_parca_kalınligı - ap_4), 0, (ap_1 - ap_3 - ap_7),
(ap_parca_kalınligı - ap_4), 0
part.CreateLine2 (ap_1 - ap_3 - ap_7), (ap_parca_kalınligı - ap_4), 0, (ap_1 - ap_3 - ap_7), ap_parca_kalınligı, 0
part.CreateLine2 (ap_1 - ap_3 - ap_7), ap_parca_kalınligı, 0, (ap_1 - ap_3 - ap_7 - ap_8), ap_parca_kalınligı, 0
part.CreateLine2 (ap_1 - ap_3 - ap_7 - ap_8), ap_parca_kalınligı, 0, (ap_1 - ap_3 - ap_7 - ap_8), (ap_parca_kalınligı - ap_4), 0
part.CreateLine2 (ap_1 - ap_3 - ap_7 - ap_8), (ap_parca_kalınligı - ap_4), 0, ((ap_1 - ap_3 - ap_7 - ap_8) + (ap_6 - ap_7) / 2),
(ap_parca_kalınligı - ap_4), 0
part.CreateLine2 ((ap_1 - ap_3 - ap_7 - ap_8) + (ap_6 - ap_7) / 2), (ap_parca_kalınligı - ap_4), 0, ((ap_1 - ap_3 - ap_7 - ap_8) +
(ap_6 - ap_7) / 2), (ap_parca_kalınligı - ap_5 - ap_4), 0
part.CreateLine2 ((ap_1 - ap_3 - ap_7 - ap_8) + (ap_6 - ap_7) / 2), (ap_parca_kalınligı - ap_5 - ap_4), 0, (((ap_1 - ap_3 - ap_7 -
ap_8) + (ap_6 - ap_7) / 2) - ap_6), (ap_parca_kalınligı - ap_5 - ap_4), 0
part.CreateLine2 (((ap_1 - ap_3 - ap_7 - ap_8) + (ap_6 - ap_7) / 2) - ap_6), (ap_parca_kalınligı - ap_5 - ap_4), 0, (((ap_1 - ap_3
- ap_7 - ap_8) + (ap_6 - ap_7) / 2) - ap_6), (ap_parca_kalınligı - ap_4), 0
part.CreateLine2 (((ap_1 - ap_3 - ap_7 - ap_8) + (ap_6 - ap_7) / 2) - ap_6), (ap_parca_kalınligı - ap_4), 0, ap_3,
(ap_parca_kalınligı - ap_4), 0
part.CreateLine2 ap_3, (ap_parca_kalınligı - ap_4), 0, ap_3, ap_parca_kalınligı, 0
part.CreateLine2 ap_3, ap_parca_kalınligı, 0, 0, ap_parca_kalınligı, 0
part.CreateLine2 0, ap_parca_kalınligı, 0, 0, 0, 0
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, ap_parca_genisligi, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
```

```

Call part.Extension.SelectByID2("", "FACE", ((ap_1) / 9), (ap_parca_genisligi / 10), ap_parca_kalnlığı, True, 0, Nothing, 0)
Call part.Extension.SelectByID2("", "FACE", ((ap_1) / 2), (ap_parca_genisligi / 10), ap_parca_kalnlığı, True, 0, Nothing, 0)
Call part.Extension.SelectByID2("", "FACE", ((ap_1) * 8 / 9), (ap_parca_genisligi / 10), ap_parca_kalnlığı, True, 0, Nothing, 0)

```

```

' F2 Açıklamasında ModelDoc2----FeatureChamfer yazıyor, (metre cinsinde genişlik, radyancinsinden açı,flip)
part.FeatureChamfer 0.003, (3.141592654 / 4), 0

```

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

```

```

part.CreateCircleByRadius2 (ap_1 / 2), -(ap_parca_genisligi / 2), 0, ap_r1
boolstatus = part.Extension.SelectByID2("Arc1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, ap_parca_kalnlığı, 0, False, False, False, False, 0, 0, False, False,
False, False, False, True, True
part.SelectionManager.EnableContourSelection = 0

```

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("", "FACE", (ap_1 / 2), ap_parca_kalnlığı, (ap_parca_genisligi / 3), True, 0, Nothing, 0)

```

```

part.CreateCircleByRadius2 (ap_1 / 2), -(ap_parca_genisligi / 2), 0, ap_r2
boolstatus = part.Extension.SelectByID2("Arc2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, False, 0, 0, ap_buyuk_delik_derinligi, 0, False, False, False, False, 0, 0, False,
False, False, False, True, True
part.SelectionManager.EnableContourSelection = 0

```

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("", "FACE", (ap_1 / 2), ap_parca_kalnlığı, (ap_parca_genisligi / 3), True, 0, Nothing, 0)

```

```

part.CreatePoint ap_1 / 2 - ap_vida_x_koord, (ap_vida_y_koord - ap_parca_genisligi / 2), 0
part.CreatePoint ap_1 / 2 + ap_vida_x_koord, (ap_vida_y_koord - ap_parca_genisligi / 2), 0
part.CreatePoint ap_1 / 2 - ap_vida_x_koord, -(ap_vida_y_koord + ap_parca_genisligi / 2), 0
part.CreatePoint ap_1 / 2 + ap_vida_x_koord, -(ap_vida_y_koord + ap_parca_genisligi / 2), 0
part.CreateCircleByRadius2 ap_1 / 2 - ap_vida_x_koord, (ap_vida_y_koord - ap_parca_genisligi / 2), 0, ap_vida_yaricapi
part.CreateCircleByRadius2 ap_1 / 2 + ap_vida_x_koord, (ap_vida_y_koord - ap_parca_genisligi / 2), 0, ap_vida_yaricapi
part.CreateCircleByRadius2 ap_1 / 2 - ap_vida_x_koord, -(ap_vida_y_koord + ap_parca_genisligi / 2), 0, ap_vida_yaricapi
part.CreateCircleByRadius2 ap_1 / 2 + ap_vida_x_koord, -(ap_vida_y_koord + ap_parca_genisligi / 2), 0, ap_vida_yaricapi
boolstatus = part.Extension.SelectByID2("Sketch4", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, False, 0, 0, 0.05, 0, False, False, False, False, 0, 0, False, False, False,
False, True, True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", ap_1 / 2, ap_parca_kalnlığı / 2, ap_parca_genisligi, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
boolstatus = part.EditRebuild3

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", ap_1 / 2, ap_parca_kalnlığı / 2, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", 0, ap_parca_kalnlığı / 2, ap_parca_genisligi / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", 0.002, 0, 0.002, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", 0.002, 0, 0.002, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = Part.EditRebuild3
part.SaveAs2 fl0 + "Alt Plaka.SLDPRT", 0, False, False

```

```

' PLAKA TAKVİYESİ NİN OLUSTURULMASI ARADAKİ MESAFEYE BAGLI OLARAK

```

```

Set swApp = GetObject("sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

```

'Dim pt_parca_kalınligı As Double
'pt_parca_kalınligı = 0.03

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, (mesafe - 2 * cs_parca_kalınligı), pt_boyu, 0, 0
part.ClearSelection2 True

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, pt_parca_kalınligı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

part.SaveAs2 fl0 + "Plaka Takviyesi.SLDPRT", 0, False, False
End Sub

Private Sub Command2_Click()
Alt_Plaka.Hide
End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
parca_listesi.AddItem "Birc Federi"
Case Is = 21

```

```

        parca_listesi.AddItem "Birc Takviyesi"
    'Case Else
        ' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

    End Select
End If
Next j
Next i
End Sub

Private Sub Form_Load()
Show
fl0 = "D:\Dirinler_Makina A.Ş\GövDe\"
flsw = "C:\Program Files\SolidWorks\lang\english\Tutorial\"

Combo1.AddItem "ap_1"
Combo1.AddItem "ap_3"
Combo1.AddItem "ap_parca_kalınligı"
Combo1.AddItem "ap_4"
Combo1.AddItem "ap_5"
Combo1.AddItem "ap_6"
Combo1.AddItem "ap_7"
Combo1.AddItem "ap_8"
Combo1.AddItem "ap_parca_genisligi"
Combo1.AddItem "ap_r1"
Combo1.AddItem "ap_r2"
Combo1.AddItem "ap_buyuk_delik_derinligi"
Combo1.AddItem "ap_vida_yaricapi"
Combo1.AddItem "ap_vida_x_koord"
Combo1.AddItem "ap_vida_y_koord"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "cs_parca_kalınligı"

parca_listesi.Visible = False

End Sub

Private Sub mnu_dosya_alt_plaka_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "ap_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear

End Sub

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc
Call snap

Dim msg As String

as_parca_genisligi = mesafe

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

.....

'C ye gelen sac ın geometrisi nde ki küçük bir problemden dolayı, parçalar birleşmemektedir.
'Dolayısı ile yeni bir Cozum yolu kullanılacaktır.
'Parca, c ye gelen sac profiline uygun olarak kesilecektir, bundan dolayı asagıda ki Go to Komutu ile Daha önce kullanılan
yöntem es geçilecektir.

```

```

.....
GoTo 68
part.CreateLine2 0, 0, 0, as_parca_kalnlığı, 0, 0
part.ViewZoomtofit2
part.CreateLine2 as_parca_kalnlığı, 0, 0, as_parca_kalnlığı, as_1, 0
part.CreateLine2 as_parca_kalnlığı, as_1, 0, 0, as_2, 0
part.CreateLine2 0, as_2, 0, 0, 0, 0
68
part.CreateLine2 0, 0, 0, as_parca_kalnlığı, 0, 0
part.ViewZoomtofit2
part.CreateLine2 as_parca_kalnlığı, 0, 0, as_parca_kalnlığı, as_1 + (as_parca_kalnlığı / 2), 0
part.CreateLine2 as_parca_kalnlığı, as_1 + (as_parca_kalnlığı / 2), 0, 0, as_1 + (as_parca_kalnlığı / 2), 0
part.CreateLine2 0, as_1 + (as_parca_kalnlığı / 2), 0, 0, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, as_parca_genisligi, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle -((as_parca_genisligi - as_4) / 2), as_3, 0, -(((as_parca_genisligi - as_4) / 2) + as_4), as_3 + as_5, 0, 0
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Point4", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 as_r, 2

boolstatus = part.Extension.SelectByID2("Point2", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 as_r, 2

boolstatus = part.Extension.SelectByID2("Point3", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 as_r, 2

boolstatus = part.Extension.SelectByID2("Point1", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 as_r, 2

part.FeatureManager.FeatureCut True, False, True, 0, 0, as_parca_kalnlığı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1

boolstatus = part.Extension.SelectByID2("", "FACE", as_parca_kalnlığı / 2, as_3, as_parca_genisligi / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 as_parca_kalnlığı, False, True
part.ClearSelection2 True

.....

Call C_yegelen_Rutin

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

swApp.SetUserPreferenceToggle swSketchInference, False

part.CreateLine2 0, as_2, 0, 0, (as_2 + cygs_m), 0
part.ViewZoomtofit2

If ct_x < (pi / 2) Then
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r,
0
part.ViewZoomtofit2

```

```

part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalınligı ^ 2) - cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) - cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0,
0, as_2, 0
Else
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0,
0, as_2, 0
End If
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.SketchAddConstraints "sgVERTICAL2D" ' Seçili olan çizgiye Diklik özelliği atanmıştır
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 1)
part.SketchAddConstraints "sgPARALLEL" ' Seçili olan iki çizginin birbirlerine paralel olma özelliği atanmıştır.
part.ClearSelection2 True

part.FeatureManager.FeatureCut True, False, True, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0 ' Aktif olan Sketch kullanılarak Kesme (Boşluk Oluşturma) işlemi
yapılmıştır.

swApp.SetUserPreferenceToggle swSketchInference, False

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "VERTEX", 0, as_2, mesafe, True, 0, Nothing, 0)
part.InsertAxis2 True

swApp.SetUserPreferenceToggle swSketchInference, True

part.SaveAs2 fi0 + "Ayak Sacı.SLDPRT", 0, False, False

End Sub

Private Sub Command2_Click()
Ayak_Sacı.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9

```



```

    parca_listesi.AddItem "Ön Yatak"
Case Is = 10
    parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
    parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
    parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
    parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
    parca_listesi.AddItem "Yan Kapak"
Case Is = 15
    parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
    parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
    parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
    parca_listesi.AddItem "Borc Federi"
Case Is = 21
    parca_listesi.AddItem "Borc Takviyesi"
'Case Else
    'parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()
    Combo1.AddItem "as_parca_kalınlığı"
    Combo1.AddItem "as_1"
    Combo1.AddItem "as_2"
    Combo1.AddItem "as_3"
    Combo1.AddItem "as_4"
    Combo1.AddItem "as_5"
    Combo1.AddItem "as_parca_genisligi"
    Combo1.AddItem "as_r"
    'Combo1.AddItem "gys_6"
    'Combo1.AddItem "gys_5"
    'Combo1.AddItem "gys_1"
    'Combo1.AddItem "gys_3"
    'Combo1.AddItem "r1"
    'Combo1.AddItem "gys_4"
    'Combo1.AddItem "r2"
    'Combo1.AddItem "cs_8"
    'Combo1.AddItem "gys_2"
    'Combo1.AddItem "cs_10"
    'Combo1.AddItem "cs_3"
    'Combo1.AddItem "cs_2"
    'Combo1.AddItem "cs_1"
    'Combo1.AddItem "cs_r1"
    'Combo1.AddItem "cygs_KM"
    'Combo1.AddItem "cygs_parca_kalınlığı"
    'Combo1.AddItem "mesafe"
    'Combo1.AddItem "ks_parca_kalınlığı"
    'Combo1.AddItem "ct_parca_boyu"
    'Combo1.AddItem "ms_parca_kalınlığı"
    'Combo1.AddItem "C_Muhafaza_Sacı_KM"

parca_listesi.Visible = False
End Sub

Private Sub mnu_dosya_ayaksacı_Click()
Dim bb
    bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "ayak_sacı_parametrik_degerler.txt", vbNormalFocus)
End Sub

```

Option Explicit

```
Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub
```

```
Private Sub Command1_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "bt_parametrik_degerler.txt", vbNormalFocus)
End Sub
```

```
Private Sub Command2_Click()
Call VeriOkuma
```

```
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc
```

Call snap

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.SketchRectangle 0, 0, 0, bt_2, bt_1, 0, 0
part.ViewZoomtofit2
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, bt_parca_kalınligı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
```

```
boolstatus = part.Extension.SelectByID2("", "EDGE", 0, bt_1, bt_parca_kalınligı / 2, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "EDGE", bt_2, bt_1, bt_parca_kalınligı / 2, True, 0, Nothing, 0)
' "4" Chamfer ile ilgili bir secenek, bt_3 "0.015" chamfer da kullanılan kesilme miktarı yani pah mesafesi,"0.785398..." de 45
derecelik acının radyan karşılığı
part.FeatureManager.InsertFeatureChamfer 4, 1, bt_3, 0.7853981633975, 0, 0, 0, 0
```

```
boolstatus = part.Extension.SelectByID2("", "VERTEX", bt_3, bt_1, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "VERTEX", bt_2 - bt_3, bt_1, 0, True, 0, Nothing, 0)
part.InsertAxis2 True
```

```
boolstatus = part.Extension.SelectByID2("", "VERTEX", bt_3, bt_1, bt_parca_kalınligı, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "VERTEX", bt_2 - bt_3, bt_1, bt_parca_kalınligı, True, 0, Nothing, 0)
part.InsertAxis2 True
```

```
.....
....."KAYNAKLI BÖLÜM".....
.....
```

' extrusion

```
.....
boolstatus = part.Extension.SelectByID2("", "FACE", bt_2 / 2, bt_1 / 2, bt_parca_kalınligı, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
boolstatus = part.SketchUseEdge2(False)
part.ClearSelection2 True
```

```
part.FeatureManager.FeatureExtrusion True, False, False, 0, 0, parcalar_arasi_bosluk * 2, 0, False, False, False, False, 0, 0,
False, False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", bt_2 / 2, bt_1 / 2, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
boolstatus = part.SketchUseEdge2(False)
part.ClearSelection2 True
```

```
part.FeatureManager.FeatureExtrusion True, False, False, 0, 0, parcalar_arasi_bosluk * 2, 0, False, False, False, False, 0, 0,
False, False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

' cut out

```

*****
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, parcalar_arasi_bosluk, bt_1, 0, 0
part.SketchRectangle bt_2, 0, 0, bt_2 - parcalar_arasi_bosluk, bt_1, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, False, 6, 0, bt_parca_kalinligi * 4, 0, False, False, False, False,
0.01745329251994, 0.01745329251994, False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.SaveAs2 fl0 + "Burc Takviyesi.SLDPRPT", 0, False, False
End Sub

Private Sub Command3_Click()
Burc_Federi_ve_Takviyesi.Hide

End Sub

Private Sub Command4_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazilacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16

```

```

    parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
    parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
    parca_listesi.AddItem "Borc Federi"
Case Is = 21
    parca_listesi.AddItem "Borc Takviyesi"
'Case Else
    ' parca_listesi.AddItem "Aradığınız Degisken Baska Bir Parçayı Etkilememktedir."

    End Select
End If
Next j
Next i

End Sub

Private Sub Command5_Click()
Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + ".part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, 0, 0, bf_2, 0, 0
part.CreateLine2 bf_2, 0, 0, bf_2, bf_1, 0
part.CreateLine2 bf_2, bf_1, 0, bf_2 - bf_3, bf_1, 0
part.CreateLine2 bf_2 - bf_3, bf_1, 0, 0, bf_3, 0
part.CreateLine2 0, bf_3, 0, 0, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, bf_parca_kalınligı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "EDGE", bf_2, 0, bf_parca_kalınligı / 2, True, 0, Nothing, 0)
' "4" Chamfer ile ilgili bir secenek, "bf_4" 10 mm chamfer da kullanılan kesilme miktarı yani pah mesafesi,"0.785398..." de 45
derecelik acının radyan karşılığı
part.FeatureManager.InsertFeatureChamfer 4, 1, bf_4, 0.7853981633975, 0, 0, 0, 0

boolstatus = part.Extension.SelectByID2("", "VERTEX", 0, bf_3, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.InsertAxis2 True

boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 bf_parca_kalınligı, False, True

boolstatus = part.Extension.SelectByID2("right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 bf_2, False, True

boolstatus = part.Extension.SelectByID2("", "VERTEX", 0, 0, bf_parca_kalınligı, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "VERTEX", 0, bf_3, bf_parca_kalınligı, True, 0, Nothing, 0)
part.InsertAxis2 True

' Bundan sonra ki kısım federin tabanına küresel bir yüzey kazandırmak ve burc la olan temasının sağlanması için
yapılmaktadır.
'

Dim kesme_derinligi As Double
kesme_derinligi = 0.001

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True

```

```

part.CreateCircleByRadius2 -bf_parca_kalınlgı / 2, -(oy_r2 - kesme_derinligi), 0, oy_r2

part.ClearSelection2 True
part.FeatureManager.FeatureCut True, False, True, 0, 0, bf_2 * 2, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0
boolstatus = part.Extension.SelectByID2("", "FACE", bf_2 / 2, kesme_derinligi, bf_parca_kalınlgı / 2, True, 0, Nothing, 0)
part.InsertAxis2 True
part.SaveAs2 fl0 + "Burc Federi.SLDPR1", 0, False, False

End Sub
Private Sub Command6_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "bf_parametrik_degerler.txt", vbNormalFocus)
End Sub

Private Sub Form_Load()
Combo1.AddItem "bf_1"
Combo1.AddItem "bf_2"
Combo1.AddItem "bf_3"
Combo1.AddItem "bf_4"
Combo1.AddItem "bf_parca_kalınlgı"
Combo1.AddItem "bt_1"
Combo1.AddItem "bt_2"
Combo1.AddItem "bt_3"
Combo1.AddItem "bt_parca_kalınlgı"
Combo1.AddItem "bt_uzaklık"

parca_listesi.Visible = False
End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma
"*****GYS Olusumundan Buraya Yansıyacak Degerler İçin*****"

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
*****
*****

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

Öteleme_x = ((gys_1 - gys_3 - r1) - cs_8)
Öteleme_y = (gys_2 - cs_10)

xp1 = cs_2
yp1 = cs_1
xp2 = 0
yp2 = 0
R = cs_r1

```

Call newtonR

r1_csac degismiyor,yukarıdaki xp1 ve yp1,R degerleri degisiyor

```
cs_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
cs_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
```

```
part.CreateArc2 cs_merk_x + Öteleme_x, cs_merk_y + Öteleme_y, 0, xp2 + Öteleme_x, yp2 + Öteleme_y, 0, xp1 +
Öteleme_x, yp1 + Öteleme_y, 0, -1
part.ViewZoomtofit2
part.CreateLine2 cs_2 + Öteleme_x, cs_1 + Öteleme_y, 0, cs_2 + cs_3 + Öteleme_x, cs_1 + Öteleme_y, 0
part.CreateLine2 cs_2 + cs_3 + Öteleme_x, cs_1 + Öteleme_y, 0, cs_2 + cs_3 + Öteleme_x, gys_5, 0
part.CreateLine2 cs_2 + cs_3 + Öteleme_x, gys_5, 0, gys_6, gys_5, 0
```

""ASAGIDAKİLERE GEREK KALMAMISTIR, CUNKU c_SACI NIN İC RADIUSU GYS_YE GÖRE CIZILECEKTİR.""

```
'xp1 = cs_7
'yp1 = cs_6
'xp2 = cs_8
'yp2 = cs_9
'R = r2
'Call newtonR
'cs_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
'cs_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
```

```
part.CreateArc2 gys_merk_x, gys_merk_y, 0, gys_6, gys_5, 0, gys_1 - gys_3 - r1, gys_4, 0, 1
part.CreateLine2 0 + Öteleme_x, 0 + Öteleme_y, 0, cs_11 + Öteleme_x, 0 + Öteleme_y, 0
part.CreateLine2 cs_11 + Öteleme_x, 0 + Öteleme_y, 0, cs_11 + Öteleme_x, gys_2, 0
```

```
part.CreateLine2 cs_11 + Öteleme_x, gys_2, 0, (gys_1 - gys_3 - r1), gys_2, 0
part.CreateLine2 (gys_1 - gys_3 - r1), gys_2, 0, (gys_1 - gys_3 - r1), gys_4, 0
```

```
boolstatus = part.Extension.SelectByID2("Point9", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
' 1. değişken acıdır ve radyan cinsinden girilmektedir, 45 derece acı olarak, 2. deg. mesafedir.
part.SketchChamfer 0.7853981633975, cs_12, 0
```

```
boolstatus = part.Extension.SelectByID2("Point1", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 cs_r6, 1
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Point2", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 cs_r2, 1
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Point6", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 cs_r3, 1
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Point11", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 cs_r5, 1
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.SketchAddConstraints "sgVERTICAL2D"
boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.SketchAddConstraints "sgHORIZONTAL2D"
```

part.ViewZoomtofit2

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, cs_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", (cs_2 + cs_3) + Öteleme_x, gys_2, cs_parca_kalnlığı / 2, True, 0,
Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
```

```

boolstatus = part.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1, (gys_2 + ((gys_4 - gys_2) / 2)), cs_parca_kalnlığı / 2,
False, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1, cs_12 + Öteleme_y, cs_parca_kalnlığı, False, 0,
Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + Öteleme_x, ((cs_1 + Öteleme_y - gys_5) / 2) + gys_5,
cs_parca_kalnlığı / 2, False, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
part.SaveAs2 fl0 + "C_Saci.SLDPRT", 0, False, False

```

```

*****C TAKVİYESİ *****

```

```

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

```

Call snap
Call c_takviyesi_parametre_okutma

```

```

Dim ct_parca_genisligi As Double
ct_parca_genisligi = mesafe - 2 * cs_parca_kalnlığı

```

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

```

```

part.SketchRectangle 0, 0, 0, ct_parca_boyu, ct_parca_kalnlığı, 0, 0
part.ClearSelection2 True

```

```

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, ct_parca_genisligi, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

```

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

```

```

part.CreateLineVB ct_parca_boyu / 2, 0, 0, ct_parca_boyu / 2, -ct_parca_genisligi, 0
part.SetPickMode
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("", "FACE", ct_parca_genisligi / 3, 0, ct_parca_boyu / 2, False, 1, Nothing, 0)
part.InsertSplitLineProject False, False

```

```

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "VERTEX", ct_parca_boyu / 2, 0, ct_parca_genisligi, True, 0, Nothing, 0)
part.InsertAxis2 True

```

```

*****KAYNAKLI BÖLÜM*****

```

```

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, -parcalar_arasi_bosluk, ct_parca_kalnlığı, 0, 0
part.SketchRectangle -(mesafe - 2 * cs_parca_kalnlığı), 0, 0, -(mesafe - 2 * cs_parca_kalnlığı - parcalar_arasi_bosluk),
ct_parca_kalnlığı, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

```

```

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

```

```

boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, ct_parca_boyu, 0, False, False, False, False, 0, 0, False, False, False,
False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
part.SketchRectangle 0, ks_parca_kalınligı / 2, 0, -(mesafe - 2 * cs_parca_kalınligı), ct_parca_kalınligı, 0, 0

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.ClearSelection2 True

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, ct_parca_boyu, parcalar_arasi_bosluk, 0, 0

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, (mesafe - 2 * cs_parca_kalınligı), 0, False, False, False, False, 0, 0,
False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.SaveAs2 fl0 + "C_Takviyesi.SLDPRT", 0, False, False

End Sub

Private Sub Command2_Click()
C_saci.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5

```



```

        parca_listesi.AddItem "Ayak Sacı"
    Case Is = 6
        parca_listesi.AddItem "Ön Pano"
    Case Is = 7
        parca_listesi.AddItem "P ye Gelen Sac"
    Case Is = 8
        parca_listesi.AddItem "Kızak Sacı"
    Case Is = 9
        parca_listesi.AddItem "Ön Yatak"
    Case Is = 10
        parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
    Case Is = 11
        parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
    Case Is = 12
        parca_listesi.AddItem "C ye Gelen Sac"
    Case Is = 13
        parca_listesi.AddItem "No 5 Sacı"
    Case Is = 14
        parca_listesi.AddItem "Yan Kapak"
    Case Is = 15
        parca_listesi.AddItem "Krank Orta Yatak"
    Case Is = 16
        parca_listesi.AddItem "Krank Arka Yatak"
    Case Is = 17
        parca_listesi.AddItem "Arka Yatak Destek Sacı"
    Case Is = 18
        parca_listesi.AddItem "Ön Yatak Destek Sacı"
    Case Is = 19
        parca_listesi.AddItem "No 4 Sacı"
    Case Is = 20
        parca_listesi.AddItem "Borc Federi"
    Case Is = 21
        parca_listesi.AddItem "Borc Takviyesi"
    'Case Else
        ' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

    End Select
End If
Next j
Next i
End Sub

Private Sub Form_Load()
    'Combo1.AddItem "gys_6"
    'Combo1.AddItem "gys_5"
    'Combo1.AddItem "gys_1"
    'Combo1.AddItem "gys_3"
    'Combo1.AddItem "r1"
    'Combo1.AddItem "gys_4"
    'Combo1.AddItem "r2"
    'Combo1.AddItem "cs_1"
    'Combo1.AddItem "gys_2"
    'Combo1.AddItem "cs_2"
    'Combo1.AddItem "cs_3"
    'Combo1.AddItem "cs_8"
    'Combo1.AddItem "cs_10"
    'Combo1.AddItem "cs_11"
    'Combo1.AddItem "cs_12"
    'Combo1.AddItem "cs_parca_kalınlığı"
    'Combo1.AddItem "cs_r1"
    'Combo1.AddItem "cs_r2"
    'Combo1.AddItem "cs_r3"
    'Combo1.AddItem "cs_r5"
    'Combo1.AddItem "cs_r6"
    'Combo1.AddItem "mesafe"
    'Combo1.AddItem "ct_parca_boyu"
    'Combo1.AddItem "ct_parca_kalınlığı"

    parca_listesi.Visible = False
End Sub

Private Sub mnu_dosya_c_sac_Click()
    Dim bb

```

```

bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "c_sacı_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Command1_Click()

c_kaynak_dikis_boyutu = Val(Text1.Text) / 1000
c_kaynak_dikis_boyu = c_kaynak_dikis_boyutu * Sqr(2)

Call VeriOkuma
"*****GYS Olusumundan Buraya Yansiyacak Degerler Icin*****"

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)

*****
*****

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

Öteleme_x = ((gys_1 - gys_3 - r1) - cs_8)
Öteleme_y = (gys_2 - cs_10)

xp1 = cs_2
yp1 = cs_1
xp2 = 0
yp2 = 0
R = cs_r1

Call newtonR

r1_csac degismiyor,yukarıdaki xp1 ve yp1,R degerleri degisiyor

cs_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
cs_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)

part.CreateArc2 cs_merk_x + Öteleme_x, cs_merk_y + Öteleme_y, 0, xp2 + Öteleme_x, yp2 + Öteleme_y, 0, xp1 +
Öteleme_x, yp1 + Öteleme_y, 0, -1
part.ViewZoomtofit2
part.CreateLine2 cs_2 + Öteleme_x, cs_1 + Öteleme_y, 0, cs_2 + cs_3 + Öteleme_x - ks_parca_kalınligı, cs_1 + Öteleme_y, 0
part.CreateLine2 cs_2 + cs_3 + Öteleme_x, cs_1 + Öteleme_y, 0, cs_2 + cs_3 + Öteleme_x, gys_5, 0
part.CreateLine2 cs_2 + cs_3 + Öteleme_x, gys_5, 0, gys_6, gys_5, 0

part.CreateArc2 gys_merk_x, gys_merk_y, 0, gys_6, gys_5, 0, gys_1 - gys_3 - r1, gys_4, 0, 1
part.CreateLine2 0 + Öteleme_x, 0 + Öteleme_y, 0, cs_11 + Öteleme_x, 0 + Öteleme_y, 0
part.CreateLine2 cs_11 + Öteleme_x, 0 + Öteleme_y, 0, cs_11 + Öteleme_x, gys_2, 0

part.CreateLine2 cs_11 + Öteleme_x, gys_2, 0, (gys_1 - gys_3 - r1), gys_2, 0
part.CreateLine2 (gys_1 - gys_3 - r1), gys_2, 0, (gys_1 - gys_3 - r1), gys_4, 0

boolstatus = part.Extension.SelectByID2("Point5", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
' 1. deęişken açıdır ve radyan cinsinden girilmektedir, 45 derece açı olarak, 2. deg. mesafedir.

```



```

part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r,
0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalınligı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r - cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r - cygs_m, 0, 0, as_2, 0
Else
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınligı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_m, 0,
0, as_2, 0

```

End If

' p ye gelen sac kesimi

Call VeriOkuma

```

pygs_y = gys_4 - 0.02
pygs_parca_kalınligı = 0.016 ' 16 mm
'0.02 = 20 mm sabit olacak olan gys_4 mesafesinden saca uzaklık
alfa1 = Atn((pygs_y - as_3) / (gys_1 - gys_3 - r1 - as_parca_kalınligı))
'P_ye_gelen_sac.Print "r1=", r1, (gys_1 - gys_3 - r1 - as_parca_kalınligı), (pygs_y)
'alfa1 = Atn(217.2 / 802)
alfa = alfa1 * 180 / 3.141592654
beta = 90 - alfa
' alfa = 15.15, beta = 74.85 derece
'P_ye_gelen_sac.Print alfa, beta

```

' küçük üçgen için

```

beta1 = Tan(3.141592654 / 2 - alfa1)
'beta1=(16/k)

```

```

pygs_k = (pygs_parca_kalınligı / beta1)
'P_ye_gelen_sac.Print beta1, pygs_k
' ayak sacına yapışacak olan yüzeyin yandan görünen hattının uzunluğu m olsun
m = Sqr(pygs_k ^ 2 + pygs_parca_kalınligı ^ 2)
Kaynak.Print m

```

```

"""" Pye gelen sacı yeni değişkenler kullanmaya gerek kalmadan çizdirelim, text. file kullanmadan """"

```

' Y ekseninde (as_3 - m) kadar yukarı ötelimiz.

' X ekseninde de as_parca_kalınligı kadar ötelimiz.

```

part.CreateLine2 as_parca_kalınligı, (as_3 - m), 0, as_parca_kalınligı, as_3, 0
part.CreateLine2 as_parca_kalınligı, as_3, 0, (gys_1 - gys_3 - r1), pygs_y, 0
part.CreateLine2 as_parca_kalınligı, (as_3 - m), 0, ((gys_1 - gys_3 - r1)), (pygs_y - m), 0
part.CreateLine2 ((gys_1 - gys_3 - r1)), (pygs_y - m), 0, ((gys_1 - gys_3 - r1)), pygs_y, 0
part.ViewZoomtofit2

```

part.ClearSelection2 True

Call part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)

' aşağıdaki 0.05 ve 0.045 değeri olesine bir degerdir, burada mesafe degeri okunmadığı için onun yerine yazılmış bir degerdir.

```

part.FeatureManager.FeatureCut False, False, False, 0, 0, 0.05, 0.045, False, False, False, False, 0, 0, False, False, False, False,
0, 1, 1

```

part.SelectionManager.EnableContourSelection = 0

45

```

"""" KAYNAKLI BÖLÜM """"

```

boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchManager.InsertSketch True

```

part.CreateArc2 gys_merk_x, gys_merk_y, 0, gys_6, gys_5, 0, gys_1 - gys_3 - r1, gys_4, 0, 1
part.CreateLine2 cs_11 + Öteleme_x, gys_2, 0, (gys_1 - gys_3 - r1), gys_2, 0
part.CreateLine2 (gys_1 - gys_3 - r1), gys_2, 0, (gys_1 - gys_3 - r1), gys_4, 0

```

```

part.CreateLine2 gys_6, gys_5, 0, cs_2 + cs_3 + Öteleme_x, gys_5, 0
part.ClearSelection2 True

part.ViewZoomtofit2

part.SketchManager.InsertSketch True

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 (Öteleme_x + cs_11), False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Plane2", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True

swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle 0, gys_2, 0, parcalar_arasi_bosluk, gys_2 - c_kaynak_dikis_boyutu, 0, 0

swApp.SetUserPreferenceToggle swSketchInference, True

part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True

boolstatus = part.Extension.SelectByID2("Sketch4", "SKETCH", 0, 0, 0, False, 1, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, True, 4, Nothing, 0)
'Dim SweepFeature As Object -----ikinci kez yazmaya gerek yok
Set SweepFeature = part.FeatureManager.InsertProtrusionSwept3(False, False, 0, False, False, 0, 0, False, 0, 0, 0, 1, 1, 1, 0, 1)

part.SaveAs2 fl0 + "C Kaynak Dikisi_sag.SLDPRT", 0, False, False

*****
*****2 . kAYNAK *****
Call VeriOkuma
*****"GYS Olusumundan Buraya Yansiyacak Degerler Icin"*****

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
*****
*****

Set swApp = GetObject("sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

Öteleme_x = ((gys_1 - gys_3 - r1) - cs_8)
Öteleme_y = (gys_2 - cs_10)

xp1 = cs_2
yp1 = cs_1
xp2 = 0
yp2 = 0
R = cs_r1

Call newtonR

```

r1_csac degismiyor,yukarıdaki xp1 ve yp1,R degerleri degisiyor

```
cs_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
cs_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
```

```
part.CreateArc2 cs_merk_x + Öteleme_x, cs_merk_y + Öteleme_y, 0, xp2 + Öteleme_x, yp2 + Öteleme_y, 0, xp1 +
Öteleme_x, yp1 + Öteleme_y, 0, -1
part.ViewZoomtofit2
part.CreateLine2 cs_2 + Öteleme_x, cs_1 + Öteleme_y, 0, cs_2 + cs_3 + Öteleme_x - ks_parca_kalınligı, cs_1 + Öteleme_y, 0
'part.CreateLine2 cs_2 + cs_3 + Öteleme_x, cs_1 + Öteleme_y, 0, cs_2 + cs_3 + Öteleme_x, gys_5, 0
'part.CreateLine2 cs_2 + cs_3 + Öteleme_x, gys_5, 0, gys_6, gys_5, 0
```

```
'part.CreateArc2 gys_merk_x, gys_merk_y, 0, gys_6, gys_5, 0, gys_1 - gys_3 - r1, gys_4, 0, 1
part.CreateLine2 0 + Öteleme_x, 0 + Öteleme_y, 0, cs_11 + Öteleme_x, 0 + Öteleme_y, 0
part.CreateLine2 cs_11 + Öteleme_x, 0 + Öteleme_y, 0, cs_11 + Öteleme_x, gys_2, 0
```

```
'part.CreateLine2 cs_11 + Öteleme_x, gys_2, 0, (gys_1 - gys_3 - r1), gys_2, 0
'part.CreateLine2 (gys_1 - gys_3 - r1), gys_2, 0, (gys_1 - gys_3 - r1), gys_4, 0
```

```
boolstatus = part.Extension.SelectByID2("Point5", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
' 1. deęişken acıdır ve radyan cinsinden girilmektedir, 45 derece acı olarak, 2. deg. mesafedir.
part.SketchChamfer 0.7853981633975, cs_12, 0
```

```
boolstatus = part.Extension.SelectByID2("Point1", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 cs_r6, 1
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Point2", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 cs_r2, 1
part.ClearSelection2 True
```

part.ViewZoomtofit2

part.SketchManager.InsertSketch True

```
boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 gys_2, False, True
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True
```

```
swApp.SetUserPreferenceToggle swSketchInference, False
part.CreateLine2 -(cs_11 + Öteleme_x), 0 + parcalar_arasi_bosluk, 0, -(cs_11 + Öteleme_x), -c_kaynak_dikis_boyu +
parcalar_arasi_bosluk, 0
part.CreateLine2 -(cs_11 + Öteleme_x), -c_kaynak_dikis_boyu + parcalar_arasi_bosluk, 0, -(cs_11 + Öteleme_x +
c_kaynak_dikis_boyu), 0 + parcalar_arasi_bosluk, 0
part.CreateLine2 -(cs_11 + Öteleme_x + c_kaynak_dikis_boyu), 0 + parcalar_arasi_bosluk, 0, -(cs_11 + Öteleme_x), 0 +
parcalar_arasi_bosluk, 0
swApp.SetUserPreferenceToggle swSketchInference, True
```

```
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
```

```
boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 1)
part.SketchAddConstraints "sgPERPENDICULAR"
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 1, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, True, 4, Nothing, 0)
```

```
Set SweepFeature = part.FeatureManager.InsertProtrusionSwept3(False, False, 0, False, False, 0, 0, False, 0, 0, 0, 1, 1, 1, 0,
1)
```

GoTo 90 ' kesimi iptal ediyoruz .. icice gecmesi problem degil
Call C_yegelen_Rutin

```
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
```

```
part.CreateLine2 0, as_2, 0, 0, (as_2 + cygs_m), 0
part.ViewZoomtofit2
If ct_x < (pi / 2) Then
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r,
0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r - cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r - cygs_m, 0, 0, as_2, 0
Else
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_m, 0,
0, as_2, 0
```

```
End If
```

```
' p ye gelen sac kesimi
```

```
Call VeriOkuma
```

```
pygs_y = gys_4 - 0.02
pygs_parca_kalnlığı = 0.016 ' 16 mm
'0.02 = 20 mm sabit olacak olan gys_4 mesafesinden saca uzaklık
alfa1 = Atn((pygs_y - as_3) / (gys_1 - gys_3 - r1 - as_parca_kalnlığı))
'P_ye_gelen_sac.Print "r1=", r1, (gys_1 - gys_3 - r1 - as_parca_kalnlığı), (pygs_y)
'alfa1 = Atn(217.2 / 802)
alfa = alfa1 * 180 / 3.141592654
beta = 90 - alfa
' alfa = 15.15, beta = 74.85 derece
'P_ye_gelen_sac.Print alfa, beta
```

```
' küçük üçgen için
```

```
beta1 = Tan(3.141592654 / 2 - alfa1)
'beta1=(16/k)
```

```
pygs_k = (pygs_parca_kalnlığı / beta1)
'P_ye_gelen_sac.Print beta1, pygs_k
'ayak sacına yapışacak olan yüzeyin yandan görünen hattının uzunluğu m olsun
m = Sqr(pygs_k ^ 2 + pygs_parca_kalnlığı ^ 2)
'P_ye_gelen_sac.Print m
```

```
"" Pye gelen sacı yeni değişkenler kullanmaya gerek kalmadan çizdirelim, text. file kullanmadan ""
```

```
' Y ekseninde (as_3 - m) kadar yukarı ötelerez.
```

```
'X ekseninde de as_parca_kalnlığı kadar ötelendir.
```

```
part.CreateLine2 as_parca_kalnlığı, (as_3 - m), 0, as_parca_kalnlığı, as_3, 0
part.CreateLine2 as_parca_kalnlığı, as_3, 0, (gys_1 - gys_3 - r1), pygs_y, 0
part.CreateLine2 as_parca_kalnlığı, (as_3 - m), 0, ((gys_1 - gys_3 - r1)), (pygs_y - m), 0
part.CreateLine2 ((gys_1 - gys_3 - r1)), (pygs_y - m), 0, ((gys_1 - gys_3 - r1)), pygs_y, 0
part.ViewZoomtofit2
```

```
part.ClearSelection2 True
```

```
Call part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
```

```
' aşağıdaki 0.05 degeri olesine bir degerdir, burada mesafe degeri okunamadığı için onun yerine yazılmış bir degerdir.
```

```
part.FeatureManager.FeatureCut False, False, False, 0, 0, 0.05, 0.045, False, False, False, False, 0, 0, False, False, False, False,
0, 1, 1
```

```
part.SelectionManager.EnableContourSelection = 0
```

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```

.....
....."KAYNAKLI BÖLÜM".....
.....

```

```

boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

```

```

part.CreateArc2 gys_merk_x, gys_merk_y, 0, gys_6, gys_5, 0, gys_1 - gys_3 - r1, gys_4, 0, 1
part.CreateLine2 cs_11 + Öteleme_x, gys_2, 0, (gys_1 - gys_3 - r1), gys_2, 0
part.CreateLine2 (gys_1 - gys_3 - r1), gys_2, 0, (gys_1 - gys_3 - r1), gys_4, 0
part.CreateLine2 gys_6, gys_5, 0, cs_2 + cs_3 + Öteleme_x, gys_5, 0
part.ClearSelection2 True

```

```

part.ViewZoomtofit2

```

```

part.SketchManager.InsertSketch True

```

```

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 (Öteleme_x + cs_11), False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("Plane2", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True

```

```

swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle 0, gys_2, 0, -parcalar_arasi_bosluk, gys_2 - c_kaynak_dikis_boyutu, 0, 0

```

```

swApp.SetUserPreferenceToggle swSketchInference, True

```

```

part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True

```

```

boolstatus = part.Extension.SelectByID2("Sketch4", "SKETCH", 0, 0, 0, False, 1, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, True, 4, Nothing, 0)
Set SweepFeature = part.FeatureManager.InsertProtrusionSwept3(False, False, 0, False, False, 0, 0, False, 0, 0, 0, 1, 1, 1, 0, 1)

```

```

part.SaveAs2 fl0 + "C Kaynak Dikisi_sol.SLDPRT", 0, False, False

```

```

End Sub

```

```

Private Sub Command2_Click()
C_Saci_Kaynak_Dikisi.Hide

```

```

End Sub

```

```

Private Sub Form_Load()

```

```

End Sub

```

```

Option Explicit

```

```

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

```

```

Private Sub Command1_Click()

```

```

Call C_yegelen_Rutin

```

```

Set swApp = GetObject("sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

```

Call snap

```

```

part.SketchManager.InsertSketch True

```



```

boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

Rem part.CreateLine2 0, as_2, 0, 0, (as_2 + cygs_m), 0
Rem part.ViewZoomtofit2
Rem If ct_x < (pi / 2) Then
Rem part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
Rem part.ViewZoomtofit2
Rem part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx -
Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_e), gys_5 - cygs_f, 0
Rem part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_e), gys_5 - cygs_f, 0, 0, as_2, 0
Rem Else
Rem part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
Rem part.ViewZoomtofit2
Rem part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx -
Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_e), gys_5 - cygs_f, 0
Rem part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_e), gys_5 - cygs_f, 0, 0, as_2, 0
Rem End If
Rem part.ViewZoomtofit2

Rem C_ye_gelen_sac.Print cXx, Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2), cygs_e, cygs_f

*****
part.CreateLine2 0, as_2, 0, 0, (as_2 + cygs_m), 0
part.ViewZoomtofit2
C_ye_gelen_sac.Print cXx, ms_x, ms_parca_kalnlığı, cygs_r, ct_alfa, gys_5

If ct_x < (pi / 2) Then
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r,
0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0,
0, as_2, 0
Else
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, gys_5 - cygs_r - cygs_y, 0,
0, as_2, 0
End If

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.SketchAddConstraints "sgVERTICAL2D"
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 1)
part.SketchAddConstraints "sgPARALLEL"
part.ClearSelection2 True

*****

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 mesafe, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True

' Asagıda, ct_x acısına 0.017 ekledim yani 1 derece cunku aci tam olusmadığı için kesince yuzeyler birbirine tam degmezi
mesh de prob. olur , 1 derece fazla olursa parca icine girer ama mesh de problem olmaz!!!
If ct_x < (pi / 2) Then

```

```

part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (((cXx - Sqr(ms_x
^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x + 0.017)), gys_5 - cygs_r - cygs_ms_boyu *
Sin(ct_x + 0.017), 0
part.CreateLine2 (((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x +
0.017)), ((gys_5 - cygs_r) - cygs_ms_boyu * Sin(ct_x + 0.017)), 0, (((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r *
Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x + 0.017)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2), ((gys_5 - cygs_r) -
cygs_ms_boyu * Sin(ct_x + 0.017)), 0
part.CreateLine2 (((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x +
0.017)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2), gys_5 - cygs_r - cygs_ms_boyu * Sin(ct_x + 0.017), 0, (cXx - Sqr(ms_x ^ 2
+ ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2), gys_5 - cygs_r, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^
2), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0
Else
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (((cXx - Sqr(ms_x
^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x + 0.017)), gys_5 - cygs_r - cygs_ms_boyu
* Sin(ct_x + 0.017), 0
part.CreateLine2 (((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x +
0.017)), ((gys_5 - cygs_r) - cygs_ms_boyu * Sin(ct_x + 0.017)), 0, (((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r *
Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x + 0.017)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2), ((gys_5 - cygs_r) -
cygs_ms_boyu * Sin(ct_x + 0.017)), 0
part.CreateLine2 (((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa))) - cygs_ms_boyu * Cos(ct_x +
0.017)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2), gys_5 - cygs_r - cygs_ms_boyu * Sin(ct_x + 0.017), 0, (cXx - Sqr(ms_x ^ 2
+ ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2), gys_5 - cygs_r, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı
^ 2), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0
End If

boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.SketchAddConstraints "sgHORIZONTAL2D"
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

' Asagıda kesimi iptal ettik icine girmesi icin " meshde problem oluyor"
part.FeatureManager.FeatureCut True, False, False, 0, 0, mesafe, 0.01, False, False, False, False, 0, 0, False, False, False, False,
0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("", "VERTEX", 0, as_2, mesafe, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.InsertAxis2 True

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateCircleByRadius2 cs_merk_x + Öteleme_x, cs_merk_y + Öteleme_y, 0, cs_r1

part.FeatureManager.FeatureCut True, False, True, 0, 0, cs_parca_kalnlığı + parcalar_arasi_bosluk, 0, False, False, False,
False, 0, 0, False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateCircleByRadius2 cs_merk_x + Öteleme_x, cs_merk_y + Öteleme_y, 0, cs_r1

part.FeatureManager.FeatureCut True, False, False, 0, 0, cs_parca_kalnlığı + parcalar_arasi_bosluk, 0, False, False, False,
False, 0, 0, False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

.....
....."KABLO KANALI".....
.....

GoTo 55
'ana_form.Print(((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) + cygs_x) / 2, ((gys_5 - cygs_r -
cygs_y) + as_2) / 2, mesafe / 2
If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa))
+ cygs_x) / 2, ((gys_5 - cygs_r - cygs_y) + as_2) / 2, mesafe / 2, False, 0, Nothing, 0)
Else

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa))
+ cygs_x) / 2, ((gys_5 - cygs_r - cygs_y) + as_2) / 2, mesafe / 2, False, 0, Nothing, 0)
End If
part.SketchManager.InsertSketch True
part.ClearSelection2 True " Asagıdaki 0.375274157, 0, 0, 0.440274157.... degerleri Orjinal montaj uzerinden olculerek
alınmıştır.Bu nokta dikdortgenin baslangıç noktası alt kenara olan uzaklık 0.059 alınarak 65 x 43 lük bir dikdörtgen çizilmiştir.
'0.3162663509097=Cızım yapılan yuzey sol alt kose X koordinatı
part.SketchRectangle 0.375274157, 0, 0, 0.440274157, 0.043, 0, 1

```

```

boolstatus = part.Extension.SelectByID2("Sketch5", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, False, 0, 0, cygs_parca_kalnlığı * 2, 0, False, False, False, False, 0, 0, False,
False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0
55

```

```

.....
....."KAYNAKLI BÖLÜM".....
.....

```

```

Kullanilan_Parca_Kalnlığı = cygs_parca_kalnlığı
'x_factor = 0.5
x_factor = CGSD
Call Kaynak

```

```

If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
Else
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
End If
part.SketchManager.InsertSketch True
part.ClearSelection2 True
If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
Else
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
End If
boolstatus = part.SketchUseEdge2(False)
part.ClearSelection2 True
part.ClearSelection2 True
part.FeatureManager.FeatureCut True, False, False, 0, 0, parcalar_arasi_bosluk, 0, True, False, True, False, 1.047197551197, 0,
False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

```

```

'eeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeeee
" Kaynak dikislerinin olusturulması

```

```

If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
Else
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
End If
part.SketchManager.InsertSketch True
part.ClearSelection2 True
If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
Else
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
End If
boolstatus = part.SketchUseEdge2(False)
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("Line27", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line28", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line29", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line30", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line31", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line32", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.EditDelete
part.SketchManager.InsertSketch True

boolstatus = part.Extension.SelectByID2("Line25@Sketch6", "EXTSKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlanePerCurveAndPassPoint3 False, True

boolstatus = part.Extension.SelectByID2("Plane2", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True

' """"""ÖNEMLİ... Asagıda 10 mm eklememizin amacı kaynak dikisini büyük yapıp parca icine girmesini saglamak..
' ve böylece c sacının sırt ı ile cygs nin birlestigi yerdeki kaynak dikişlerinin bosluklarını ortadan kaldırmak
swApp.SetUserPreferenceToggle swSketchInference, False
part.CreateLine2 -0.01, 0, 0, -0.01, (Kaynak_Dikis_Boyu + 0.01), 0
part.CreateLine2 -0.01, (Kaynak_Dikis_Boyu + 0.01), 0, Kaynak_Dikis_Boyu, 0, 0
part.CreateLine2 Kaynak_Dikis_Boyu, 0, 0, -0.01, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.SketchAddConstraints "sgPERPENDICULAR"

'C_ye_gelen_sac.Print Sqr(cygs_m ^ 0.5 - cygs_k ^ 0.5), cygs_parca_kalınlığı

part.SketchManager.InsertSketch True
part.ClearSelection2 True

' eksen atama
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.SketchUseEdge2(False)
part.ClearSelection2 True
part.SketchManager.InsertSketch True

boolstatus = part.Extension.SelectByID2("Point9@Sketch8", "EXTSKETCHPOINT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Point10@Sketch8", "EXTSKETCHPOINT", 0, 0, 0, True, 0, Nothing, 0)

part.InsertAxis2 True
boolstatus = part.Extension.SelectByID2("Sketch8", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

boolstatus = part.Extension.SelectByID2("Sketch7", "SKETCH", 0, 0, 0, False, 1, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch6", "SKETCH", 0, 0, 0, True, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Axis2", "AXIS", 0, 0, 0, True, 128, Nothing, 0)
Dim SweepFeature As Object
Set SweepFeature = part.FeatureManager.InsertProtrusionSwept3(False, True, 0, False, False, 0, 0, False, 0, 0, 2, 1, 1, 1, 0, 1)

' asagıda 2 mm uzatıyoruz amac dikisi parcanın icine sokmak, 2 mm montaj ortamından okunan bir deger, 5,6.. mm de olabilir
mesela, min 2 mm olmalı
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Sketch7", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, 0.002, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0
' ikinci kaynak dikışı

' asagıda if/else li secim yok cunku alt yuzey hep aynı, sabit

```

```

If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2 - cygs_m) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
Else
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2 - cygs_m) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
End If
part.SketchManager.InsertSketch True
part.ClearSelection2 True
If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2 - cygs_m) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
Else
boolstatus = part.Extension.SelectByID2("", "FACE", (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) /
2, (gys_5 - cygs_r - as_2 - cygs_m) / 2 + as_2, mesafe / 2, False, 0, Nothing, 0)
End If

boolstatus = part.SketchUseEdge2(False)
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Line32", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line25", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line26", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line27", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line28", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line29", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.EditDelete
part.SketchManager.InsertSketch True

boolstatus = part.Extension.SelectByID2("Line31@Sketch9", "EXTSKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlanePerCurveAndPassPoint3 False, True

boolstatus = part.Extension.SelectByID2("Plane3", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True

' """"""ÖNEMLİ... Asagıda 10 mm eklememizin amacı kaynak dikisini büyük yapıp parca icine girmesini saglamak..
' ve böylece c sacının sırt ı ile cygs nin birlestigi yerdeki kaynak dikişlerinin bosluklarını ortadan kaldırmak
swApp.SetUserPreferenceToggle swSketchInference, False
part.CreateLine2 0.01, 0, 0, 0.01, -(Kaynak_Dikis_Boyu + 0.01), 0
part.CreateLine2 0.01, -(Kaynak_Dikis_Boyu + 0.01), 0, -Kaynak_Dikis_Boyu, 0, 0
part.CreateLine2 -Kaynak_Dikis_Boyu, 0, 0, 0.01, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.SketchAddConstraints "sgPERPENDICULAR"

C_ye_gelen_sac.Print Sqr(cygs_m ^ 2 - cygs_k ^ 2), cygs_parca_kalnlığı

part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch10", "SKETCH", 0, 0, 0, False, 1, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch9", "SKETCH", 0, 0, 0, True, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Axis2", "AXIS", 0, 0, 0, True, 128, Nothing, 0)

Set SweepFeature = part.FeatureManager.InsertProtrusionSwept3(False, True, 0, False, False, 0, 0, False, 0, 0, 0, 2, 1, 1, 1, 0, 1)

part.SaveAs2 fl0 + "C ye Gelen Sac.SLDPRT", 0, False, False

End Sub

Private Sub Command2_Click()
C_ye_gelen_sac.Hide

```

```

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazilacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
parca_listesi.AddItem "Borc Federi"
Case Is = 21
parca_listesi.AddItem "Borc Takviyesi"
'Case Else
' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()
'Combo1.AddItem "gys_6"
'Combo1.AddItem "gys_5"
'Combo1.AddItem "gys_1"
'Combo1.AddItem "gys_3"
'Combo1.AddItem "r1"
'Combo1.AddItem "gys_4"
'Combo1.AddItem "r2"

```

```

'Combo1.AddItem "cs_8"
'Combo1.AddItem "gys_2"
'Combo1.AddItem "cs_10"
'Combo1.AddItem "cs_3"
'Combo1.AddItem "cs_2"
'Combo1.AddItem "cs_1"
'Combo1.AddItem "cs_r1"
Combo1.AddItem "cygs_KM"
Combo1.AddItem "cygs_parca_kalnlığı"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "ks_parca_kalnlığı"
'Combo1.AddItem "ct_parca_boyu"
'Combo1.AddItem "ms_parca_kalnlığı"
'Combo1.AddItem "as_1"
'Combo1.AddItem "as_2"
'Combo1.AddItem "cygs_ms_boyu"
'Combo1.AddItem "cs_parca_kalnlığı"
'Combo1.AddItem "as_parca_kalnlığı"
'Combo1.AddItem "C_Muhafaza_Sacı_KM"

parca_listesi.Visible = False
End Sub

Private Sub mnu_CyeGelenDosya_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "cygs_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit
Public boolstatus As Boolean
Public fl0 As String
Public flasmb As String, flsw As String
Public swApp As Object, asmb1 As Object, comp As String

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Rem Public parca_kalnlığı As Double
Rem Public gys_1, gys_2, gys_3, gys_4, gys_5, gys_6, gys_7, gys_8, gys_9, gys_10 As Double
Rem Public gys_11, gys_12, gys_13, gys_14, gys_15, gys_16, gys_17 As Double
Rem Public r1, r2, r3, r4, r5, r6 As Double
Rem Public l1, pi, teta1, x1, y1, x2, y2, x3, y3, x4, y4, gys_merk_x, gys_merk_y As Double
Private Sub Command2_Click()
gys_sag1.Hide
End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik , en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8

```

```

    parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
    parca_listesi.AddItem "Ön Yatak"
Case Is = 10
    parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
    parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
    parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
    parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
    parca_listesi.AddItem "Yan Kapak"
Case Is = 15
    parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
    parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
    parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
    parca_listesi.AddItem "Burç Federi"
Case Is = 21
    parca_listesi.AddItem "Burç Takviyesi"
'Case Else
    ' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()
Show
fl0 = "D:\Dirinler_Makina A.Ş\GövDe\"
flsw = "C:\Program Files\SolidWorks\lang\english\Tutorial\"

Combo1.AddItem "gys_1"
Combo1.AddItem "gys_2"
Combo1.AddItem "gys_3"
Combo1.AddItem "gys_4"
Combo1.AddItem "gys_5"
Combo1.AddItem "gys_6"
Combo1.AddItem "gys_7"
Combo1.AddItem "gys_9"
Combo1.AddItem "gys_10"
Combo1.AddItem "gys_11"
Combo1.AddItem "gys_12"
Combo1.AddItem "gys_13"
Combo1.AddItem "gys_14"
Combo1.AddItem "gys_15"
Combo1.AddItem "gys_16"
Combo1.AddItem "gys_17"
Combo1.AddItem "r1"
Combo1.AddItem "r2"
Combo1.AddItem "r3"
Combo1.AddItem "r4"
Combo1.AddItem "r5"
Combo1.AddItem "r6"
'Combo1.AddItem "ap_parca_kalnlığı"
'Combo1.AddItem "eksen_1"
Combo1.AddItem "parca_kalnlığı"

parca_listesi.Visible = False

End Sub

```


Private Sub Command1_Click()

Call VeriOkuma 'Yandaki kod yardımı ile başka bir Subroutine çağırılarak programın kullandığı parametreler okunur
Bkz.Şek.3.8

Set swApp = GetObject("sldworks.application") ' Yandaki kod takımı ile SolidWorks Programında
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#) 'Part dosyası açılır ve yeni bir Part Dosyası oluşturmak
Set part = swApp.ActiveDoc ' mümkün olur

Call snap ' Snap subroutine'ini ile Programdaki otomatik ilişkilendirmeler kapatılır.

Dim msg As String

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, 0, 0, gys_1, 0, 0
part.ViewZoomtofit2
part.CreateLine2 gys_1, 0, 0, gys_1, gys_2, 0
part.CreateLine2 gys_1, gys_2, 0, (gys_1 - gys_3), gys_2, 0
part.CreateArc2 (gys_1 - gys_3), (gys_2 + r1), 0, (gys_1 - gys_3), gys_2, 0, (gys_3 + r1), (gys_2 + r1), 0, -1
part.ViewZoomtofit2
part.CreateLine2 (gys_1 - gys_3 - r1), (gys_2 + r1), 0, (gys_1 - gys_3 - r1), gys_4, 0

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)

part.CreateArc2 gys_merk_x, gys_merk_y, 0, xp2, yp2, 0, xp1, yp1, 0, -1
part.ViewZoomtofit2
part.CreateLine2 gys_6, gys_5, 0, gys_7, gys_5, 0
part.ViewZoomtofit2
boolstatus = part.Extension.SelectByID2("Point8", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r3, 1
part.ClearSelection2 True

part.CreateLine2 gys_7, gys_5, 0, gys_7, gys_2 + ap_parca_kalınlığı + eksen_1 - eksen_3, 0
part.CreateLine2 gys_7, gys_2 + ap_parca_kalınlığı + eksen_1 - eksen_3, 0, gys_9, gys_2 + ap_parca_kalınlığı + eksen_1 - eksen_3, 0
part.CreateLine2 gys_9, gys_2 + ap_parca_kalınlığı + eksen_1 - eksen_3, 0, gys_9, gys_10, 0

part.CreateLine2 gys_9, gys_10, 0, gys_11, gys_10, 0
part.CreateLine2 gys_11, gys_10, 0, gys_11, gys_15, 0
part.ViewZoomtofit2
part.CreateLine2 0, 0, 0, 0, gys_12, 0
part.CreateLine2 0, gys_12, 0, gys_13, gys_16, 0

part.CreateLine2 gys_13, gys_16, 0, gys_13, gys_14, 0
part.CreateLine2 gys_13, gys_14, 0, gys_17, gys_14, 0
part.CreateLine2 gys_17, gys_14, 0, gys_11, gys_15, 0

boolstatus = part.Extension.SelectByID2("Point20", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r6, 1
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Point22", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r5, 1
part.ClearSelection2 True
part.ViewZoomtofit2
boolstatus = part.Extension.SelectByID2("Point18", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r4, 1
part.ClearSelection2 True

```

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, parca_kalınlgı, 0, False, False, False, 0, 0, False, False,
False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", gys_2, gys_2, parca_kalınlgı, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True

boolstatus = part.Extension.SelectByID2("", "FACE", (gys_6 + (gys_3 / 2)), gys_2, parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", (gys_1 - gys_3 - r1), (gys_4 - r1), parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True

boolstatus = part.Extension.SelectByID2("", "FACE", gys_1, gys_2 / 2, parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", (gys_7 - gys_6) / 2 + gys_6, gys_5, parca_kalınlgı / 2, True, 0, Nothing,
0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", gys_9, ((gys_10 - (gys_2 + ap_parca_kalınlgı + eksen_1 - eksen_3)) / 2)
+ (gys_2 + ap_parca_kalınlgı + eksen_1 - eksen_3), parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", gys_1 / 2, gys_10, parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", gys_7, (gys_2 + ap_parca_kalınlgı + eksen_1 - eksen_3 - gys_5) / 2 +
gys_5, parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", (gys_13 + (gys_17 - gys_13) / 2), gys_14, parca_kalınlgı / 2, True, 0,
Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

.....
....."KAYNAKLI BÖLÜM".....
.....

Call Kaynak

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
part.SketchRectangle -(parca_kalınlgı), 0, 0, -(parca_kalınlgı - parcalar_arasi_bosluk), gys_10, 0, 0

part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, (gys_1 * 3 / 2), 0, False, False, False, False, 0, 0, False, False, False,
False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0
part.ClearSelection2 True

part.SaveAs2 fl0 + "Govde Yan Sacı Sag_M.SLDPRT", 0, False, False
End Sub

Private Sub mnu_dosya_Click()

```

```

Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "gys_sag_parametrik_degerler.txt", vbNormalFocus)
End Sub

```

```

Option Explicit
Public boolstatus As Boolean
Public fl0 As String
Public flasmb As String, flsw As String
Public swApp As Object, asmb1 As Object, comp As String
Rem Public parca_kalinligi As Double
Rem Public gys_1, gys_2, gys_3, gys_4, gys_5, gys_6, gys_7, gys_8, gys_9, gys_10 As Double
Rem Public gys_11, gys_12, gys_13, gys_14, gys_15, gys_16, gys_17 As Double
Rem Public r1, r2, r3, r4, r5, r6 As Double
Rem Public l1, pi, tet1, x1, y1, x2, y2, x3, y3, x4, y4, gys_merk_x, gys_merk_y As Double

```

```

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

```

```

Private Sub Command3_Click()
parca_listesi.Visible = True

```

```

Dim gir As String
gir = Combo1.Text

```

```

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik , en cok olan deger yazilacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
parca_listesi.AddItem "Burc Federi"
Case Is = 21
parca_listesi.AddItem "Burc Takviyesi"

```

```

'Case Else
  'parca_listesi.AddItem "Aradığınız Degisken Baska Bir Parçayı Etkilememtedir."

  End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()
Show
fl0 = "D:\Dirinler_Makina A.Ş\GövDe\"
flsw = "C:\Program Files\SolidWorks\lang\english\Tutorial\"

  Combo1.AddItem "gys_1"
  Combo1.AddItem "gys_2"
  Combo1.AddItem "gys_3"
  Combo1.AddItem "gys_4"
  Combo1.AddItem "gys_5"
  Combo1.AddItem "gys_6"
  Combo1.AddItem "gys_7"
  Combo1.AddItem "gys_9"
  Combo1.AddItem "gys_10"
  Combo1.AddItem "gys_11"
  Combo1.AddItem "gys_12"
  Combo1.AddItem "gys_13"
  Combo1.AddItem "gys_14"
  Combo1.AddItem "gys_15"
  Combo1.AddItem "gys_16"
  Combo1.AddItem "gys_17"
  Combo1.AddItem "r1"
  Combo1.AddItem "r2"
  Combo1.AddItem "r3"
  Combo1.AddItem "r4"
  Combo1.AddItem "r5"
  Combo1.AddItem "r6"
  'Combo1.AddItem "ap_parca_kalınligı"
  'Combo1.AddItem "eksen_1"
  Combo1.AddItem "parca_kalınligı"

  parca_listesi.Visible = False

End Sub

Private Sub Command1_Click()

  Call VeriOkuma

  Set swApp = GetObject( "sldworks.application")
  Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
  Set part = swApp.ActiveDoc

  Call snap

  Dim msg As String

  part.SketchManager.InsertSketch True ' Sketch ortamında yeni bir sketch açılmasını sağlayan komuttur.
  boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0) ' Sketch ortamında bir düzlem-
  '(plane) seçilmesini sağlayan komuttur.
  part.CreateLine2 0, 0, 0, gys_1, 0, 0 ' Çizgi çizilmesini sağlayan komuttur.
  part.ViewZoomtofit2 ' Görüntüyü ekrana fit hale getirir.
  part.CreateLine2 gys_1, 0, 0, gys_1, gys_2, 0
  part.CreateLine2 gys_1, gys_2, 0, (gys_1 - gys_3), gys_2, 0
  part.CreateArc2 (gys_1 - gys_3), (gys_2 + r1), 0, (gys_1 - gys_3), gys_2, 0, (gys_3 + r1), (gys_2 + r1), 0, -1
  part.ViewZoomtofit2
  part.CreateLine2 (gys_1 - gys_3 - r1), (gys_2 + r1), 0, (gys_1 - gys_3 - r1), gys_4, 0

  xp1 = gys_6
  yp1 = gys_5
  xp2 = gys_1 - gys_3 - r1

```

```
yp2 = gys_4
R = r2
```

```
Call newtonR
```

```
gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
```

```
part.CreateArc2 gys_merk_x, gys_merk_y, 0, xp2, yp2, 0, xp1, yp1, 0, -1
part.ViewZoomtofit2
part.CreateLine2 gys_6, gys_5, 0, gys_7, gys_5, 0
part.ViewZoomtofit2
boolstatus = part.Extension.SelectByID2("Point8", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r3, 1
part.ClearSelection2 True
```

```
part.CreateLine2 gys_7, gys_5, 0, gys_7, gys_2 + ap_parca_kalınlgı + eksen_1 - eksen_3, 0
part.CreateLine2 gys_7, gys_2 + ap_parca_kalınlgı + eksen_1 - eksen_3, 0, gys_9, gys_2 + ap_parca_kalınlgı + eksen_1 -
eksen_3, 0
part.CreateLine2 gys_9, gys_2 + ap_parca_kalınlgı + eksen_1 - eksen_3, 0, gys_9, gys_10, 0
```

```
part.CreateLine2 gys_9, gys_10, 0, gys_11, gys_10, 0
part.CreateLine2 gys_11, gys_10, 0, gys_11, gys_15, 0
part.ViewZoomtofit2
part.CreateLine2 0, 0, 0, 0, gys_12, 0
```

```
part.CreateLine2 0, gys_12, 0, gys_13, gys_16, 0
```

```
part.CreateLine2 gys_13, gys_16, 0, gys_13, gys_14, 0
part.CreateLine2 gys_13, gys_14, 0, gys_17, gys_14, 0
part.CreateLine2 gys_17, gys_14, 0, gys_11, gys_15, 0
```

```
boolstatus = part.Extension.SelectByID2("Point20", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r6, 1
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Point22", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r5, 1
part.ClearSelection2 True
part.ViewZoomtofit2
boolstatus = part.Extension.SelectByID2("Point18", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 r4, 1
part.ClearSelection2 True
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, parca_kalınlgı, 0, False, False, False, False, 0, 0, False, False,
False, False, 1, 1, 1, 0, 0, False
```

```
'part.SketchManager.InsertSketch True
'Call part.Extension.SelectByID2("", "FACE", 0.002, -0.003, 0.25, False, 0, Nothing, 0)
'part.ClearSelection2 True
```

```
'part.CreateArc2 1.083, 1.542, 0, 1.075, 1.542, 0, 1.091, 1.542, 0, -1
'part.CreateLine2 1.05, 1.47, 0, 1.15, 1.47, 0
'part.CreateLine2 1.075, 1.542, 0, 1.075, 1.47, 0
'part.CreateLine2 1.091, 1.542, 0, 1.091, 1.47, 0
'part.CreateArc2 1.083, 1.398, 0, 1.091, 1.398, 0, 1.075, 1.398, 0, -1
```

```
boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", (gys_7 - gys_6) / 2 + gys_6, gys_5, parca_kalınlgı / 2, True, 0, Nothing,
0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
```

```

boolstatus = part.Extension.SelectByID2("", "FACE", gys_7, (gys_2 + ap_parca_kalınligı + eksen_1 - eksen_3 - gys_5) / 2 +
gys_5, parca_kalınligı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", gys_11, gys_2 + ap_parca_kalınligı + eksen_1 - eksen_3, parca_kalınligı
/ 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", gys_9 / 2, gys_10, parca_kalınligı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 parca_kalınligı, False, True

```

```

.....
....."KAYNAKLI BÖLÜM".....
.....

```

Call Kaynak

```

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

```

```

part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, -parcalar_arasi_bosluk, gys_10, 0, 0

```

```

part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, (gys_1 * 3 / 2), 0, False, False, False, False, 0, 0, False, False, False,
False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0
part.ClearSelection2 True

```

```

part.SaveAs2 fl0 + "Govde Yan Sacı Sol_M.SLDPRT", 0, False, False

```

```

End Sub
Private Sub Command2_Click()
gys_sol1.Hide

```

```

End Sub

```

```

Private Sub mnu_dosya2_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "gys_sol_parametrik_degerler.txt", vbNormalFocus)
End Sub

```

```

Private Sub Command1_Click()

```

```

OKUD = Val(Kaynak_Dikis_Boyutlari.Text1.Text)
OKAD = Val(Kaynak_Dikis_Boyutlari.Text2.Text)
AKOD = Val(Kaynak_Dikis_Boyutlari.Text3.Text)
AKAD = Val(Kaynak_Dikis_Boyutlari.Text4.Text)
N4SD = Val(Kaynak_Dikis_Boyutlari.Text5.Text)
CGSD = Val(Kaynak_Dikis_Boyutlari.Text6.Text)
PGSD = Val(Kaynak_Dikis_Boyutlari.Text7.Text)
GYSN3N4AD = Val(Kaynak_Dikis_Boyutlari.Text8.Text)
CSKSAD = Val(Kaynak_Dikis_Boyutlari.Text9.Text)

```

```

If OKUD < 0.3 Or OKUD > 0.7 Then
MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
Kaynak_Dikis_Boyutlari.Text1.SetFocus ' hata oldugunda ilgili text kutusuna gider

```

```

GoTo 10
End If
If OKAD < 0.3 Or OKUD > 0.7 Then
MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
Kaynak_Dikis_Boyutlari.Text2.SetFocus
GoTo 10

```

```

End If
If AKOD < 0.3 Or OKUD > 0.7 Then
  MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
  Kaynak_Dikis_Boyutlari.Text3.SetFocus
GoTo 10
End If
If AKAD < 0.3 Or OKUD > 0.7 Then
  MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
  Kaynak_Dikis_Boyutlari.Text4.SetFocus
GoTo 10
End If
If N4SD < 0.3 Or OKUD > 0.7 Then
  MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
  Kaynak_Dikis_Boyutlari.Text5.SetFocus
GoTo 10
End If
If CGSD < 0.3 Or OKUD > 0.7 Then
  MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
  Kaynak_Dikis_Boyutlari.Text6.SetFocus
GoTo 10
End If
If PGSD < 0.3 Or OKUD > 0.7 Then
  MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
  Kaynak_Dikis_Boyutlari.Text7.SetFocus
GoTo 10
End If
If GYSN3N4AD < 0.3 Or OKUD > 0.7 Then
  MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
  Kaynak_Dikis_Boyutlari.Text8.SetFocus
GoTo 10
End If
If CSKSAD < 0.3 Or OKUD > 0.7 Then
  MsgBox ("0.3 ile 0.7 Arasında Bir Değer Giriniz")
  Kaynak_Dikis_Boyutlari.Text9.SetFocus
GoTo 10
End If

If Val(Kaynak_Dikis_Boyutlari.Text6.Text) <> 0.5 Then
xxx = MsgBox("C ye GELEN SAC ISIMLI PARCAYI MONTAJ ISLEMINE BASLAMADAN ONCE TEKRAR
OLUSTURUNUZ", vbExclamation)
End If

If Val(Kaynak_Dikis_Boyutlari.Text7.Text) <> 0.5 Then
xxx = MsgBox("P ye GELEN SAC ISIMLI PARCAYI MONTAJ ISLEMINE BASLAMADAN ONCE TEKRAR
OLUSTURUNUZ", vbExclamation)
End If

Open "D:\Dirinler_Makina A.Ş\GövDe\Kaynak_Dikis_Degerleri.txt" For Output As 1

Write #1, OKUD
Write #1, OKAD
Write #1, AKOD
Write #1, AKAD
Write #1, N4SD
Write #1, CGSD
Write #1, PGSD
Write #1, GYSN3N4AD
Write #1, CSKSAD
Close #1

10
End Sub

Private Sub Command2_Click()
Dim kaynak_cevap

kaynak_cevap = MsgBox("Yeni Kaynak Dikis Boyutlarını Kaydettiniz mi?", vbQuestion + vbYesNo, "Kaydetme Uyarısı")
If kaynak_cevap = vbYes Then
GoTo 20
ElseIf kaynak_cevap = vbNo Then
GoTo 30

```

```

End If

20 Kaynak_Dikis_Boyutlari.Hide

30
End Sub

Private Sub Form_Load()

End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

No_1_parca_kalnlığı = 0.016
No_2_parca_kalnlığı = 0.016
No_2_3_parca_kalnlığı = 0.016
No_3_parca_kalnlığı = 0.016
uzaklık = gys_9 - öyms_parca_kalnlığı - (X_1 + No_1_parca_kalnlığı + X_3 + No_2_parca_kalnlığı + No_3_parca_kalnlığı
+ X_5 + no_4s_parca_kalnlığı + X_6 + no_5s_parca_kalnlığı + X_7)

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

Öteleme_x = ((gys_1 - gys_3 - r1) - cs_8)
Öteleme_y = (gys_2 - cs_10)

part.CreateLine2 cs_parca_kalnlığı, 0, 0, cs_parca_kalnlığı, cs_1 + Öteleme_y - gys_5, 0
part.ViewZoomtofit2
part.CreateLine2 cs_parca_kalnlığı, cs_1 + Öteleme_y - gys_5, 0, 0, cs_1 + Öteleme_y - gys_5, 0
part.CreateLine2 0, cs_1 + Öteleme_y - gys_5, 0, 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0
part.ViewZoomtofit2
part.CreateLine2 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, (mesafe - 2 * oy_r2) / 2, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalnlığı)), 0
part.CreateArc2 (mesafe / 2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, (mesafe / 2 + oy_r2), (eksen_1 - (gys_5 -
gys_2 - ap_parca_kalnlığı)), 0, (mesafe - 2 * oy_r2) / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, -1
part.CreateLine2 (mesafe / 2 + oy_r2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, mesafe, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalnlığı)), 0
part.ViewZoomtofit2
part.CreateLine2 mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, mesafe, cs_1 + Öteleme_y - gys_5, 0
part.CreateLine2 mesafe, cs_1 + Öteleme_y - gys_5, 0, mesafe - cs_parca_kalnlığı, cs_1 + Öteleme_y - gys_5, 0
part.CreateLine2 mesafe - cs_parca_kalnlığı, cs_1 + Öteleme_y - gys_5, 0, mesafe - cs_parca_kalnlığı, 0, 0
part.CreateLine2 mesafe - cs_parca_kalnlığı, 0, 0, cs_parca_kalnlığı, 0, 0
part.ViewZoomtofit2
' burada asagidaki extrusion yöntemini kullanıyoruz cunku yarım cemberin merkez noktası sagındaki ve solundaki çizgilerle
aynı hizada old. icin coincident görüyo,ve diger yöntem calismiyo...
part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", mesafe / 2, ((eksen_1 - (gys_5 - gys_2 -
ap_parca_kalnlığı)) / 2), 0, True, 4, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, ks_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

```



```

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) / 2,
ks_parca_kalnlığı, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) - oy_r2,
ks_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.InsertAxis2 True

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 mesafe, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), False, True
part.ClearSelection2 True

part.SaveAs2 fl0 + "Kızak Sacı.SLDPRT", 0, False, False

*****
*****"NO 1 ALIN SACI"*****
*****

Call VeriOkuma

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

No_1_parca_kalnlığı = 0.016
No_2_parca_kalnlığı = 0.016
No_3_parca_kalnlığı = 0.016

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, (mesafe - 2 * oy_r2) / 2, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalnlığı)), 0
part.ViewZoomtofit2
part.CreateArc2 (mesafe / 2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, (mesafe / 2 + oy_r2), (eksen_1 - (gys_5 -
gys_2 - ap_parca_kalnlığı)), 0, (mesafe - 2 * oy_r2) / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, 1
part.ViewZoomtofit2
part.CreateLine2 (mesafe / 2 + oy_r2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, mesafe, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalnlığı)), 0
'Asagıdaki çizimin ikinci noktasında kullanılan y degerinde 10 mm çıkarılmasının sebebi,parcanın ust kısmı ile GYS nin üstü
arası 10 mm bulunmasıdır.yani "üst_girinti"
part.CreateLine2 mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, mesafe, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 - üst_girinti), 0
part.ViewZoomtofit2
part.CreateLine2 mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 -
üst_girinti), 0, 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 -
üst_girinti), 0
part.CreateLine2 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 -
üst_girinti), 0, 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0

part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", mesafe / 2, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalnlığı)) + ((gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 - üst_girinti) / 2), 0, True, 4, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No_1_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + oy_r2,
No_1_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.InsertAxis2 True

```

```
boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 12, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (oy_r2 * 2), No_1_parca_kalnlığı, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
```

```
part.SaveAs2 fl0 + "No 1 Alın Sacı.SLDPRT", 0, False, False
```

```

.....
....."NO 2 SACI".....
.....

```

Call VeriOkuma

```
Set swApp = GetObject("sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc
```

Call snap

```
No_2_parca_kalnlığı = 0.016
No_3_parca_kalnlığı = 0.016
```

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.CreateLine2 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, (mesafe - 2 * oy_r2) / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0
part.ViewZoomtofit2
part.CreateArc2 (mesafe / 2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, (mesafe / 2 + oy_r2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, (mesafe - 2 * oy_r2) / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, 1
part.ViewZoomtofit2
```

```
part.CreateLine2 (mesafe / 2 + oy_r2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0
'Asagıdaki çizgimin ikinci noktasında kullanılan y degerinde 10 mm çıkarılmasının sebebi,parcanın ust kısmı ile GYS nin üstü arası 10 mm bulunmasıdır.
```

```
part.CreateLine2 mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0, mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 - üst_girinti), 0
part.ViewZoomtofit2
```

```
part.CreateLine2 mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 - üst_girinti), 0, 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 - üst_girinti), 0
```

```
part.CreateLine2 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 - üst_girinti), 0, 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), 0
part.ViewZoomtofit2
```

```
part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", mesafe / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + ((gys_10 - eksen_1 - ap_parca_kalnlığı - gys_2 - üst_girinti) / 2), 0, True, 4, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No_2_parca_kalnlığı, 0, False, False, False, False, 0, 0, False, False, False, 1, 1, 1, 0, 0, False
part.SelectionManager.EnableContourSelection = 0
```

```
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 5, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)), No_2_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
```

```
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Extrude1", "BODYFEATURE", 0, 0, 0, True, 1, Nothing, 0)
part.FeatureManager.InsertMirrorFeature False, False, False, False
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + oy_r2, No_2_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.InsertAxis2 True
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalnlığı)) + (oy_r2 * 2), No_2_parca_kalnlığı, True, 0, Nothing, 0)
```

```

part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

part.SaveAs2 fl0 + "No 2 Sacı.SLDPRT", 0, False, False

.....
....."2-3 NOLU ARA SAC".....
.....

Call VeriOkuma

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

No_2_3_parca_kalınlığı = 0.016

uzaklık = gys_9 - öyms_parca_kalınlığı - (X_1 + No_1_parca_kalınlığı + X_3 + No_2_parca_kalınlığı + No_3_parca_kalınlığı
+ X_5 + no_4s_parca_kalınlığı + X_6 + no_5s_parca_kalınlığı + X_7)

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, mesafe, uzaklık, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No_2_3_parca_kalınlığı, 0, False, False, False, False, 0, 0,
False, False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, uzaklık / 2, No_2_3_parca_kalınlığı, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

.....
....."KAYNAKLI BÖLÜM".....
.....

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, -No_2_3_parca_kalınlığı, parcalar_arasi_bosluk, 0, 0
part.SketchRectangle 0, uzaklık, 0, -No_2_3_parca_kalınlığı, uzaklık - parcalar_arasi_bosluk, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0

part.SaveAs2 fl0 + "2_3 NOLU ARA SAC.SLDPRT", 0, False, False
.....
....."NO 3 SACI".....
.....

Call VeriOkuma

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

Call snap

No_3_parca_kalınlgı = 0.016

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)), 0, (mesafe - 2 * ay_r2) / 2, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalınlgı)), 0
part.ViewZoomtofit2
part.CreateArc2 (mesafe / 2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)), 0, (mesafe / 2 + ay_r2), (eksen_1 - (gys_5 -
gys_2 - ap_parca_kalınlgı)), 0, (mesafe - 2 * ay_r2) / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)), 0, 1
part.ViewZoomtofit2
part.CreateLine2 (mesafe / 2 + ay_r2), (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)), 0, mesafe, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalınlgı)), 0
'Asagıdaki çizgimin ikinci noktasında kullanılan y degerinde 10 mm çıkarılmasının sebebi,parcanın ust kısmı ile GYS nin üstü
arası 10 mm bulunmasıdır.
part.CreateLine2 mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)), 0, mesafe, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalınlgı)) + (gys_10 - eksen_1 - ap_parca_kalınlgı - gys_2 - üst_girinti), 0
part.ViewZoomtofit2
part.CreateLine2 mesafe, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)) + (gys_10 - eksen_1 - ap_parca_kalınlgı - gys_2 -
üst_girinti), 0, 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)) + (gys_10 - eksen_1 - ap_parca_kalınlgı - gys_2 -
üst_girinti), 0
part.CreateLine2 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)) + (gys_10 - eksen_1 - ap_parca_kalınlgı - gys_2 -
üst_girinti), 0, 0, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)), 0
part.ViewZoomtofit2

part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", mesafe / 2, (eksen_1 - (gys_5 - gys_2 -
ap_parca_kalınlgı)) + ((gys_10 - eksen_1 - ap_parca_kalınlgı - gys_2 - üst_girinti) / 2), 0, True, 4, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No_3_parca_kalınlgı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 5, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)),
No_3_parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Extrude1", "BODYFEATURE", 0, 0, 0, True, 1, Nothing, 0)
part.FeatureManager.InsertMirrorFeature False, False, False, False

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)) + ay_r2,
No_3_parca_kalınlgı / 2, True, 0, Nothing, 0)
part.InsertAxis2 True

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 No_3_parca_kalınlgı, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 5, (eksen_1 - (gys_5 - gys_2 - ap_parca_kalınlgı)) + (gys_10 -
eksen_1 - ap_parca_kalınlgı - gys_2 - üst_girinti), No_3_parca_kalınlgı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

part.SaveAs2 f10 + "No 3 Sacı.SLDPRT", 0, False, False

```

```

.....
....."KIZAKLAR".....
....."KARSI KIZAK".....

```

Call VeriOkuma

```
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc
```

```
Call snap
```

```
'karsı_kızak_parca_kalınlığı = 0.04
'yan_kızak_parca_kalınlığı = 0.03
'karsı_kızak_x = 0.15
```

```
!***** ÖNEMLİ OKU !!!!!!!*****
```

```
' aşağıda eksen 3'ü 5 ile çarptık çünkü eksen1 eksenini ile kızakın üst yüzeyi arası mesafe yaklaşık 65 mm dir..İstenirse bu değer değiştirilebilir.
```

```
' eksen_3 ün default değeri de 13 olduğu için onu 5 ile çarptık. 5 ile çarpmadan direkt eksen_3 yazarsak
```

```
' eksen_1 ile kızak üst yüzeyi arası kaynak yapmak için yeterli mesafe kalmayacaktır.
```

```
kızak_y = (gys_2 + ap_parca_kalınlığı + eksen_1) - gys_5 - eksen_3 * 5
```

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.SketchRectangle 0, 0, 0, karsı_kızak_x, kızak_y, 0, 0
part.ViewZoomtofit2
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, karsı_kızak_parca_kalınlığı, 0, False, False, False, False, 0, 0, False, False, False, False, 1, 1, 1, 0, 0, False
```

```
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 karsı_kızak_parca_kalınlığı, False, True
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 karsı_kızak_x, False, True
part.ClearSelection2 True
```

```
*****
*****"KAYNAKLI BÖLÜM"*****
*****
```

```
Kullanılan_Parca_Kalınlığı = karsı_kızak_parca_kalınlığı
x_factor = 0.2
Call Kaynak
```

```
boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, -parcalar_arasi_bosluk, kızak_y, 0, 0
part.SketchRectangle -karsı_kızak_parca_kalınlığı, 0, 0, -(karsı_kızak_parca_kalınlığı - parcalar_arasi_bosluk), kızak_y, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True
```

```
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
```

```
boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, karsı_kızak_x, 0, False, False, False, False, 0, 0, False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

```
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
```

```

part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, karsı_kızak_x, Kaynak_Dikis_Boyu, 0, 0
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle -parcalar_arasi_bosluk, 0, 0, -karsı_kızak_parca_kalnlığı - parcalar_arasi_bosluk, Kaynak_Dikis_Boyu,
0, 0
part.SketchRectangle -parcalar_arasi_bosluk, kızak_y, 0, -karsı_kızak_parca_kalnlığı - parcalar_arasi_bosluk, kızak_y -
Kaynak_Dikis_Boyu, 0, 0
swApp.SetUserPreferenceToggle swSketchInference, False

part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, 0.0001, 0.01, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.SaveAs2 fl0 + "Karsı Kızak.SLDPRPT", 0, False, False

*****YAN KIZAK*****

Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, yan_kızak_x, kızak_y, 0, 0
part.ViewZoomtofit2

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, yan_kızak_parca_kalnlığı, 0, False, False, False, False, 0, 0,
False, False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 yan_kızak_parca_kalnlığı, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 yan_kızak_x, False, True
part.ClearSelection2 True

```

```

.....
....."KAYNAKLI BÖLÜM".....
.....
Kullanilan_Parca_Kalinligi = yan_kizak_parca_kalinligi
x_factor = 0.2
Call Kaynak

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle 0, 0, 0, -yan_kizak_parca_kalinligi, Kaynak_Dikis_Boyu, 0, 0
part.SketchRectangle 0, kizak_y, 0, -yan_kizak_parca_kalinligi, kizak_y - Kaynak_Dikis_Boyu, 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle -parcalar_arasi_bosluk, 0, 0, yan_kizak_x, Kaynak_Dikis_Boyu, 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.SaveAs2 fl0 + "Yan Kizak.SLDPR2", 0, False, False

.....
....."NO 1".....
.....

Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

No1_parca_kalinligi = 0.016

No1_y = 0.15

part.SketchManager.InsertSketch True

```

```

boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, X_3, No1_y, 0, 0
part.ViewZoomtofit2

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No1_parca_kalinligi, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 No1_parca_kalinligi, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 X_3, False, True
part.ClearSelection2 True

=====
"KAYNAKLI BÖLÜM"
=====

Kullanilan_Parca_Kalinligi = No1_parca_kalinligi
x_factor = 0.5
Call Kaynak

boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, parcalar_arasi_bosluk, -No1_parca_kalinligi, 0, 0
part.ClearSelection2 True
part.SketchRectangle X_3, 0, 0, X_3 - parcalar_arasi_bosluk, -No1_parca_kalinligi, 0, 0

swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

part.FeatureManager.FeatureCut True, False, True, 0, 0, No1_y, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, X_3, -parcalar_arasi_bosluk, 0, 0
part.SketchRectangle 0, -No1_parca_kalinligi, 0, X_3, -(No1_parca_kalinligi - parcalar_arasi_bosluk), 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

part.FeatureManager.FeatureCut True, False, True, 0, 0, No1_y, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0

```


'-----ASAGIDAKI FEATURE ALIN KAYNAGIDIR-----'

```
boolstatus = part.Extension.SelectByID2("Plane2", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
```

```
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle -(No1_parca_kalnlığı - parcalar_arasi_bosluk), 0, 0, -(No1_parca_kalnlığı - Kaynak_Dikis_Boyu),
No1_y, 0, 0
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
```

```
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

```
part.SaveAs2 fl0 + "No 1.SLDPRT", 0, False, False
```

```

=====
=====1_2 Ara Kapama Sacı=====
=====

```

Call VeriOkuma

```
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc
```

Call snap

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.SketchRectangle 0, 0, 0, mesafe, X_3, 0, 0
part.ViewZoomtofit2
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No_1_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
```

```
boolstatus = part.Extension.SelectByID2("", "EDGE", 0, 0, No_1_parca_kalnlığı / 2, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "EDGE", 0, X_3, No_1_parca_kalnlığı / 2, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "EDGE", mesafe, X_3, No_1_parca_kalnlığı / 2, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "EDGE", mesafe, 0, No_1_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.FeatureManager.InsertFeatureChamfer 4, 1, 0.015, 0.7853981633975, 0, 0, 0, 0
```

```
boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 No_1_parca_kalnlığı, False, True
```

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.ClearSelection2 True
```

```
part.CreateLine2 0, 0, 0, 0.015, 0, 0
part.CreateLine2 0.015, 0, 0, 0, 0.015, 0
part.CreateLine2 0, 0.015, 0, 0, 0, 0
```

```
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.SketchAddConstraints "sgPERPENDICULAR"
part.ClearSelection2 True
```



```

boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, mesafe, X_5, 0, 0
part.ViewZoomtofit2

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No_1_parca_kalınligı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "EDGE", 0, 0, No_1_parca_kalınligı / 2, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "EDGE", 0, X_5, No_1_parca_kalınligı / 2, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "EDGE", mesafe, X_5, No_1_parca_kalınligı / 2, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "EDGE", mesafe, 0, No_1_parca_kalınligı / 2, True, 0, Nothing, 0)
part.FeatureManager.InsertFeatureChamfer 4, 1, 0.01, 0.7853981633975, 0, 0, 0, 0

boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 No_1_parca_kalınligı, False, True

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.ClearSelection2 True

part.CreateLine2 0, 0, 0, 0.01, 0, 0
part.CreateLine2 0.01, 0, 0, 0, 0.01, 0
part.CreateLine2 0, 0.01, 0, 0, 0, 0

part.ClearSelection2 True
Call part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, No_1_parca_kalınligı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

.....
....."KAYNAKLI BÖLÜM".....
.....

Kullanilan_Parca_Kalınligı = No1_parca_kalınligı
x_factor = 0.5
Call Kaynak

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, mesafe, parcalar_arasi_bosluk, 0, 0
part.SketchRectangle 0, X_5, 0, mesafe, X_5 - parcalar_arasi_bosluk, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, No_1_parca_kalınligı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True
'asagıdaki 0.099 yukarıda ki champfer degerinden geliyor 10 olması gerekiyor aslında ancak çizim ortamında 9.9 olduğu için
böyle alınmıştır. bu deger champfer degeri degisirse degismelidir.
part.SketchRectangle 0.0099, 0, 0, mesafe - (0.0099), -Kaynak_Dikis_Boy, 0, 0

part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, False, 0, 0, X_5, 0, False, False, False, False, 0, 0, False, False, False, False,
1, 1, 1
part.SelectionManager.EnableContourSelection = 0

```

```
part.SaveAs2 fl0 + "3_4 Ara Kapama Sacı.SLDPRT", 0, False, False
```

```
End Sub
```

```
Private Sub Command2_Click()
Kızak_Sacı.Hide
```

```
End Sub
```

```
Private Sub Command3_Click()
parca_listesi.Visible = True
```

```
Dim gir As String
gir = Combo1.Text
```

```

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik , en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
parca_listesi.AddItem "Burd Federi"
Case Is = 21
parca_listesi.AddItem "Burd Takviyesi"
Case Else
' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememektedir."

```

```

        End Select
    End If
Next j
Next i

End Sub

Private Sub Form_Load()
'Combo1.AddItem "gys_9"
'Combo1.AddItem "öyms_parca_kalınlığı"
'Combo1.AddItem "X_1"
'Combo1.AddItem "X_3"
'Combo1.AddItem "X_5"
'Combo1.AddItem "no_4s_parca_kalınlığı"
'Combo1.AddItem "X_6"
'Combo1.AddItem "no_5s_parca_kalınlığı"
'Combo1.AddItem "X_7"
'Combo1.AddItem "gys_1"
'Combo1.AddItem "gys_3"
'Combo1.AddItem "r1"
'Combo1.AddItem "cs_8"
'Combo1.AddItem "gys_2"
'Combo1.AddItem "cs_10"
'Combo1.AddItem "cs_parca_kalınlığı"
'Combo1.AddItem "cs_1"
'Combo1.AddItem "gys_5"
'Combo1.AddItem "eksen_1"
'Combo1.AddItem "ap_parca_kalınlığı"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "oy_r2"
Combo1.AddItem "ks_parca_kalınlığı"
'Combo1.AddItem "gys_10"
'Combo1.AddItem "üst_girinti"
'Combo1.AddItem "No_1_Parca_kalınlığı"
'Combo1.AddItem "No_2_Parca_kalınlığı"
'Combo1.AddItem "No_3_Parca_kalınlığı"
'Combo1.AddItem "uzaklık"
'Combo1.AddItem "ay_r2"
'Combo1.AddItem "No1_Parca_kalınlığı"

parca_listesi.Visible = False
End Sub

Private Sub mnu_kızaklar_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "kızaklar_parametrik_degerler.txt", vbNormalFocus)
End Sub

Private Sub mnu_kızaksacı_2_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "ks_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma

' Asagıdaki bilgi alt plaka montajında distance mate degeri olarak kullanılmaktadır.
'ap_parca_genisligi = mesafe + parca_kalınlığı * 2 + 0.105 * 2
Dim msg

min = 200
max = 3000

```

```

mesafe = Val(Montaj.Text4.Text) / 1000
eksen_1 = Val(Montaj.Text2.Text) / 1000
eksen_2 = Val(Montaj.Text3.Text) / 1000
eksen_3 = Val(Montaj.Text12.Text) / 1000
X_1 = Val(Montaj.Text1.Text) / 1000
X_2 = Val(Montaj.Text5.Text) / 1000
X_3 = Val(Montaj.Text6.Text) / 1000
X_4 = Val(Montaj.Text7.Text) / 1000
X_5 = Val(Montaj.Text9.Text) / 1000
X_6 = Val(Montaj.Text10.Text) / 1000
X_7 = Val(Montaj.Text11.Text) / 1000
Y_1 = Val(Montaj.Text8.Text) / 1000

If Val(Text1.Text) < min Or Val(Text1.Text) > max Then
  msg = MsgBox("Yan Plakalar Arasındaki Mesefe Değeri 200 mm'den Küçük 3000 mm'den Büyük Olamaz. Değeri Kontrol Ediniz", vbCritical + vbOKOnly, "Mesafe Hatası")
  Text1.SetFocus
  GoTo 250
End If

Set swApp = GetObject( "sldworks.application")
Set asmb1 = swApp.ActiveDoc

Call snap

Dim boolstatus As Boolean

Set asmb1 = Nothing
swApp.CloseDoc "Dirinler_Pres"
Call parcalistesi
Call parcaları_cagir
Call kaydet
asmb1.ViewZoomtofit2
asmb1.EditRebuild3
Call parcaların_lokasyonu

Form1.Cls

Rem XXXYYY Call asmb1.Extension.SelectByID2("plane1@part2-1@dirinler", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Rem XXXYYY Call asmb1.Extension.SelectByID2("plane1@part1-1@dirinler", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
'Call asmb1.AddMate2(5, 0, False, 0.0005, 0.0005, 0.0005, 0, 0, 0, 0, 0, longstatus, False)

'Call asmb1.AddMate3(5, 0, False, 0.0005, 0.0005, 0.0005, 0, 0, 0, 0, False, mateError)
'BU KOMUTTA KULLANILABİLİR =====>!!!!!!

' Add the mate

Rem XXXYYY Call asmb1.AddMate(5, 0, False, 0.0005, 0) '( 1.si mate tipi coince, 2.si antialign,3 sü flip yok..EGER Yuzeyin diger tarafından mate yapmak istiyorsan yani yuzeyi 180 derece ceviriip aynı yuzeyden mate yapmak istiyorsan 1. ci ye "0" 2.ci ye de "0" yaz diğerlerini deęiştirme)
' COK ONEMLI !!!!!!!
'!!!!!!!!!!!!Distance mate verirken arada bosluk bırakmak isteniyorsa birbirine bakan yuzeylere sahip parcalardan hareket edecek olanın pozitif degerde hareket edeceęi göz önüne alınmalı bu örnekte de olduęu gibi part 2 önce locate_part komutu ile part1 ile bağlanacağı yuzeyin altına cekilir sonra pozitif degerli distance komutu ile mate yapılır. NEGATIF DEGER TANIMLANAMIYOR !

Call asmb1.Extension.SelectByID2("Top@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.Extension.SelectByID2("Top@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.AddMate(0, 0, False, 0, 0)

Call asmb1.Extension.SelectByID2("Right@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.Extension.SelectByID2("Right@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.AddMate(0, 0, False, 0, 0)

Call asmb1.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asmb1.Extension.SelectByID2("Plane1@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)

```

```

Call asubl.AddMate(5, 0, False, mesafe, 0)

' C_SACI MONTAJ
  Call asubl.Extension.SelectByID2("Plane1@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Plane2@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1,
Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)
  asubl.ClearSelection2 True
  Call asubl.Extension.SelectByID2("Plane2@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Plane3@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1,
Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)
  asubl.ClearSelection2 True
  Call asubl.Extension.SelectByID2("Front@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1,
Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)
  asubl.ClearSelection2 True
  Call asubl.Extension.SelectByID2("Plane1@C_Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Plane1@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)
  asubl.ClearSelection2 True
  Call asubl.Extension.SelectByID2("Plane2@C_Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Plane2@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)
  asubl.ClearSelection2 True
  Call asubl.Extension.SelectByID2("Plane3@C_Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
  Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1,
Nothing, 0)
  Call asubl.AddMate(0, 0, False, 0, 0)
  asubl.ClearSelection2 True

' ALT PLAKA MONTAJ
Call asubl.Extension.SelectByID2("Plane2@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing,
0)
Call asubl.Extension.SelectByID2("Plane4@Alt Plaka-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane3@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing,
0)
Call asubl.Extension.SelectByID2("Plane3@Alt Plaka-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)

Call asubl.Extension.SelectByID2("Plane2@Alt Plaka-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane4@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing,
0)
Call asubl.AddMate(5, 1, False, ((ap_parca_genisligi - mesafe - parca_kalınlığı * 2) / 2), 0)
asubl.ClearSelection2 True

' AYAK SACI MONTAJ
Call asubl.Extension.SelectByID2("Front@Ayak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing,
0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Ayak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@Ayak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

' ÖN PANO MONTAJ
Call asubl.Extension.SelectByID2("Plane1@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane4@Alt Plaka-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Plane5@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Plane2@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 1, False, 0.105 + parca_kalınlığı, 0)
asubl.ClearSelection2 True

```

' AYAK_1 MONTAJ

```

Call asubl.Extension.SelectByID2("Front Plane@Ayak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
Call asubl.Extension.SelectByID2("Top Plane@Ayak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane3@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
Call asubl.Extension.SelectByID2("Right Plane@Ayak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

```

' AYAK_2 MONTAJ

```

Call asubl.Extension.SelectByID2("Front Plane@Ayak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Top Plane@Ayak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Right Plane@Ayak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane3@Ön Pano-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

```

' AYAK_3 MONTAJ

```

Call asubl.Extension.SelectByID2("Front Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Right Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Top Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right Plane@Ayak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

' AYAK_4 MONTAJ

```

Call asubl.Extension.SelectByID2("Front Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Right Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top Plane@Ayak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Top Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)

```



```
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
' AYAK_FEDERI_1 MONTAJ
```

```
Call asubl.Extension.SelectByID2("Top Plane@Ayak Federi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front Plane@Ayak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right Plane@Ayak Federi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top Plane@Ayak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front Plane@Ayak Federi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1 @Ayak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 1, False, 0.026, 0)
asubl.ClearSelection2 True
'ASAGIDAKI DISTANCE 174 MM DEGERI AYAK GENİSLİĞİ OLAN 200 mm den 16 mm olan ayak federı parca kalınlığının ve 10 mm bosluk payının çıkarılması ile bulunmıstır.
```

```
' AYAK_FEDERI_2 MONTAJ
Call asubl.Extension.SelectByID2("Top Plane@Ayak Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right Plane@Ayak Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front Plane@Ayak Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, False, 0.174, 0)
asubl.ClearSelection2 True
```

```
' AYAK_FEDERI_3 MONTAJ
```

```
Call asubl.Extension.SelectByID2("Top Plane@Ayak Federi-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right Plane@Ayak Federi-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front Plane@Ayak Federi-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top Plane@Ayak-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
' AYAK_FEDERI_4 MONTAJ
```

```
Call asubl.Extension.SelectByID2("Top Plane@Ayak Federi-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front Plane@Ayak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right Plane@Ayak Federi-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right Plane@Ayak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Plane1 @Ayak Federi-4@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2 @Ayak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, False, 0.026, 0)
asubl.ClearSelection2 True
```

' AYAK_FEDERI_5 MONTAJ

Call asubl.Extension.SelectByID2("Top Plane@Ayak Federi-5@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Front Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right Plane@Ayak Federi-5@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Top Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1 @Ayak Federi-5@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

' AYAK_FEDERI_6 MONTAJ

Call asubl.Extension.SelectByID2("Top Plane@Ayak Federi-6@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Front Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right Plane@Ayak Federi-6@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Top Plane@Ayak-3@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front Plane@Ayak Federi-5@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane1 @Ayak Federi-6@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(5, 0, False, 0.158, 0)
 asubl.ClearSelection2 True

' PLAKA TAKVİYESİ MONTAJ

Call asubl.Extension.SelectByID2("Right@Plaka Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane3@C_Saci-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Plaka Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane4@Alt Plaka-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane3@Alt Plaka-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Front@Plaka Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(5, 1, False, pt_mesafe, 0)
 asubl.ClearSelection2 True

' P YE GELEN SACIN MONTAJI

Call asubl.Extension.SelectByID2("Plane1 @Ayak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane2@P ye Gelen Sac-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane2@Ayak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right@P ye Gelen Sac-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Ayak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Front@P ye Gelen Sac-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

' KIZAK SACI MONTAJI

Call asubl.Extension.SelectByID2("Right@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)

```
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Top@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane6@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane4@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

' ÖN YATAK MESAFE SACI MONTAJI

```
Call asubl.Extension.SelectByID2("Right@Ön Yatak Mesafe Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front@Ön Yatak Mesafe Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane7@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Plane8@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Ön Yatak Mesafe Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

' ÖN YATAK MONTAJI

```
Call asubl.Extension.SelectByID2("Top@Ön Yatak Mesafe Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Ön Yatak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Plane2@Ön Yatak Mesafe Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Ön Yatak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right@Ön Yatak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front@E18-B-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane6@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Plane1@E18-B-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane9@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right@E18-B-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```

Call asubl.Extension.SelectByID2("Top@E18-A-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@E18-A-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@E18-B-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@E18-A-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@E18-B-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@E18-B-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@E18-B-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@E18-B-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@E18-B-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@E18-B-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@E18-A-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@E18-B-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@E18-A-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1@E18-A-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@E18-B-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

''' C TAKVİYESİ MONTAJI
Call asubl.Extension.SelectByID2("Top@C_Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@C_Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@C_Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@C_Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

' İKİ C ARASI MUHAFAZA SACI MONTAJI''''''

Call asubl.Extension.SelectByID2("Plane3@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@İki C Arası Muhafaza Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1@İki C Arası Muhafaza Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)

```

Call asubl.Extension.SelectByID2("Top@C_Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1@İki C Arası Muhafaza Sacı-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Axis1@C_Takviyesi-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

' C YE GELEN SAC MONTAJI''''''''''

Call asubl.Extension.SelectByID2("Front@C ye Gelen Sac-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1@C ye Gelen Sac-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Axis1@Ayak Sacı-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@Ayak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right@C ye Gelen Sac-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

' NO 5 SACI MONTAJI

Call asubl.Extension.SelectByID2("Front@No 5 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane2@No 5 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane10@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1@No 5 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(5, 0, True, X_7, 0)
 asubl.ClearSelection2 True

' YAN KAPAK VE DİKMELERİN MONTAJI

' YAN DİKME _1

Call asubl.Extension.SelectByID2("Plane6@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Top@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane5@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(5, 1, True, ((yk_3 - yk_parca_kalınlığı) - (gys_10 - gys_14)), 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane4@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane1@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(5, 0, False, girinti, 0)
 asubl.ClearSelection2 True

' YAN DİKME _2

Call asubl.Extension.SelectByID2("Plane6@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Top@Yan Dikme_3-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

```

Call asubl.Extension.SelectByID2("Front@Yan Dikme_3-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
Call asubl.Extension.SelectByID2("Front@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@Yan Dikme_3-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
'YAN DİKME_3
Call asubl.Extension.SelectByID2("Top@Yan Dikme_2-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane6@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@Yan Dikme_2-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Yan Dikme_3-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Yan Dikme_2-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Yan Dikme_3-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
'YAN DİKME_4
Call asubl.Extension.SelectByID2("Front@Yan Dikme_4-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@Yan Dikme_2-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Yan Dikme_4-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane6@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@Yan Dikme_4-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
'YAN KAPAK
Call asubl.Extension.SelectByID2("Plane3@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Yan Kapak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@Yan Kapak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1@Yan Kapak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Yan Dikme_1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
'YAN DİKME_1B
Call asubl.Extension.SelectByID2("Top@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane8@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 1, True, ((yk_3 - yk_parca_kalınlığı) - (gys_10 - gys_14)), 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane4@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 1, False, girinti, 0)
asubl.ClearSelection2 True

```

YAN DİKME _2B

Call asubl.Extension.SelectByID2("Front@Yan Dikme_3-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane2@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1 @Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right@Yan Dikme_3-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Yan Dikme_3-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

YAN DİKME _3B

Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Top@Yan Dikme_2-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1 @Yan Dikme_2-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane2@Yan Dikme_3-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Yan Dikme_3-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right@Yan Dikme_2-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

YAN DİKME _4B

Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Top@Yan Dikme_4-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Yan Dikme_4-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane2@Yan Dikme_2-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@Yan Dikme_4-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane1 @Yan Dikme_2-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

YAN KAPAK_B

Call asubl.Extension.SelectByID2("Front@Yan Kapak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane3@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(5, 1, True, yk_parca_kalınligı, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane2@Yan Kapak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 1, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1 @Yan Kapak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Plane1 @Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

'NO 1 ALIN SACI

Call asubl.Extension.SelectByID2("Plane1 @Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Right@No 1 Alın Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1 @No 1 Alın Sacı-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.Extension.SelectByID2("Axis1 @Kızak Sacı-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
 Call asubl.AddMate(0, 0, False, 0, 0)
 asubl.ClearSelection2 True

```

Call asubl.Extension.SelectByID2("Front@Ön Yatak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 1 Alın Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, True, X_1, 0)
asubl.ClearSelection2 True

```

'KRANK ORTA YATAK

```

Call asubl.Extension.SelectByID2("Axis1@No 1 Alın Sacı-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1@Krank Orta Yatak-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Front@Krank Orta Yatak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 1 Alın Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, False, X_2, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Right@Krank Orta Yatak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(3, 0, False, 0, 0)
asubl.ClearSelection2 True

```

No 2 SACI

```

Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Top@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@No 1 Alın Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Front@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@No 1 Alın Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, True, X_3, 0)
asubl.ClearSelection2 True

```

'2 3 NOLU ARA SAC VE NO 3 SACI

```

Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@2_3 Nolu Ara Sac-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Right@No 3 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Top@2_3 Nolu Ara Sac-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 3 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Front@No 3 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, True, uzaklık, 0)
asubl.ClearSelection2 True

```

```

Call asubl.Extension.SelectByID2("Top@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@No 3 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)

```



```

Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1 @2_3 Nolu Ara Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1 @No 3 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, True, Y_1, 0)
asubl.ClearSelection2 True

' ARKA YATAK DESTEK SACI
Call asubl.Extension.SelectByID2("Front@Arka Yatak Destek Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1 @No 5 Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1 @No 5 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1 @Arka Yatak Destek Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

' KRANK ARKA YATAK
Call asubl.Extension.SelectByID2("Axis1 @Krank Arka Yatak-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1 @No 3 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Krank Arka Yatak-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 3 Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, True, X_4, 0)
asubl.ClearSelection2 True
' NO 4 SACI

Call asubl.Extension.SelectByID2("Plane1 @No 4 Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 3 Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 1, True, X_5 + no_4s_parca_kalınlığı + No_3_parca_kalınlığı, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1 @No 4 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1 @No 3 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis2 @No 4 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1 @No 5 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
' ÖN YATAK DESTEK SACI
Call asubl.Extension.SelectByID2("Front@Ön Yatak Destek Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1 @No 4 Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1 @Ön Yatak Destek Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis2 @No 4 Sacı-1 @Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
' 3_4 ARA KAPAMA SACI
Call asubl.Extension.SelectByID2("Plane1 @Govde Yan Sacı Sag_M-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@3_4 Ara Kapama Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@3_4 Ara Kapama Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2 @No 3 Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane3 @No 3 Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@3_4 Ara Kapama Sacı-1 @Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
' 1_2 ARA KAPAMA SACI

```

```

Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@1_2 Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@1_2 Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

""""Bu sacın Burc takviyesinin ustune binmesi istenmiştir, ilgili mate burc takviyesinin mate işlemlerinin sonuna eklenmiştir.
'Call asubl.Extension.SelectByID2("Plane1@1_2 Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
'Call asubl.Extension.SelectByID2("Plane3@No 3 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
'Call asubl.AddMate(0, 0, False, 0, 0)
'asubl.ClearSelection2 True
' KARSİ KIZAK 2
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@Karsı Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1@Karsı Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Karsı Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
' KARSİ KIZAK 1
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Karsı Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1@Karsı Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Karsı Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

' YAN KIZAK 1
Call asubl.Extension.SelectByID2("Front@Yan Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Karsı Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane2@Yan Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Yan Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Karsı Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
' YAN KIZAK 2

Call asubl.Extension.SelectByID2("Front@Karsı Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Yan Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Front@Yan Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)

```

```
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Top@Yan Kızak-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Karsı Kızak-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
' NO 1-1
Call asubl.Extension.SelectByID2("Plane2@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@No 1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right@No 1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front@No 1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane3@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
' NO 1-2
Call asubl.Extension.SelectByID2("Top@No 1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right@No 1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@No 2 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Plane1@No 1-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane3@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

' ARKA ARA KAPAMA SACI

```
Call asubl.Extension.SelectByID2("Front@Arka Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@3_4 Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Right@Arka Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Plane1@No 4 Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Arka Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

' YAN KAPAK DESTEK SACI

```
Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@Yan Kapak Destek Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True
```

```
Call asubl.Extension.SelectByID2("Front@Yan Kapak Destek Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
```

```

Call asubl.Extension.SelectByID2("Plane1@Yan Kapak Destek Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing,
0)
Call asubl.Extension.SelectByID2("Right@Yan Dikme_1-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True
' BURC TAKVIYESİ
' 1
Call asubl.Extension.SelectByID2("Right@Burc Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@No 1 Alın Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1@Krank Orta Yatak-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis2@Burc Takviyesi-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, -1, True, oy_r2, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Burc Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane8@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing,
0)
Call asubl.AddMate(3, 1, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Plane1@Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing,
0)
Call asubl.Extension.SelectByID2("Front@Burc Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, 0, True, bt_uzaklık, 0)
asubl.ClearSelection2 True

' 2

Call asubl.Extension.SelectByID2("Right@Burc Takviyesi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1@No 1 Alın Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1@Burc Takviyesi-2@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1@Krank Orta Yatak-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, -1, True, oy_r2, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Burc Takviyesi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@Burc Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

""""""burc takviyesi ile 1_2 nolu sac in mate i asagıdaki
Call asubl.Extension.SelectByID2("Top@Burc Takviyesi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@1_2 Ara Kapama Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 1, False, 0, 0)
asubl.ClearSelection2 True

' BURC FEDERİ
' 2

Call asubl.Extension.SelectByID2("Plane2@Burc Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis3@Burc Federi-2@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1@Krank Orta Yatak-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

GoTo 22
Call asubl.Extension.SelectByID2("Axis1@Krank Orta Yatak-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1@Burc Federi-2@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, -1, True, 0.01, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@Burc Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1@Krank Orta Yatak-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)

```

```

Call asubl.AddMate(5, -1, True, oy_r2, 0)
asubl.ClearSelection2 True
22
Call asubl.Extension.SelectByID2("Plane1 @Burc Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Plane1 @Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(6, 1, True, 0, 0.5235987755983) "'0.523598..." 30 derece acının radyanı
asubl.ClearSelection2 True

' 1
Call asubl.Extension.SelectByID2("Plane2@Burc Federi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Kızak Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1 @Krank Orta Yatak-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis3 @Burc Federi-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

GoTo 25
Call asubl.Extension.SelectByID2("Top@Burc Federi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1 @Kızak Sacı-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, -1, True, oy_r2, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Axis1 @Burc Federi-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Axis1 @Kızak Sacı-1@Dirinler_Pres", "AXIS", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(5, -1, True, 0.01, 0)
asubl.ClearSelection2 True
25
Call asubl.Extension.SelectByID2("Plane1 @Burc Federi-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@Burc Federi-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(6, 1, False, 0, 1.047197551197) "'0.523598..." 60 derece acının radyanı
asubl.ClearSelection2 True

' kaynak dikisi
' C KAYNAK DİKİSİ-1

Call asubl.Extension.SelectByID2("Plane1 @Govde Yan Sacı Sag_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@C Kaynak Dikisi_sag-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@C Kaynak Dikisi_sag-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@C_Sacı-2@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@C Kaynak Dikisi_sag-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

' C KAYNAK DİKİSİ-2

Call asubl.Extension.SelectByID2("Front@Govde Yan Sacı Sol_M-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Front@C Kaynak Dikisi_sol-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Top@C Kaynak Dikisi_sol-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Top@C_Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

Call asubl.Extension.SelectByID2("Right@C_Sacı-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.Extension.SelectByID2("Right@C Kaynak Dikisi_sol-1@Dirinler_Pres", "PLANE", 0, 0, 0, True, 1, Nothing, 0)
Call asubl.AddMate(0, 0, False, 0, 0)
asubl.ClearSelection2 True

```

```

' *****"BU KISIMDA ALIN KAYNAGI OLAN KISIMLARI SUPPRESS
EDİYORUZ*****
Call asubl.Extension.SelectByID2("Cut-Extrude2@C_Takviyesi-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0,
Nothing, 0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude2@No 1-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude2@Karsı Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude3@Karsı Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude2@Yan Kızak-2@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude3@Yan Kızak-2@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude2@E18-A-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude2@E18-B-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditSuppress

Call asubl.Extension.SelectByID2("Extrude2@Ön Yatak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditSuppress

Call Kaynak_Dikisi_Atama ' bu komut ile kaynak dikişleri atanır.

' *****"BU KISIMDA ALIN KAYNAGI OLAN KISIMLARI UNsUPPRESS
EDİYORUZ*****
Call asubl.Extension.SelectByID2("Cut-Extrude2@C_Takviyesi-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0,
Nothing, 0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude2@No 1-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude2@Karsı Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude3@Karsı Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude2@Yan Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude3@Yan Kızak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing,
0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude2@E18-A-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude2@E18-B-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditUnsuppress

Call asubl.Extension.SelectByID2("Extrude2@Ön Yatak-1@Dirinler_Pres", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
asubl.EditUnsuppress

Call kaydet
asubl.ClearSelection2 True

```

```

asmb1.ViewZoomtofit2
asmb1.EditRebuild3

Call part_olarak_kaydet

Call Assembly_Gövdeyi_Kapat

Call Part_Gövdeyi_Cagır

Call Kesme

Call Birlestirme

Call Simetrik_Mirror_ile_Tekrar_Birlestirme

250
End Sub

Private Sub Command2_Click()
Montaj.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20

```

```

    parca_listesi.AddItem "Burc Federi"
Case Is = 21
    parca_listesi.AddItem "Burc Takviyesi"
Case Else
    ' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i
End Sub

Private Sub Form_Load()

Text1.Text = "253" ' X1
Text4.Text = "580" ' MESAFE
Text2.Text = "1191" ' EKSEN 1
Text3.Text = "450" ' EKSEN 2
Text5.Text = "36" ' X2
Text6.Text = "123" ' X3
Text7.Text = "15" ' X4
Text8.Text = "258" ' Y1
Text9.Text = "34" ' X5
Text10.Text = "213" ' X6
Text11.Text = "114" ' X7
Text12.Text = "13" ' EKSEN_3

Combo1.AddItem "mesafe"
Combo1.AddItem "eksen_1"
Combo1.AddItem "eksen_2"
Combo1.AddItem "X_1"
Combo1.AddItem "X_2"
Combo1.AddItem "X_3"
Combo1.AddItem "X_4"
Combo1.AddItem "X_5"
Combo1.AddItem "X_6"
Combo1.AddItem "X_7"
Combo1.AddItem "Y_1"
Combo1.AddItem "eksen_3"

End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()
.....
"İKİ C ARASI MUHAFAZA SACI"
.....

Call VeriOkuma
Call Cyegelensac

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

swApp.SetUserPreferenceToggle swSketchInference, False
'part.SetUserPreferenceIntegerValue swImageQualityWireframe, swWireframeImageQualityCustom

'Muhafaza_Saci.Print ms_x, ms_1 - ms_x

```



```

If ct_x < (pi / 2) Then

part.CreateLine2 0, 0, 0, ms_parca_kalinligi, 0, 0
part.CreateLine2 ms_parca_kalinligi, 0, 0, ms_parca_kalinligi, ms_1, 0
part.CreateLine2 ms_parca_kalinligi, ms_1, 0, 0, (ms_1 - ms_x), 0

swApp.SetUserPreferenceToggle swSketchInference, True
'part.SetUserPreferenceIntegerValue swImageQualityWireframe, swWireframeImageQualityCustom

part.CreateLine2 0, (ms_1 - ms_x), 0, 0, 0, 0
Else

part.CreateLine2 0, 0, 0, ms_parca_kalinligi, 0, 0
part.CreateLine2 ms_parca_kalinligi, 0, 0, ms_parca_kalinligi, ms_1, 0
part.CreateLine2 ms_parca_kalinligi, ms_1, 0, 0, (ms_1 + ms_x), 0

swApp.SetUserPreferenceToggle swSketchInference, True
'part.SetUserPreferenceIntegerValue swImageQualityWireframe, swWireframeImageQualityCustom

part.CreateLine2 0, (ms_1 + ms_x), 0, 0, 0, 0
End If

part.ClearSelection2 True
'Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
'part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, (mesafe - 2 * cs_parca_kalinligi), 0, False, False, False, False,
0, 0, False, False, False, False, 1, 1, 1, 0, 0, False

part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", ms_parca_kalinligi / 2, ms_1 / 2, 0, True, 4,
Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, (mesafe - 2 * cs_parca_kalinligi), 0, False, False, False, False,
0, 0, False, False, False, False, 1, 1, 1, 0, 0, False
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.SketchRectangle -ms_3, 0, 0, -(mesafe - 2 * cs_parca_kalinligi - ms_3), ms_r + 0.001, 0, 0

part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Point2", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 ms_r, 2

boolstatus = part.Extension.SelectByID2("Point3", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 ms_r, 2

part.FeatureManager.FeatureCut True, False, True, 0, 0, ms_parca_kalinligi, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("", "VERTEX", ms_parca_kalinligi, ms_1, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "VERTEX", ms_parca_kalinligi, ms_1, (mesafe - 2 * cs_parca_kalinligi), True, 0,
Nothing, 0)
part.InsertAxis2 True
If ct_x < (pi / 2) Then
boolstatus = part.Extension.SelectByID2("", "FACE", ms_parca_kalinligi / 2, (((ms_1) + (ms_1 - ms_x)) / 2), (mesafe - 2 *
cs_parca_kalinligi) / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
Else
boolstatus = part.Extension.SelectByID2("", "FACE", ms_parca_kalinligi / 2, (((ms_1) + (ms_1 + ms_x)) / 2), (mesafe - 2 *
cs_parca_kalinligi) / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
End If

```

.....

=====KAYNAKLI BÖLÜM=====

```

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, -parcalar_arasi_bosluk, ms_1 + 0.1, 0, 0 ' y değerine 100 mm eklenmiştir amaç kesim işlemi
yapılırken hiç bir parça kalıntı olmaması için büyük bir y değeri olan dikdörtgen yapıp kesmektir. sadece ms_1 dediğimiz
zaman arka taraf acılı olduğu için milimetrik bir kısım kalıyor kaynak atmaya engel oluyor...
part.SketchRectangle -(mesafe - 2 * cs_parca_kalınlığı), 0, 0, -(mesafe - 2 * cs_parca_kalınlığı - parcalar_arasi_bosluk), ms_1 +
0.1, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, ms_parca_kalınlığı * 2, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.SaveAs2 fl0 + "İki C Arası Muhafaza Sacı.SLDPRT", 0, False, False
End Sub

Private Sub Command2_Click()
Muhafaza_Sacı.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik , en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14

```

```

        parca_listesi.AddItem "Yan Kapak"
    Case Is = 15
        parca_listesi.AddItem "Krank Orta Yatak"
    Case Is = 16
        parca_listesi.AddItem "Krank Arka Yatak"
    Case Is = 17
        parca_listesi.AddItem "Arka Yatak Destek Sacı"
    Case Is = 18
        parca_listesi.AddItem "Ön Yatak Destek Sacı"
    Case Is = 19
        parca_listesi.AddItem "No 4 Sacı"
    Case Is = 20
        parca_listesi.AddItem "Birc Federi"
    Case Is = 21
        parca_listesi.AddItem "Birc Takviyesi"
    'Case Else
        ' parca_listesi.AddItem "Aradığınız Degisken Baska Bir Parçayı Etkilememtedir."

    End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()

    Combo1.AddItem "ms_1"
    Combo1.AddItem "ms_3"
    'Combo1.AddItem "gys_1"
    'Combo1.AddItem "gys_3"
    'Combo1.AddItem "r1"
    'Combo1.AddItem "cs_8"
    'Combo1.AddItem "gys_2"
    'Combo1.AddItem "cs_10"
    'Combo1.AddItem "gys_6"
    'Combo1.AddItem "gys_5"
    'Combo1.AddItem "gys_4"
    'Combo1.AddItem "r2"
    'Combo1.AddItem "cs_2"
    'Combo1.AddItem "cs_3"
    'Combo1.AddItem "ks_parca_kalınlığı"
    'Combo1.AddItem "ct_parca_boyu"
    'Combo1.AddItem "cs_parca_kalınlığı"
    'Combo1.AddItem "mesafe"
    Combo1.AddItem "ms_r"
    Combo1.AddItem "ms_parca_kalınlığı"
    'Combo1.AddItem "C_Muhafaza_Sacı_KM"

    parca_listesi.Visible = False
End Sub

Private Sub mnu_muhafazosya_Click()
Dim bb
    bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "ms_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Combo1_Click()
    parca_listesi.Visible = False
    parca_listesi.Clear
End Sub

Private Sub Command1_Click()
    Call VeriOkuma

    Set swApp = GetObject( "sldworks.application")
    Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
    Set part = swApp.ActiveDoc

    pi = 3.141592654

```

```

Call snap
cygs_beta = Atn(as_parca_kalnlığı / (as_1 - as_2))

Tan(cygs_beta) = no_4s_parca_kalnlığı / no_4s_y
no_4s_y = no_4s_parca_kalnlığı / Tan(cygs_beta)

no4s_alfa = (pi / 2) - cygs_beta

Tan(no4s_alfa) = no4s_m/(X_6+X_7+no_5s_parca_kalnlığı)
no4s_m = Tan(no4s_alfa) * (X_6 + X_7 + no_5s_parca_kalnlığı)

cygs_k = cygs_parca_kalnlığı / (Tan(cygs_beta))

cygs_m = Sqr(cygs_k ^ 2 + cygs_parca_kalnlığı ^ 2)

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
'
'
' C ye gelen sac ın geometrisi nde ki küçük bir problemden dolayı, parçalar birleşmemektedir.
' Dolayısı ile yeni bir Cozum yolu kullanılacaktır.
' Parca, c ye gelen sac profiline uygun olarak kesilecektir, bundan dolayı asagıda ki Go to Komutu ile Daha önce kullanılan
yöntem es geçilecektir.
'
GoTo 66
part.CreateLine2 0, as_2 + cygs_m + no4s_m, 0, 0, gys_10 - üst_girinti, 0
part.ViewZoomtofit2
part.CreateLine2 0, gys_10 - üst_girinti, 0, no_4s_parca_kalnlığı, gys_10 - üst_girinti, 0
part.ViewZoomtofit2
part.CreateLine2 no_4s_parca_kalnlığı, gys_10 - üst_girinti, 0, no_4s_parca_kalnlığı, as_2 + cygs_m + no4s_m + no_4s_y, 0
part.CreateLine2 no_4s_parca_kalnlığı, as_2 + cygs_m + no4s_m + no_4s_y, 0, 0, as_2 + cygs_m + no4s_m, 0
66

part.CreateLine2 (X_7 + no_5s_parca_kalnlığı + X_6), as_2 + cygs_m, 0, (X_7 + no_5s_parca_kalnlığı + X_6), gys_10 -
üst_girinti, 0
part.ViewZoomtofit2
part.CreateLine2 (X_7 + no_5s_parca_kalnlığı + X_6), gys_10 - üst_girinti, 0, (X_7 + no_5s_parca_kalnlığı + X_6) +
no_4s_parca_kalnlığı, gys_10 - üst_girinti, 0
part.ViewZoomtofit2
part.CreateLine2 (X_7 + no_5s_parca_kalnlığı + X_6) + no_4s_parca_kalnlığı, gys_10 - üst_girinti, 0, (X_7 +
no_5s_parca_kalnlığı + X_6) + no_4s_parca_kalnlığı, as_2 + cygs_m, 0
part.CreateLine2 (X_7 + no_5s_parca_kalnlığı + X_6) + no_4s_parca_kalnlığı, as_2 + cygs_m, 0, (X_7 +
no_5s_parca_kalnlığı + X_6), as_2 + cygs_m, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.CreatePlaneAtOffset3 (X_7 + no_5s_parca_kalnlığı + X_6), False, True

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

'ana_form.Print eksen_2, ayds_r1
' asagıdaki 10 mm (0.01) en üst noktadan 10 mm asagıda olan no 4 sacının ust kısmı için YANI "üst_girinti"
part.CreateCircleByRadius2 -mesafe / 2, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalnlığı +
üst_girinti))), (X_7 + no_5s_parca_kalnlığı + X_6), ay_r2
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Arc1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, no_4s_parca_kalnlığı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
'ana_form.Print eksen_2, ayds_r1
' asagıdaki 10 mm (0.01) en üst noktadan 10 mm asagıda olan no 4 sacının ust kısmı için
part.CreateCircleByRadius2 -mesafe / 2, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalnlığı) - eksen_2 +
üst_girinti)), (X_7 + no_5s_parca_kalnlığı + X_6), öyds_r1
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("Arc1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, no_4s_parca_kalnlığı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("", "FACE", ((X_7 + no_5s_parca_kalnlığı + X_6) + (no_4s_parca_kalnlığı / 2)),
(((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalnlığı + üst_girinti))), (mesafe - (ay_r2 * 2)) / 2, True, 0,
Nothing, 0)
part.InsertAxis2 True

boolstatus = part.Extension.SelectByID2("", "FACE", ((X_7 + no_5s_parca_kalnlığı + X_6) + (no_4s_parca_kalnlığı / 2)),
((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalnlığı)) - eksen_2 + üst_girinti), (mesafe - (öyds_r1 * 2)) / 2,
True, 0, Nothing, 0)
part.InsertAxis2 True

Call C_yegelen_Rutin

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

If ct_x < (pi / 2) Then
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r,
0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), as_2 + cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), as_2 + cygs_m, 0, as_2 + cygs_m,
0
Else
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, as_2 + cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, as_2 + cygs_m, 0, as_2 +
cygs_m, 0

End If

part.ClearSelection2 True
Call part.Extension.SelectByID2("Sketch4", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0

'boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
'part.SketchAddConstraints "sgVERTICAL2D"
'part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.CreateLine2 -((mesafe / 2) + öyds_r1), (eksen_1 + gys_2 + ap_parca_kalnlığı) - eksen_2, 0, -((mesafe / 2) - 2 * öyds_r1),
(eksen_1 + gys_2 + ap_parca_kalnlığı) - eksen_2, 0
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch5", "SKETCH", 0, 0, 0, False, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "FACE", X_6 + X_7 + no_5s_parca_kalnlığı + (no_4s_parca_kalnlığı / 2),
(eksen_1 + gys_2 + ap_parca_kalnlığı) - eksen_2, ((mesafe / 2) - (öyds_r1)), True, 1, Nothing, 0)
part.InsertSplitLineProject False, False

part.SaveAs2 fl0 + "No 4 Sacı.SLDPRT", 0, False, False
End Sub

Private Sub Command2_Click()

```

```

No_4_Sacı.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
parca_listesi.AddItem "Burc Federi"
Case Is = 21
parca_listesi.AddItem "Burc Takviyesi"
Case Else
'parca_listesi.AddItem "Aradığınız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()

'Combo1.AddItem "as_parca_kalınligı"
'Combo1.AddItem "as_1"
'Combo1.AddItem "as_2"
Combo1.AddItem "no_4s_parca_kalınligı"
'Combo1.AddItem "X_7"

```

```

'Combo1.AddItem "cygs_parca_kalınlgı"
'Combo1.AddItem "X_6"
'Combo1.AddItem "no_5s_parca_kalınlgı"
'Combo1.AddItem "gys_10"
'Combo1.AddItem "üst_girinti"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "eksen_1"
'Combo1.AddItem "gys_2"
'Combo1.AddItem "ap_parca_kalınlgı"
'Combo1.AddItem "gys_6"
'Combo1.AddItem "gys_5"
'Combo1.AddItem "gys_1"
'Combo1.AddItem "gys_3"
'Combo1.AddItem "r1"
'Combo1.AddItem "gys_4"
'Combo1.AddItem "r2"
'Combo1.AddItem "cs_8"
'Combo1.AddItem "cs_10"
'Combo1.AddItem "cs_2"
'Combo1.AddItem "cs_1"
'Combo1.AddItem "cs_r1"
'Combo1.AddItem "cs_3"
'Combo1.AddItem "ks_parca_kalınlgı"
'Combo1.AddItem "ct_parca_boyu"
'Combo1.AddItem "ms_parca_kalınlgı"
'Combo1.AddItem "cygs_KM"
'Combo1.AddItem "cygs_parca_kalınlgı"
'Combo1.AddItem "cygs_ms_boyu"
'Combo1.AddItem "ay_r2"
'Combo1.AddItem "öyds_r1"
'Combo1.AddItem "eksen_2"
'Combo1.AddItem "C_Muhafaza_Sacı_KM"

parca_listesi.Visible = False
End Sub
Private Sub mnu_no_4s_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "no_4s_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

pi = 3.141592654
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
"" ASAGIDA NO_5S SACINDA DA NO 4S DEĞİŞKENLERİ İLE AYNI DEGERLER KULLANILDIGINDAN NO_4S
İLE BASLAYAN DEĞİŞKENLER FAZLADAN DEĞİŞKEN YARATMAMAK İÇİN NO_5S OLARAK
DEĞİŞTİRİLMEMİŞTİR
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Call snap
cygs_beta = Atn(as_parca_kalınlgı / (as_1 - as_2))

'Tan(cygs_beta) = no_4s_parca_kalınlgı / no_4s_y
no_4s_y = no_4s_parca_kalınlgı / Tan(cygs_beta)

no4s_alfa = (pi / 2) - cygs_beta

'Tan(no4s_alfa) = no4s_m/(X_6+X_7+no_5s_parca_kalınlgı)
no5s_m = Tan(no4s_alfa) * (X_7)

cygs_k = cygs_parca_kalınlgı / (Tan(cygs_beta))

```

```
cygs_m = Sqr(cygs_k ^ 2 + cygs_parca_kalınlgı ^ 2)
```

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

'C ye gelen sac ın geometrisi nde ki küçük bir problemden dolayı, parçalar birleşmemektedir.

'Dolayısı ile yeni bir Cozum yolu kullanılacaktır.

' Parca, c ye gelen sac profiline uygun olarak kesilecektir, bundan dolayı asagıda ki Go to Komutu ile Daha önce kullanılan yöntem es geçilecektir.

```
GoTo 67
```

```
part.CreateLine2 0, as_2 + cygs_m + no5s_m, 0, 0, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 0, gys_14, 0, no_4s_parca_kalınlgı, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 no_4s_parca_kalınlgı, gys_14, 0, no_4s_parca_kalınlgı, as_2 + cygs_m + no5s_m + no_4s_y, 0
part.CreateLine2 no_4s_parca_kalınlgı, as_2 + cygs_m + no5s_m + no_4s_y, 0, 0, as_2 + cygs_m + no5s_m, 0
67
```

```
part.CreateLine2 X_7, as_2 + cygs_m, 0, X_7, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 X_7, gys_14, 0, X_7 + no_5s_parca_kalınlgı, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 X_7 + no_5s_parca_kalınlgı, gys_14, 0, X_7 + no_5s_parca_kalınlgı, as_2 + cygs_m, 0
part.CreateLine2 X_7 + no_5s_parca_kalınlgı, as_2 + cygs_m, 0, X_7, as_2 + cygs_m, 0
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False
```

```
boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.CreatePlaneAtOffset3 X_7, False, True
```

```
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
'ana_form.Print eksen_2, ayds_r1
' asagıdaki 10 mm (0.01) en üst noktadan 10 mm asagıda olan no 4 sacının ust kısmı için
part.CreateCircleByRadius2 -mesafe / 2, (eksen_1 + gys_2 + ap_parca_kalınlgı) - eksen_2, X_7, ayds_r1
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Arc1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, no_5s_parca_kalınlgı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", (no_5s_parca_kalınlgı / 2) + X_7, gys_14, mesafe / 2, True, 0, Nothing,
0)
```

```
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", (no_5s_parca_kalınlgı / 2) + X_7, (eksen_1 + gys_2 +
ap_parca_kalınlgı) - eksen_2, mesafe / 2 - ayds_r1, True, 0, Nothing, 0)
part.InsertAxis2 True
```

```
Call C_yegelen_Rutin
```

```
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
```

```
If ct_x < (pi / 2) Then
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınlgı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r,
0
part.ViewZoomtofit2
```



```

part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), as_2 + cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)), as_2 + cygs_m, 0, 0, as_2 + cygs_m,
0
Else
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, as_2 + cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, as_2 + cygs_m, 0, 0, as_2 +
cygs_m, 0

End If

part.ClearSelection2 True
Call part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.CreateLine2 -((mesafe / 2) + (ayds_r1)), (eksen_1 + gys_2 + ap_parca_kalnlığı) - eksen_2, 0, -((mesafe / 2) - (ayds_r1)),
(eksen_1 + gys_2 + ap_parca_kalnlığı) - eksen_2, 0
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch4", "SKETCH", 0, 0, 0, False, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "FACE", X_7 + (no_5s_parca_kalnlığı / 2), (eksen_1 + gys_2 +
ap_parca_kalnlığı) - eksen_2, ((mesafe / 2) - (ayds_r1)), True, 1, Nothing, 0)
part.InsertSplitLineProject False, False

part.SaveAs2 fl0 + "No 5 Sacı.SLDPRT", 0, False, False
End Sub

Private Sub Command2_Click()
No_5_Sacı.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"

```

```

Case Is = 10
    parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
    parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
    parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
    parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
    parca_listesi.AddItem "Yan Kapak"
Case Is = 15
    parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
    parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
    parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
    parca_listesi.AddItem "Borc Federi"
Case Is = 21
    parca_listesi.AddItem "Borc Takviyesi"
'Case Else
    ' parca_listesi.AddItem "Aradığınızı Degisken Baska Bir Parçayı Etkilememktedir."

    End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()

'Combo1.AddItem "as_parca_kalınligı"
'Combo1.AddItem "as_1"
'Combo1.AddItem "as_2"
'Combo1.AddItem "no_4s_parca_kalınligı"
'Combo1.AddItem "X_7"
'Combo1.AddItem "cygs_parca_kalınligı"
'Combo1.AddItem "gys_14"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "eksen_1"
'Combo1.AddItem "gys_2"
'Combo1.AddItem "ap_parca_kalınligı"
'Combo1.AddItem "eksen_2"
'Combo1.AddItem "ayds_r1"
Combo1.AddItem "no_5s_parca_kalınligı"
'Combo1.AddItem "gys_6"
'Combo1.AddItem "gys_5"
'Combo1.AddItem "gys_1"
'Combo1.AddItem "gys_3"
'Combo1.AddItem "r1"
'Combo1.AddItem "gys_4"
'Combo1.AddItem "r2"
'Combo1.AddItem "cs_10"
'Combo1.AddItem "cs_2"
'Combo1.AddItem "cs_1"
'Combo1.AddItem "cs_r1"
'Combo1.AddItem "cs_3"
'Combo1.AddItem "ks_parca_kalınligı"
'Combo1.AddItem "ct_parca_boyu"
'Combo1.AddItem "ms_parca_kalınligı"
'Combo1.AddItem "cygs_KM"
'Combo1.AddItem "cygs_parca_kalınligı"
'Combo1.AddItem "cygs_ms_boyu"
'Combo1.AddItem "C_Muhafaza_Sacı_KM"

parca_listesi.Visible = False
End Sub

```

```

Private Sub mnu_no5sac1_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "no_5s_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

pi = 3.141592654
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
"" ASAGIDA NO_5S SACINDA DA NO 4S DEGİSKENLERİ İLE AYNI DEGERLER KULLANILDIGINDAN NO_4S
İLE BASLAYAN DEGİSKENLER FAZLADAN DEGİSKEN YARATMAMAK İÇİN NO_5S OLARAK
DEĞİSTİRİLMEMİŞTİR
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Call snap
cygs_beta = Atn(as_parca_kalınlığı / (as_1 - as_2))

Tan(cygs_beta) = no_4s_parca_kalınlığı / no_4s_y
no_4s_y = no_4s_parca_kalınlığı / Tan(cygs_beta)

no4s_alfa = (pi / 2) - cygs_beta

Tan(no4s_alfa) = no4s_m/(X_6+X_7+no_5s_parca_kalınlığı)
no5s_m = Tan(no4s_alfa) * (X_7)

cygs_k = cygs_parca_kalınlığı / (Tan(cygs_beta))

cygs_m = Sqr(cygs_k ^ 2 + cygs_parca_kalınlığı ^ 2)

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
'C ye gelen sac ın geometrisi nde ki küçük bir problemden dolayı, parçalar birleşmemektedir.
'Dolayısı ile yeni bir Cozum yolu kullanılacaktır.
' Parca, c ye gelen sac profiline uygun olarak kesilecektir, bundan dolayı asagıda ki Go to Komutu ile Daha önce kullanılan
yöntem es geçilecektir.
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
GoTo 67

part.CreateLine2 0, as_2 + cygs_m + no5s_m, 0, 0, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 0, gys_14, 0, no_4s_parca_kalınlığı, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 no_4s_parca_kalınlığı, gys_14, 0, no_4s_parca_kalınlığı, as_2 + cygs_m + no5s_m + no_4s_y, 0
part.CreateLine2 no_4s_parca_kalınlığı, as_2 + cygs_m + no5s_m + no_4s_y, 0, 0, as_2 + cygs_m + no5s_m, 0
67

part.CreateLine2 X_7, as_2 + cygs_m, 0, X_7, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 X_7, gys_14, 0, X_7 + no_5s_parca_kalınlığı, gys_14, 0
part.ViewZoomtofit2
part.CreateLine2 X_7 + no_5s_parca_kalınlığı, gys_14, 0, X_7 + no_5s_parca_kalınlığı, as_2 + cygs_m, 0
part.CreateLine2 X_7 + no_5s_parca_kalınlığı, as_2 + cygs_m, 0, X_7, as_2 + cygs_m, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False

```

```

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.CreatePlaneAtOffset3 X_7, False, True

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
'ana_form.Print eksen_2, ayds_r1
'asagidaki 10 mm (0.01) en üst noktadan 10 mm asagıda olan no 4 sacının ust kısmı için
part.CreateCircleByRadius2 -mesafe / 2, (eksen_1 + gys_2 + ap_parca_kalınlgı) - eksen_2, X_7, ayds_r1
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Arc1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, no_5s_parca_kalınlgı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("", "FACE", (no_5s_parca_kalınlgı / 2) + X_7, gys_14, mesafe / 2, True, 0, Nothing,
0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", (no_5s_parca_kalınlgı / 2) + X_7, (eksen_1 + gys_2 +
ap_parca_kalınlgı) - eksen_2, mesafe / 2 - ayds_r1, True, 0, Nothing, 0)
part.InsertAxis2 True

Call C_yegelen_Rutin

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

If ct_x < (pi / 2) Then
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınlgı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r,
0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınlgı ^ 2) - cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalınlgı ^ 2) - cygs_r * Tan(ct_alfa)), as_2 + cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınlgı ^ 2) - cygs_r * Tan(ct_alfa)), as_2 + cygs_m, 0, 0, as_2 + cygs_m,
0
Else
part.CreateLine2 0, (as_2 + cygs_m), 0, (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınlgı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 -
cygs_r, 0
part.ViewZoomtofit2
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınlgı ^ 2) + cygs_r * Tan(ct_alfa)), gys_5 - cygs_r, 0, (cXx - Sqr(ms_x ^
2 + ms_parca_kalınlgı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, as_2 + cygs_m, 0
part.CreateLine2 (cXx - Sqr(ms_x ^ 2 + ms_parca_kalınlgı ^ 2) + cygs_r * Tan(ct_alfa)) + cygs_x, as_2 + cygs_m, 0, 0, as_2 +
cygs_m, 0
End If

part.ClearSelection2 True
Call part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False, False, 0,
1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.CreateLine2 -((mesafe / 2) + (ayds_r1)), (eksen_1 + gys_2 + ap_parca_kalınlgı) - eksen_2, 0, -((mesafe / 2) - (ayds_r1)),
(eksen_1 + gys_2 + ap_parca_kalınlgı) - eksen_2, 0
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch4", "SKETCH", 0, 0, 0, False, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "FACE", X_7 + (no_5s_parca_kalınlgı / 2), (eksen_1 + gys_2 +
ap_parca_kalınlgı) - eksen_2, ((mesafe / 2) - (ayds_r1)), True, 1, Nothing, 0)
part.InsertSplitLineProject False, False

part.SaveAs2 fl0 + "No 5 Saci.SLDPRT", 0, False, False

```

```

End Sub

Private Sub Command2_Click()
No_5_Sacı.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
parca_listesi.AddItem "Birc Federi"
Case Is = 21
parca_listesi.AddItem "Birc Takviyesi"
Case Else
' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()

'Combo1.AddItem "as_parca_kalnlığı"
'Combo1.AddItem "as_1"

```

```

'Combo1.AddItem "as_2"
'Combo1.AddItem "no_4s_parca_kalnlığı"
'Combo1.AddItem "X_7"
'Combo1.AddItem "cygs_parca_kalnlığı"
'Combo1.AddItem "gys_14"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "eksen_1"
'Combo1.AddItem "gys_2"
'Combo1.AddItem "ap_parca_kalnlığı"
'Combo1.AddItem "eksen_2"
'Combo1.AddItem "ayds_r1"
'Combo1.AddItem "no_5s_parca_kalnlığı"
'Combo1.AddItem "gys_6"
'Combo1.AddItem "gys_5"
'Combo1.AddItem "gys_1"
'Combo1.AddItem "gys_3"
'Combo1.AddItem "r1"
'Combo1.AddItem "gys_4"
'Combo1.AddItem "r2"
'Combo1.AddItem "cs_10"
'Combo1.AddItem "cs_2"
'Combo1.AddItem "cs_1"
'Combo1.AddItem "cs_r1"
'Combo1.AddItem "cs_3"
'Combo1.AddItem "ks_parca_kalnlığı"
'Combo1.AddItem "ct_parca_boyu"
'Combo1.AddItem "ms_parca_kalnlığı"
'Combo1.AddItem "cygs_KM"
'Combo1.AddItem "cygs_parca_kalnlığı"
'Combo1.AddItem "cygs_ms_boyu"
'Combo1.AddItem "C_Muhafaza_Sacı_KM"

```

```

parca_listesi.Visible = False
End Sub

```

```

Private Sub mnu_no5sacı_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "no_5s_parametrik_degerler.txt", vbNormalFocus)
End Sub

```

```
Option Explicit
```

```

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

```

```

Private Sub Command1_Click()
.....
....."ÖN_PANO".....
.....

```

```
Call VeriOkuma
```

```

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

```
Call snap
```

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

```

```

part.CreateLine2 0, 0, 0, 0, gys_2, 0
part.ViewZoomtofit2
part.CreateLine2 0, gys_2, 0, (mesafe + (0.105 * 2) + (parca_kalnlığı * 2)), gys_2, 0
part.ViewZoomtofit2
part.CreateLine2 mesafe + (0.105 * 2) + (parca_kalnlığı * 2), gys_2, 0, mesafe + (0.105 * 2) + (parca_kalnlığı * 2), 0, 0
part.CreateLine2 mesafe + (0.105 * 2) + (parca_kalnlığı * 2), 0, 0, ((mesafe + (0.105 * 2) + (parca_kalnlığı * 2)) - ön_p_1), 0, 0
part.CreateLine2 ((mesafe + (0.105 * 2) + (parca_kalnlığı * 2)) - ön_p_1), 0, 0, ((mesafe + (0.105 * 2) + (parca_kalnlığı * 2)) - ön_p_1), ön_p_1, ön_p_2, 0
part.CreateLine2 (mesafe + (0.105 * 2) + (parca_kalnlığı * 2)) - ön_p_1, ön_p_2, 0, ön_p_1, ön_p_2, 0

```

```

part.CreateLine2 ön_p_1, ön_p_2, 0, ön_p_1, 0, 0
part.CreateLine2 ön_p_1, 0, 0, 0, 0, 0

boolstatus = part.Extension.SelectByID2("Point6", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 ön_p_r, 2

boolstatus = part.Extension.SelectByID2("Point7", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 ön_p_r, 2

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, ön_p_parca_kalınlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", (mesafe + (0.105 * 2) + (parca_kalınlığı * 2)) / 2, gys_2,
ön_p_parca_kalınlığı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", (mesafe + (0.105 * 2) + (parca_kalınlığı * 2)), gys_2 / 2,
ön_p_parca_kalınlığı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", (mesafe + (0.105 * 2) + (parca_kalınlığı * 2) / 2), ön_p_2 * 3 / 2,
ön_p_parca_kalınlığı, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

part.SaveAs2 fl0 + "Ön Pano.SLDPRT", 0, False, False

End Sub

Private Sub Command2_Click()
Ön_Pano.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik , en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"

```

```

Case Is = 14
    parca_listesi.AddItem "Yan Kapak"
Case Is = 15
    parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
    parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
    parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
    parca_listesi.AddItem "Borc Federi"
Case Is = 21
    parca_listesi.AddItem "Borc Takviyesi"
Case Else
    ' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()

'Combo1.AddItem "gys_2"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "parca_kalınlığı"
Combo1.AddItem "ön_p_1"
Combo1.AddItem "ön_p_2"
Combo1.AddItem "ön_p_r"
Combo1.AddItem "ön_p_parca_kalınlığı"

parca_listesi.Visible = False
End Sub

Private Sub mnu_ön_pano_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "ön_pano_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

Dim msg As String

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, 0, 0, 0, gys_10 - (gys_2 + ap_parca_kalınlığı + eksen_1 - eksen_3), 0
part.ViewZoomtofit2
part.CreateLine2 0, gys_10 - (gys_2 + ap_parca_kalınlığı + eksen_1 - eksen_3), 0, mesafe, gys_10 - (gys_2 +
ap_parca_kalınlığı + eksen_1 - eksen_3), 0
part.ViewZoomtofit2

```



```

part.CreateLine2 mesafe, gys_10 - (gys_2 + ap_parca_kalınligı + eksen_1 - eksen_3), 0, mesafe, 0, 0
part.CreateLine2 mesafe, 0, 0, (mesafe / 2 + ön_yatak_r), 0, 0
Ön_Yatak.Print ön_yatak_r
part.CreateArc2 mesafe / 2, 0, 0, (mesafe / 2 - ön_yatak_r), 0, 0, (mesafe / 2 + ön_yatak_r), 0, 0, -1
part.CreateLine2 (mesafe / 2 - ön_yatak_r), 0, 0, 0, 0

```

' burada aşağıdaki extrusion yöntemini kullanıyoruz çünkü yarım çemberin merkez noktası sağdaki ve solundaki çizgilerle aynı hizada old. için coincident görüyo,ve diğer yöntem çalışmıyo...

```

part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", mesafe / 2, (gys_10 - (gys_2 + ap_parca_kalınligı + eksen_1 - eksen_3)) / 2, 0, True, 4, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, ön_yatak_parca_kalınligı, 0, False, False, False, False, 0, 0, False, False, False, 1, 1, 1, 0, 0, False
part.SelectionManager.EnableContourSelection = 0

```

```

boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

```

```

.....
....."KAYNAKLI BÖLÜM".....
.....

```

```

Kullanilan_Parca_Kalınligı = parca_kalınligı
x_factor = 0.5
Call Kaynak

```

```

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

```

```

part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle 0, gys_10 - (gys_2 + ap_parca_kalınligı + eksen_1 - eksen_3), 0, -(ön_yatak_parca_kalınligı), (gys_10 - (gys_2 + ap_parca_kalınligı + eksen_1 - eksen_3) - Kaynak_Dikis_Boyu), 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True

```

```

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

```

```

part.SaveAs2 fl0 + "Ön Yatak.SLDPRT", 0, False, False
End Sub

```

```

Private Sub Command2_Click()
Ön_Yatak.Hide

```

```

End Sub

```

```

Private Sub Command3_Click()
parca_listesi.Visible = True

```

```

Dim gir As String
gir = Combo1.Text

```

```

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en çok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6

```

```

    parca_listesi.AddItem "Ön Pano"
Case Is = 7
    parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
    parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
    parca_listesi.AddItem "Ön Yatak"
Case Is = 10
    parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
    parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
    parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
    parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
    parca_listesi.AddItem "Yan Kapak"
Case Is = 15
    parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
    parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
    parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
    parca_listesi.AddItem "Borc Federi"
Case Is = 21
    parca_listesi.AddItem "Borc Takviyesi"
Case Else
    ' parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()

    'Combo1.AddItem "gys_10"
    'Combo1.AddItem "mesafe"
    'Combo1.AddItem "gys_2"
    'Combo1.AddItem "ap_parca_kalınligı"
    'Combo1.AddItem "eksen_1"
    Combo1.AddItem "ön_yatak_r"
    Combo1.AddItem "ön_yatak_parca_kalınligı"

    parca_listesi.Visible = False
End Sub

Private Sub mnu_önyatakdosya_Click()
    Dim bb
    bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "ön_yatak_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Combo1_Click()
    parca_listesi.Visible = False
    parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

Call snap

Dim msg As String

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, 0, 0, 0, gys_10 - (gys_2 + ap_parca_kalnlığı + eksen_1 - eksen_3), 0
part.ViewZoomtofit2
part.CreateLine2 0, gys_10 - (gys_2 + ap_parca_kalnlığı + eksen_1 - eksen_3), 0, mesafe, gys_10 - (gys_2 +
ap_parca_kalnlığı + eksen_1 - eksen_3), 0
part.ViewZoomtofit2
part.CreateLine2 mesafe, gys_10 - (gys_2 + ap_parca_kalnlığı + eksen_1 - eksen_3), 0, mesafe, 0, 0
part.CreateLine2 mesafe, 0, 0, (mesafe - öyms_1) / 2 + öyms_1, 0, 0
part.CreateLine2 (mesafe - öyms_1) / 2 + öyms_1, 0, 0, (mesafe - öyms_1) / 2 + öyms_1, öyms_2, 0
part.CreateLine2 (mesafe - öyms_1) / 2 + öyms_1, öyms_2, 0, (mesafe - öyms_1) / 2, öyms_2, 0
part.CreateLine2 (mesafe - öyms_1) / 2, öyms_2, 0, (mesafe - öyms_1) / 2, 0, 0
part.CreateLine2 (mesafe - öyms_1) / 2, 0, 0, 0, 0, 0

boolstatus = part.Extension.SelectByID2("Point6", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 öyms_r, 2

boolstatus = part.Extension.SelectByID2("Point7", "SKETCHPOINT", 0, 0, 0, False, 1, Nothing, 0)
part.SketchFillet2 öyms_r, 2

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, öyms_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, ((gys_10 - (gys_2 + ap_parca_kalnlığı + eksen_1 -
ksen_3))), öyms_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, ((gys_10 - (gys_2 + ap_parca_kalnlığı + eksen_1 -
ksen_3))) / 11 * 10, öyms_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

.....
....."KAYNAKLI BÖLÜM".....
.....

Kullanilan_Parca_Kalnlığı = parca_kalnlığı
x_factor = 0.5
Call Kaynak

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle -öyms_parca_kalnlığı, 0, 0, -(öyms_parca_kalnlığı - parcalar_arasi_bosluk), gys_10 - (gys_2 +
ap_parca_kalnlığı + eksen_1 - eksen_3), 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

part.FeatureManager.FeatureCut True, False, True, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False, False,
1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
.....
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle 0, 0, 0, -Kaynak_Dikis_Boyuu, gys_10 - (gys_2 + ap_parca_kalnlığı + eksen_1 - eksen_3), 0, 0

```

```
part.SketchRectangle 0, gys_10 - (gys_2 + ap_parca_kalinligi + eksen_1 - eksen_3), 0, -öyms_parca_kalinligi, (gys_10 - (gys_2 + ap_parca_kalinligi + eksen_1 - eksen_3) - Kaynak_Dikis_Boyu), 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True
part.ClearSelection2 True
```

```
part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCHREGION", 0, (gys_10 - (gys_2 + ap_parca_kalinligi + eksen_1 - eksen_3)) / 2, Kaynak_Dikis_Boyu / 2, True, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCHREGION", 0, (gys_10 - (gys_2 + ap_parca_kalinligi + eksen_1 - eksen_3) - Kaynak_Dikis_Boyu) + (Kaynak_Dikis_Boyu / 2), Kaynak_Dikis_Boyu / 2, True, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCHREGION", 0, (gys_10 - (gys_2 + ap_parca_kalinligi + eksen_1 - eksen_3) - Kaynak_Dikis_Boyu) + (Kaynak_Dikis_Boyu / 2), öyms_parca_kalinligi * 8 / 9, True, 4, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0.01, False, False, False, False, 0, 0, False, False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

```
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch
```

```
boolstatus = part.Extension.SelectByID2("Plane2", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
```

```
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.SketchRectangle 0, gys_10 - (gys_2 + ap_parca_kalinligi + eksen_1 - eksen_3), 0, mesafe, gys_10 - (gys_2 + ap_parca_kalinligi + eksen_1 - eksen_3) - Kaynak_Dikis_Boyu, 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False, False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

```
'boolstatus = part.Extension.SelectByID2("Extrude3", "BODYFEATURE", 0, 0, 0, False, 0, Nothing, 0)
part.EditSuppress
part.ClearSelection2 True
```

```
Kullanilan_Parca_Kalinligi = öyms_parca_kalinligi
x_factor = 0.5
Call Kaynak
```

```
boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.CreateLine2 (mesafe - öyms_1) / 2, -öyms_parca_kalinligi, 0, (mesafe - öyms_1) / 2, -(öyms_parca_kalinligi - Kaynak_Dikis_Boyu), 0
part.CreateLine2 (mesafe - öyms_1) / 2, -(öyms_parca_kalinligi - Kaynak_Dikis_Boyu), 0, (((mesafe - öyms_1) / 2) + Kaynak_Dikis_Boyu), -öyms_parca_kalinligi, 0
part.CreateLine2 (((mesafe - öyms_1) / 2) + Kaynak_Dikis_Boyu), -öyms_parca_kalinligi, 0, (mesafe - öyms_1) / 2, -öyms_parca_kalinligi, 0
swApp.SetUserPreferenceToggle swSketchInference, True
```

```
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 1)
part.SketchAddConstraints "sgPERPENDICULAR"
part.ClearSelection2 True
```

```
part.SketchManager.InsertSketch True
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Plane2", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
```

```
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("", "FACE", mesafe / 2, öyms_2 + 0.02, öyms_parca_kalinligi - parcalar_arasi_bosluk, False, 0, Nothing, 0)
```

```

boolstatus = part.SketchUseEdge2(False)
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line31", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line32", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line25", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line26", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line27", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

part.EditDelete
part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch5", "SKETCH", 0, 0, 0, False, 1, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch6", "SKETCH", 0, 0, 0, True, 4, Nothing, 0)
Dim SweepFeature As Object
Set SweepFeature = part.FeatureManager.InsertProtrusionSwept3(False, False, 0, False, False, 0, 0, False, 0, 0, 0, 1, 1, 1, 0, 1)

part.SaveAs2 fl0 + "Ön Yatak Mesafe Sacı.SLDPRТ", 0, False, False
" DESTEK PARCALARINI OLUSTURMA """"""""
"""" E18-B""""""
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

'e18_b_x = 0.065
'e18_b_y = 0.04
'e18_a_x = 0.08
'e18_a_y = 0.025

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, e18_b_x, e18_b_y, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, (gys_2 + ap_parca_kalınlgı + eksen_1 - gys_5 - eksen_3), 0, False, False, False, False, 0, 0, False, False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", e18_b_x / 2, e18_b_y, ((gys_2 + ap_parca_kalınlgı + eksen_1 - gys_5 - eksen_3)) / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

.....
....."KAYNAKLI BÖLÜM".....
.....

Kullanılan_Parca_Kalınlgı = e18_a_y
x_factor = 0.3
Call Kaynak

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.CreateLine2 0, 0, 0, 0, e18_b_y, 0
part.CreateLine2 0, e18_b_y, 0, -(gys_2 + ap_parca_kalınlgı + eksen_1 - gys_5 - eksen_3), e18_b_y, 0
part.CreateLine2 -(gys_2 + ap_parca_kalınlgı + eksen_1 - gys_5 - eksen_3), e18_b_y, 0, -(gys_2 + ap_parca_kalınlgı + eksen_1 - gys_5 - eksen_3), (e18_b_y - Kaynak_Dikis_Boyu), 0
part.CreateLine2 -(gys_2 + ap_parca_kalınlgı + eksen_1 - gys_5 - eksen_3), (e18_b_y - Kaynak_Dikis_Boyu), 0, -Kaynak_Dikis_Boyu, (e18_b_y - Kaynak_Dikis_Boyu), 0
part.CreateLine2 -Kaynak_Dikis_Boyu, (e18_b_y - Kaynak_Dikis_Boyu), 0, -Kaynak_Dikis_Boyu, 0, 0, 0, 0
part.CreateLine2 -Kaynak_Dikis_Boyu, 0, 0, 0, 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True

part.SketchRectangle 0, 0, 0, -Kaynak_Dikis_Boyu, e18_b_y, 0, 0

'boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
'boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
'boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
'boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

```

```

part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCHREGION", -Kaynak_Dikis_Boyu / 2, e18_b_y / 2, 0, True, 4,
Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, True, 0, 0, parcalar_arasi_bosluk, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

part.SaveAs2 fl0 + "E18-B.SLDPRT", 0, False, False

```

```

"""" E18-A""""
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

```

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

```

```

part.SketchRectangle 0, 0, 0, e18_a_x, e18_a_y, 0, 0

```

```

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, (gys_2 + ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3), 0,
False, False, False, False, 0, 0, False, False, False, False, 1, 1, 1, 0, 0, False

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", e18_a_x, e18_a_y / 2, ((gys_2 + ap_parca_kalinligi + eksen_1 - gys_5 -
eksen_3)) / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

.....
....."KAYNAKLI BÖLÜM".....
.....

```

```

Kullanilan_Parca_Kalinligi = e18_a_y
x_factor = 0.2
Call Kaynak

```

```

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, parcalar_arasi_bosluk, e18_a_y, 0, 0
part.SketchRectangle e18_a_x, 0, 0, e18_a_x - parcalar_arasi_bosluk, e18_a_y, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

```

```

boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)

```

```

boolstatus = part.Extension.SelectByID2("Line6", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line5", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line8", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line7", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, (gys_2 + ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3), 0, False,
False, False, False, 0, 0, False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

```

```

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, e18_a_x - parcalar_arasi_bosluk, -parcalar_arasi_bosluk, 0, 0
part.SketchRectangle parcalar_arasi_bosluk, 0, 0, 0, e18_a_y, 0, 0
part.ClearSelection2 True

```

```
swApp.SetUserPreferenceToggle swSketchInference, True
```

```
part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCHREGION", e18_a_x / 2, -parcalar_arasi_bosluk / 2, 0, True, 4,
Nothing, 0)
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCHREGION", parcalar_arasi_bosluk / 2, e18_a_y / 2, 0, True, 4,
Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, False, 0, 0, Kaynak_Dikis_Boyu, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

```
boolstatus = part.Extension.SelectByID2("Sketch3", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch
```

```
part.SaveAs2 f10 + "E18-A.SLDPRT", 0, False, False
```

```
End Sub
```

```
Private Sub Command2_Click()
Ön_Yatak_Mesafe_Saci.Hide
```

```
End Sub
```

```
Private Sub Command3_Click()
parca_listesi.Visible = True
```

```
Dim gir As String
gir = Combo1.Text
```

```
For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazilacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
```

```

        parca_listesi.AddItem "Burc Federi"
    Case Is = 21
        parca_listesi.AddItem "Burc Takviyesi"
    'Case Else
        ' parca_listesi.AddItem "Aradığınız Degisken Baska Bir Parçayı Etkilememktedir."

    End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()

    'Combo1.AddItem "gys_10"
    'Combo1.AddItem "mesafe"
    'Combo1.AddItem "gys_2"
    'Combo1.AddItem "ap_parca_kalınlgı"
    'Combo1.AddItem "eksen_1"
    Combo1.AddItem "öyms_1"
    Combo1.AddItem "öyms_2"
    Combo1.AddItem "öyms_r"
    Combo1.AddItem "öyms_parca_kalınlgı"
    'Combo1.AddItem "gys_5"

    parca_listesi.Visible = False
End Sub

Private Sub mnu_e18_Click()
Dim bb
    bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "e18_a_b_parametrik_degerler.txt", vbNormalFocus)
End Sub

Private Sub mnu_öyms_2_Click()
Dim bb
    bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "öyms_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit
Private Sub Command1_Click()

Call VeriOkuma

Call pygs_data

Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap
.....
""" Pye gelen sacı yeni deęişkenler kullanmaya gerek kalmadan cizdirelim, text. file kullanmadan """"""""""
.....

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, 0, 0, 0, m, 0
part.CreateLine2 0, m, 0, (gys_1 - gys_3 - r1 - as_parca_kalınlgı), (pygs_y - as_3 + m), 0
part.CreateLine2 0, 0, 0, ((gys_1 - gys_3 - r1 - as_parca_kalınlgı)), ((pygs_y - as_3 + m) - m), 0
part.CreateLine2 ((gys_1 - gys_3 - r1 - as_parca_kalınlgı), ((pygs_y - as_3 + m) - m), 0, ((gys_1 - gys_3 - r1 -
as_parca_kalınlgı)), (pygs_y - as_3 + m), 0
part.ViewZoomtofit2
GoTo 23

part.CreateLine2 0, (as_3 - m), 0, 0, as_3, 0
part.CreateLine2 0, as_3, 0, (gys_1 - gys_3 - r1 - as_parca_kalınlgı), pygs_y, 0
part.CreateLine2 0, (as_3 - m), 0, ((gys_1 - gys_3 - r1 - as_parca_kalınlgı)), (pygs_y - m), 0
part.CreateLine2 ((gys_1 - gys_3 - r1 - as_parca_kalınlgı), (pygs_y - m), 0, ((gys_1 - gys_3 - r1 - as_parca_kalınlgı)), pygs_y,
0
part.ViewZoomtofit2

```


23

```

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False
.....
.....

'cs_merk_x ve xs_merk_y degerlerinin hesaplanması icin
.....
.....
xp1 = cs_2
yp1 = cs_1
xp2 = 0
yp2 = 0
R = cs_r1

Call newtonR

'r1_csac degismiyor,yukarıdaki xp1 ve yp1,R degerleri degisiyor

cs_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
cs_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
.....

'Call newtonR_Pyegelensac
'P_ye_gelen_sac.Print pygs_mesafe

' yuzeyi secerken kullanacağımız y ekseninin uzunlugu (yamugun ust kenarı+ alt kenarı) / 2 dir

'part.SketchManager.InsertSketch True
'boolstatus = part.Extension.SelectByID2("", "FACE", ((gys_1 - gys_3 - r1 - as_parca_kalınlığı) / 2, ((m + (pygs_y - as_3 +
m)) / 2), (pygs_parca_kalınlığı / 2), False, 0, Nothing, 0)

'part.SketchRectangle pygs_mesafe + pygs_k, -cs_parca_kalınlığı, 0, (Sqr((gys_1 - gys_3 - r1 - as_parca_kalınlığı) ^ 2 +
(pygs_y - as_3) ^ 2) + pygs_k), 0, 0, 0
'part.ClearSelection2 True

'part.SketchRectangle pygs_mesafe + pygs_k, -(mesafe - cs_parca_kalınlığı), 0, (Sqr((gys_1 - gys_3 - r1 - as_parca_kalınlığı) ^
2 + (pygs_y - as_3) ^ 2) + pygs_k), -mesafe, 0, 0
'part.ClearSelection2 True

'boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
'part.FeatureManager.FeatureCut True, False, False, 0, 0, pygs_parca_kalınlığı * 2, 0, False, False, False, False, 0, 0, False,
False, False, False, 0, 1, 1
'part.SelectionManager.EnableContourSelection = 0

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

PRx = (((gys_1 - gys_3 - r1 - as_parca_kalınlığı) - cs_8) + cs_merk_x)
PRy = (((gys_2 - cs_10) + cs_merk_y) - (as_3 - m))

part.CreateCircleByRadius2 PRx, PRy, 0, cs_r1

part.FeatureManager.FeatureCut True, False, True, 0, 0, cs_parca_kalınlığı + parcalar_arasi_bosluk, 0, False, False, False,
False, 0, 0, False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 mesafe, False, True
part.ClearSelection2 True

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateCircleByRadius2 PRx, PRy, 0, cs_r1

```

```
part.FeatureManager.FeatureCut True, False, False, 0, 0, cs_parca_kalinligi + parcalar_arasi_bosluk, 0, False, False, False,
False, 0, 0, False, False, False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0
```

```
boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 (m), False, True
part.ClearSelection2 True
```

```
.....
....."KAYNAKLI BÖLÜM".....
.....
```

```
Kullanilan_Parca_Kalinligi = pygs_parca_kalinligi
'x_factor = 0.5
x_factor = PGSD
Call Kaynak
'P_ye_gelen_sac.Print Kaynak_Dikis_Boyu
```

```
' KAYNAK PROFİLİNİN ÇİZİMİ
```

```
'boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
'part.SketchManager.InsertSketch True
```

```
'part.ClearSelection2 True
'swApp.SetUserPreferenceToggle swSketchInference, False
'part.CreateLine2 parcalar_arasi_bosluk + 0.005, m, 0, -(Kaynak_Dikis_Boyu - parcalar_arasi_bosluk), m, 0
'part.CreateLine2 -(Kaynak_Dikis_Boyu - parcalar_arasi_bosluk), m, 0, parcalar_arasi_bosluk + 0.005, m +
Kaynak_Dikis_Boyu + 0.005, 0
'part.CreateLine2 parcalar_arasi_bosluk + 0.005, m + Kaynak_Dikis_Boyu + 0.005, 0, parcalar_arasi_bosluk + 0.005, m, 0
```

```
'part.CreateLine2 parcalar_arasi_bosluk + 0.005, 0, 0, -(Kaynak_Dikis_Boyu - parcalar_arasi_bosluk), 0, 0
'part.CreateLine2 -(Kaynak_Dikis_Boyu - parcalar_arasi_bosluk), 0, 0, parcalar_arasi_bosluk + 0.005, -(Kaynak_Dikis_Boyu +
0.005), 0
'part.CreateLine2 parcalar_arasi_bosluk + 0.005, -(Kaynak_Dikis_Boyu + 0.005), 0, parcalar_arasi_bosluk + 0.005, 0, 0
'swApp.SetUserPreferenceToggle swSketchInference, True
```

```
'part.SketchManager.InsertSketch True
```

```
'boolstatus = part.Extension.SelectByID2("", "FACE", (gys_1 - gys_3 - r1 - as_parca_kalinligi), (pygs_y - as_3 + (m / 2)),
mesafe / 2, False, 0, Nothing, 0)
```

```
boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 (gys_1 - gys_3 - r1 - as_parca_kalinligi), False, True
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Plane3", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.SketchManager.InsertSketch True
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
' "ÖNEMLİ.. Asağıda 5 mm eklememizin amacı kaynak dikisini büyük yapıp parca icine girmesini sağlamak..
' ve böylece c sacının sırt ı ile pygs nin birlestigi yerdeki kaynak dikişlerinin bosluklarını ortadan kaldırmak
part.CreateLine2 -cs_parca_kalinligi + 0.005, (pygs_y - as_3 + m), 0, -cs_parca_kalinligi + 0.005, (pygs_y - as_3 + m +
Kaynak_Dikis_Boyu + 0.005), 0
part.CreateLine2 -cs_parca_kalinligi + 0.005, (pygs_y - as_3 + m + Kaynak_Dikis_Boyu + 0.005), 0, -(cs_parca_kalinligi +
Kaynak_Dikis_Boyu), (pygs_y - as_3 + m), 0
part.CreateLine2 -(cs_parca_kalinligi + Kaynak_Dikis_Boyu), (pygs_y - as_3 + m), 0, -cs_parca_kalinligi + 0.005, (pygs_y -
as_3 + m), 0
part.ClearSelection2 True
part.CreateLine2 -cs_parca_kalinligi + 0.005, (pygs_y - as_3), 0, -cs_parca_kalinligi + 0.005, (pygs_y - as_3 -
Kaynak_Dikis_Boyu - 0.005), 0
part.CreateLine2 -cs_parca_kalinligi + 0.005, (pygs_y - as_3 - Kaynak_Dikis_Boyu - 0.005), 0, -(cs_parca_kalinligi +
Kaynak_Dikis_Boyu), (pygs_y - as_3), 0
part.CreateLine2 -(cs_parca_kalinligi + Kaynak_Dikis_Boyu), (pygs_y - as_3), 0, -cs_parca_kalinligi + 0.005, (pygs_y - as_3),
0
part.ClearSelection2 True
```

```
part.SketchManager.InsertSketch True
part.ClearSelection2 True
```



```

Dim SweepFeature As Object
Set SweepFeature = part.FeatureManager.InsertProtrusionSwept3(False, False, 0, False, False, 0, 0, False, 0, 0, 0, 1, 1, 1, 0, 1)

' cut icin
part.SketchRectangle (gys_1 - gys_3 - r1 - as_parca_kalnlığı), (pygs_y - as_3 + m) + 0.05, 0, (gys_1 - gys_3 - r1 - as_parca_kalnlığı) + 0.05, (pygs_y - as_3 + m) - 0.05, 0

part.SaveAs2 fl0 + "P ye Gelen Sac.SLDPRT", 0, False, False
End Sub

Private Sub Command2_Click()
P_ye_gelen_sac.Hide

End Sub

Private Sub Form_Load()

Combo1.AddItem "gys_4"
Combo1.AddItem "as_3"
Combo1.AddItem "gys_1"
Combo1.AddItem "gys_3"
Combo1.AddItem "r1"
Combo1.AddItem "as_parca_kalnlığı"
Combo1.AddItem "mesafe"
Combo1.AddItem "cs_2"
Combo1.AddItem "cs_1"
Combo1.AddItem "cs_r1"
Combo1.AddItem "cs_8"
Combo1.AddItem "gys_2"
Combo1.AddItem "cs_10"
Combo1.AddItem "cs_parca_kalnlığı"

End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Command1_Click()

Call VeriOkuma

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateLine2 0, 0, 0, 0, yk_1, 0
part.ViewZoomtofit2
part.CreateLine2 0, yk_1, 0, yk_2, yk_1, 0
part.ViewZoomtofit2
part.CreateLine2 yk_2, yk_1, 0, yk_2, yk_1 - yk_3, 0
part.CreateLine2 yk_2, yk_1 - yk_3, 0, yk_4, yk_1 - yk_3, 0
part.CreateLine2 yk_4, yk_1 - yk_3, 0, yk_4, 0, 0
part.CreateLine2 yk_4, 0, 0, 0, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, yk_parca_kalnlığı, 0, False, False, False, False, 0, 0, False, False, False, False, 1, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", yk_2, (yk_1 - yk_3) + (yk_3 / 2), yk_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("", "FACE", yk_2 / 2, yk_1, yk_parca_kalnlığı / 2, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0, False, True
part.ClearSelection2 True

part.SaveAs2 fl0 + "Yan Kapak.SLDPRPT", 0, False, False

' Yan Kapak Takviyeler
' Yan dikme _1
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

' 0.085 Dikme Parcaların Genisliği

part.SketchRectangle 0, 0, 0, yk_3 - yk_parca_kalnlığı, 0.085, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, yk_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 yk_parca_kalnlığı, False, True

boolstatus = part.Extension.SelectByID2("right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 (yk_3 - yk_parca_kalnlığı), False, True

boolstatus = part.Extension.SelectByID2("top", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 0.085, False, True

part.SaveAs2 fl0 + "Yan Dikme_1.SLDPRPT", 0, False, False

' Yan dikme _2
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

' 0.085 Dikme Parcaların Genisliği
part.SketchRectangle 0, 0, 0, yk_1 - yk_3, 0.085, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, yk_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 yk_parca_kalnlığı, False, True
boolstatus = part.Extension.SelectByID2("right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 yk_1 - yk_3, False, True

part.SaveAs2 fl0 + "Yan Dikme_2.SLDPRPT", 0, False, False

' Yan dikme _3
Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

' 0.085 Dikme Parcaların Genisliği
part.SketchRectangle 0, 0, 0, yk_2 - yk_4, 0.085, 0, 0

```

```

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, yk_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("front", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 yk_parca_kalnlığı, False, True

boolstatus = part.Extension.SelectByID2("right", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 yk_2 - yk_4, False, True

part.SaveAs2 fl0 + "Yan Dikme_3.SLDPRТ", 0, False, False

' Yan dikme _4
Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

' 0.085 Dikme Parcaların Genisliği
part.SketchRectangle 0, 0, 0, gys_13 + yk_parca_kalnlığı, 0.085, 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, yk_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

part.SaveAs2 fl0 + "Yan Dikme_4.SLDPRТ", 0, False, False

' ARKA ARA KAPAMA SACI
Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, mesafe, ((X_6 + X_7 + no_5s_parca_kalnlığı) - (gys_11 + girinti)), 0, 0

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No1_parca_kalnlığı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False

.....
....."KAYNAKLI BÖLÜM".....
.....
Kullanılan_Parca_Kalnlığı = No1_parca_kalnlığı / 2 ' 2 ye bölme işlemi kaynak boyutları çok büyük geldiği için yapılmıştır.
x_factor = 0.5
Call Kaynak

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

swApp.SetUserPreferenceToggle swSketchInference, False
part.ClearSelection2 True
part.SketchRectangle 0, 0, 0, mesafe, parcalar_arasi_bosluk, 0, 0
part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, True, 0, 0, No1_parca_kalnlığı, 0, False, False, False, False, 0, 0, False, False,
False, False, 0, 1, 1
part.SelectionManager.EnableContourSelection = 0

```

```

part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion True, False, False, 0, 0, Kaynak_Dikis_Boyu, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Right", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.ClearSelection2 True
swApp.SetUserPreferenceToggle swSketchInference, False
part.CreateLine2 -No1_parca_kalınligı, 0, 0, -No1_parca_kalınligı, Kaynak_Dikis_Boyu, 0
part.CreateLine2 -No1_parca_kalınligı, Kaynak_Dikis_Boyu, 0, -(No1_parca_kalınligı + Kaynak_Dikis_Boyu), 0, 0
part.CreateLine2 -(No1_parca_kalınligı + Kaynak_Dikis_Boyu), 0, 0, -No1_parca_kalınligı, 0, 0
swApp.SetUserPreferenceToggle swSketchInference, True

boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 1)
part.SketchAddConstraints "sgPERPENDICULAR"

part.ClearSelection2 True
part.FeatureManager.FeatureExtrusion True, False, False, 0, 0, mesafe, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1
part.SelectionManager.EnableContourSelection = 0

part.SaveAs2 fl0 + "Arka Ara Kapama Sacı.SLDPRPT", 0, False, False

'YAN KAPAK DESTEK SACI
Set swApp = GetObject(, "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.SketchRectangle 0, 0, 0, mesafe + (parca_kalınligı * 2), ((yk_3 - yk_parca_kalınligı) - (gys_10 - gys_14)) + üst_girinti, 0, 0
part.SketchRectangle 0, 0, 0, parca_kalınligı, üst_girinti, 0, 0
part.SketchRectangle mesafe + (parca_kalınligı * 2), 0, 0, (mesafe + parca_kalınligı), üst_girinti, 0, 0

part.SelectionManager.EnableContourSelection = 1
boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCHREGION", mesafe / 2, (((yk_3 - yk_parca_kalınligı) - (gys_10 -
gys_14))) / 2, 0, True, 4, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, No1_parca_kalınligı, 0, False, False, False, False, 0, 0, False,
False, False, False, 1, 1, 1, 0, 0, False
part.SelectionManager.EnableContourSelection = 0

boolstatus = part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.BlankSketch

boolstatus = part.Extension.SelectByID2("Top", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 ((yk_3 - yk_parca_kalınligı) - (gys_10 - gys_14)) + üst_girinti, False, True

part.SaveAs2 fl0 + "Yan Kapak Destek Sacı.SLDPRPT", 0, False, False

End Sub
Private Sub Command2_Click()
Yan_Kapak.Hide

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

```

```

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazilacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
parca_listesi.AddItem "Alt Plaka"
Case Is = 4
parca_listesi.AddItem "C Sacı"
Case Is = 5
parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
parca_listesi.AddItem "Ön Pano"
Case Is = 7
parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
parca_listesi.AddItem "Ön Yatak"
Case Is = 10
parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
parca_listesi.AddItem "Yan Kapak"
Case Is = 15
parca_listesi.AddItem "Krank Orta Yatak"
Case Is = 16
parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
parca_listesi.AddItem "Borc Federi"
Case Is = 21
parca_listesi.AddItem "Borc Takviyesi"
'Case Else
' parca_listesi.AddItem "Aradığınız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

```

```
End Sub
```

```
Private Sub Form_Load()
```

```

Combo1.AddItem "yk_1"
Combo1.AddItem "yk_2"
Combo1.AddItem "yk_3"
Combo1.AddItem "yk_4"
Combo1.AddItem "yk_parca_kalınligı"
'Combo1.AddItem "gys_13"
'Combo1.AddItem "mesafe"
'Combo1.AddItem "X_6"
'Combo1.AddItem "X_7"
'Combo1.AddItem "no_5s_parca_kalınligı"
'Combo1.AddItem "gys_11"
'Combo1.AddItem "girinti"
'Combo1.AddItem "No1_parca_kalınligı"
'Combo1.AddItem "parca_kalınligı"

```



```

'Combo1.AddItem "gys_10"
'Combo1.AddItem "gys_14"
'Combo1.AddItem "üst_girinti"

parca_listesi.Visible = False
End Sub

Private Sub mnu_ynkapak_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "yk_parametrik_degerler.txt", vbNormalFocus)
End Sub

Option Explicit

Private Sub Command1_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "orta_yatak_parametrik_degerler.txt", vbNormalFocus)
End Sub

Private Sub Command10_Click()
Yataklar_Parametre_Kontrol.Show
End Sub

Private Sub Command11_Click()
Yataklar_Parametre_Kontrol.Show
End Sub

Private Sub Command12_Click()
Yataklar_Parametre_Kontrol.Show
End Sub

Private Sub Command2_Click()

Call VeriOkuma

Set swApp = GetObject( "sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

Dim msg As String

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateCircleByRadius2 0, 0, 0, oy_r1
part.CreateCircleByRadius2 0, 0, 0, oy_r2
part.ClearSelection2 True

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, oy_boy, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", oy_r1, 0, oy_boy / 2, True, 0, Nothing, 0)
part.InsertAxis2 True

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.CreateLine2 -oy_r1, 0, 0, oy_r1, 0, 0
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "FACE", oy_r1, 0, oy_boy / 2, True, 1, Nothing, 0)
part.InsertSplitLineProject False, False

part.ClearSelection2 True

```

```

boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

' ASAGIDAKI CAP DARALTMA İSLEMİ İPTAL EDİLMİŞTİR MESH YAPAMADIGI İCİN
Rem swApp.SetUserPreferenceToggle swSketchInference, False
Rem part.CreateCircleByRadius2 0, 0, 0, (oy_r2 - parcalar_arasi_bosluk)
Rem part.CreateCircleByRadius2 0, 0, 0, (oy_r2 + parcalar_arasi_bosluk * 2)
Rem part.FeatureManager.FeatureCut True, False, True, 0, 0, oy_boy, 0, False, False, False, False, 0, 0, False, False, False,
False, 0, 1, 1
Rem part.SelectionManager.EnableContourSelection = 0
Rem swApp.SetUserPreferenceToggle swSketchInference, True

part.SaveAs2 fl0 + "Krank Orta Yatak.SLDPR2", 0, False, False

End Sub

Private Sub Command3_Click()
Call VeriOkuma

Set swApp = GetObject("sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

Call snap

Dim msg As String

part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)

part.CreateCircleByRadius2 0, 0, 0, öyds_r1
part.CreateCircleByRadius2 0, 0, 0, öyds_r2

Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, öyds_boy, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False

boolstatus = part.Extension.SelectByID2("", "FACE", öyds_r1, 0, öyds_boy / 2, True, 0, Nothing, 0)
part.InsertAxis2 True

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.CreateLine2 -öyds_r1, 0, 0, öyds_r1, 0, 0
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "FACE", öyds_r1, 0, öyds_boy / 2, True, 1, Nothing, 0)
part.InsertSplitLineProject False, False

part.SaveAs2 fl0 + "Ön Yatak Destek Sacı.SLDPR2", 0, False, False

End Sub

Private Sub Command4_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "öyds_parametrik_degerler.txt", vbNormalFocus)
End Sub

Private Sub Command5_Click()

Call VeriOkuma

Set swApp = GetObject("sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prt", 0, 0#, 0#)
Set part = swApp.ActiveDoc

```

Call snap

Dim msg As String

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.CreateCircleByRadius2 0, 0, 0, ay_r1
part.CreateCircleByRadius2 0, 0, 0, ay_r2
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, ay_boy, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", ay_r1, 0, ay_boy / 2, True, 0, Nothing, 0)
part.InsertAxis2 True
```

```
boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True
```

```
part.CreateLine2 -ay_r1, 0, 0, ay_r1, 0, 0
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
part.ClearSelection2 True
```

```
boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "FACE", ay_r1, 0, ay_boy / 2, True, 1, Nothing, 0)
part.InsertSplitLineProject False, False
```

' ASAGIDAKI CAP DARALTMA İSLEMİ İPTAL EDİLMİSTİR MESH YAPAMADIGI ICIN

```
Rem swApp.SetUserPreferenceToggle swSketchInference, False
Rem part.CreateCircleByRadius2 0, 0, 0, (ay_r2 - parcalar_arasi_bosluk)
Rem part.CreateCircleByRadius2 0, 0, 0, (ay_r2 + parcalar_arasi_bosluk * 2)
Rem part.FeatureManager.FeatureCut True, False, True, 0, 0, ay_boy, 0, False, False, False, False, 0, 0, False, False, False,
False, 0, 1, 1
Rem part.SelectionManager.EnableContourSelection = 0
Rem swApp.SetUserPreferenceToggle swSketchInference, True
```

```
part.SaveAs2 fl0 + "Krank Arka Yatak.SLDPRT", 0, False, False
```

End Sub

Private Sub Command6_Click()

```
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "arka_yatak_parametrik_degerler.txt", vbNormalFocus)
End Sub
```

Private Sub Command7_Click()

Call VeriOkuma

```
Set swApp = GetObject("sldworks.application")
Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.ActiveDoc
```

Call snap

Dim msg As String

```
part.SketchManager.InsertSketch True
boolstatus = part.Extension.SelectByID2("Front Plane", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
```

```
part.CreateCircleByRadius2 0, 0, 0, ayds_r1
part.CreateCircleByRadius2 0, 0, 0, ayds_r2
```

```
Call part.Extension.SelectByID2("Sketch1", "SKETCH", 0, 0, 0, False, 0, Nothing, 0)
part.FeatureManager.FeatureExtrusion2 True, False, False, 0, 0, ayds_boy, 0, False, False, False, False, 0, 0, False, False, False,
False, 1, 1, 1, 0, 0, False
```

```
boolstatus = part.Extension.SelectByID2("", "FACE", ayds_r1, 0, ayds_boy / 2, True, 0, Nothing, 0)
```

```

part.InsertAxis2 True

boolstatus = part.Extension.SelectByID2("Front", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

part.CreateLine2 -ayds_r1, 0, 0, ayds_r1, 0, 0
part.SetPickMode
part.ClearSelection2 True
part.SketchManager.InsertSketch True
part.ClearSelection2 True

boolstatus = part.Extension.SelectByID2("Sketch2", "SKETCH", 0, 0, 0, False, 4, Nothing, 0)
boolstatus = part.Extension.SelectByID2("", "FACE", ayds_r1, 0, ayds_boy / 2, True, 1, Nothing, 0)
part.InsertSplitLineProject False, False

part.SaveAs2 fl0 + "Arka Yatak Destek Saci.SLDPRJT", 0, False, False

End Sub

Private Sub Command8_Click()
Dim bb
bb = Shell("C:\WINDOWS\notepad.exe" + " " + fl0 + "ayds_parametrik_degerler.txt", vbNormalFocus)
End Sub

Private Sub Command9_Click()
Yataklar_Parametre_Kontrol.Show

End Sub

Private Sub Form_Load()

End Sub

Option Explicit

Private Sub Combo1_Click()
parca_listesi.Visible = False
parca_listesi.Clear
End Sub

Private Sub Combo2_Click()
List1.Visible = False
List1.Clear
End Sub

Private Sub Combo3_Click()
List2.Visible = False
List2.Clear
End Sub

Private Sub Combo4_Click()
List3.Visible = False
List3.Clear
End Sub

Private Sub Command1_Click()
List1.Visible = True

Dim gir As String
gir = Combo2.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazilacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
List1.AddItem "Gövde Yan Sac Sag"
Case Is = 2
List1.AddItem "Gövde Yan Sac Sol"
Case Is = 3
List1.AddItem "Alt Plaka"
Case Is = 4
List1.AddItem "C Saci"

```

```

Case Is = 5
  List1.AddItem "Ayak Sacı"
Case Is = 6
  List1.AddItem "Ön Pano"
Case Is = 7
  List1.AddItem "P ye Gelen Sac"
Case Is = 8
  List1.AddItem "Kızak Sacı"
Case Is = 9
  List1.AddItem "Ön Yatak"
Case Is = 10
  List1.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
  List1.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
  List1.AddItem "C ye Gelen Sac"
Case Is = 13
  List1.AddItem "No 5 Sacı"
Case Is = 14
  List1.AddItem "Yan Kapak"
Case Is = 15
  List1.AddItem "Krank Orta Yatak"
Case Is = 16
  List1.AddItem "Krank Arka Yatak"
Case Is = 17
  List1.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
  List1.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
  List1.AddItem "No 4 Sacı"
Case Is = 20
  List1.AddItem "Burc Federi"
Case Is = 21
  List1.AddItem "Burc Takviyesi"
'Case Else
  'parca_listesi.AddItem "Aradığınız Degisken Baska Bir Parçayı Etkilememktedir."

  End Select
End If
Next j
Next i

End Sub

Private Sub Command2_Click()
List2.Visible = True

Dim gir As String
gir = Combo3.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
  List2.AddItem "Gövde Yan Sac Sag"
Case Is = 2
  List2.AddItem "Gövde Yan Sac Sol"
Case Is = 3
  List2.AddItem "Alt Plaka"
Case Is = 4
  List2.AddItem "C Sacı"
Case Is = 5
  List2.AddItem "Ayak Sacı"
Case Is = 6
  List2.AddItem "Ön Pano"
Case Is = 7
  List2.AddItem "P ye Gelen Sac"
Case Is = 8
  List2.AddItem "Kızak Sacı"
Case Is = 9
  List2.AddItem "Ön Yatak"
Case Is = 10

```

```

    List2.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
    List2.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
    List2.AddItem "C ye Gelen Sac"
Case Is = 13
    List2.AddItem "No 5 Sacı"
Case Is = 14
    List2.AddItem "Yan Kapak"
Case Is = 15
    List2.AddItem "Krank Orta Yatak"
Case Is = 16
    List2.AddItem "Krank Arka Yatak"
Case Is = 17
    List2.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    List2.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    List2.AddItem "No 4 Sacı"
Case Is = 20
    List2.AddItem "Birc Federi"
Case Is = 21
    List2.AddItem "Birc Takviyesi"
'Case Else
'parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Command3_Click()
parca_listesi.Visible = True

Dim gir As String
gir = Combo1.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
    parca_listesi.AddItem "Gövde Yan Sac Sag"
Case Is = 2
    parca_listesi.AddItem "Gövde Yan Sac Sol"
Case Is = 3
    parca_listesi.AddItem "Alt Plaka"
Case Is = 4
    parca_listesi.AddItem "C Sacı"
Case Is = 5
    parca_listesi.AddItem "Ayak Sacı"
Case Is = 6
    parca_listesi.AddItem "Ön Pano"
Case Is = 7
    parca_listesi.AddItem "P ye Gelen Sac"
Case Is = 8
    parca_listesi.AddItem "Kızak Sacı"
Case Is = 9
    parca_listesi.AddItem "Ön Yatak"
Case Is = 10
    parca_listesi.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
    parca_listesi.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
    parca_listesi.AddItem "C ye Gelen Sac"
Case Is = 13
    parca_listesi.AddItem "No 5 Sacı"
Case Is = 14
    parca_listesi.AddItem "Yan Kapak"
Case Is = 15
    parca_listesi.AddItem "Krank Orta Yatak"

```

```

Case Is = 16
    parca_listesi.AddItem "Krank Arka Yatak"
Case Is = 17
    parca_listesi.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    parca_listesi.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    parca_listesi.AddItem "No 4 Sacı"
Case Is = 20
    parca_listesi.AddItem "Birc Federi"
Case Is = 21
    parca_listesi.AddItem "Birc Takviyesi"
'Case Else
    'parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Command4_Click()
List3.Visible = True

Dim gir As String
gir = Combo4.Text

For i = 1 To par_adet
For j = 1 To deg_adet ' simdilik ,en cok olan deger yazılacak buraya
If gir = degiskenler(i, j) Then
Select Case i
Case Is = 1
    List3.AddItem "Gövde Yan Sac Sag"
Case Is = 2
    List3.AddItem "Gövde Yan Sac Sol"
Case Is = 3
    List3.AddItem "Alt Plaka"
Case Is = 4
    List3.AddItem "C Sacı"
Case Is = 5
    List3.AddItem "Ayak Sacı"
Case Is = 6
    List3.AddItem "Ön Pano"
Case Is = 7
    List3.AddItem "P ye Gelen Sac"
Case Is = 8
    List3.AddItem "Kızak Sacı"
Case Is = 9
    List3.AddItem "Ön Yatak"
Case Is = 10
    List3.AddItem "Ön Yatak Mesafe Sacı"
Case Is = 11
    List3.AddItem "İki C Arası Muhafaza Sacı"
Case Is = 12
    List3.AddItem "C ye Gelen Sac"
Case Is = 13
    List3.AddItem "No 5 Sacı"
Case Is = 14
    List3.AddItem "Yan Kapak"
Case Is = 15
    List3.AddItem "Krank Orta Yatak"
Case Is = 16
    List3.AddItem "Krank Arka Yatak"
Case Is = 17
    List3.AddItem "Arka Yatak Destek Sacı"
Case Is = 18
    List3.AddItem "Ön Yatak Destek Sacı"
Case Is = 19
    List3.AddItem "No 4 Sacı"
Case Is = 20
    List3.AddItem "Birc Federi"
Case Is = 21

```

```

List3.AddItem "Birc Takviyesi"
'Case Else
'parca_listesi.AddItem "Aradığımız Degisken Baska Bir Parçayı Etkilememktedir."

End Select
End If
Next j
Next i

End Sub

Private Sub Form_Load()
Combo1.AddItem "oy_r1"
Combo1.AddItem "oy_r2"
Combo1.AddItem "oy_boy"

Combo3.AddItem "öyds_r1"
Combo3.AddItem "öyds_r2"
Combo3.AddItem "öyds_boy"

Combo2.AddItem "ay_r1"
Combo2.AddItem "ay_r2"
Combo2.AddItem "ay_boy"

Combo4.AddItem "ayds_r1"
Combo4.AddItem "ayds_r2"
Combo4.AddItem "ayds_boy"

parca_listesi.Visible = False
End Sub

Public parca() As String
Public n As Integer
Public fl0 As String
Public n1 As Long, n2 As Long
Public flasmb As String, flsw As String, cfix As String
Public swApp As Object, asmb1 As Object, comp As String
Public xt, yt, zt As Double, boolstatus As Boolean
Public Feature As SldWorks.Feature
'gys_sol parca kalınlığı
Public parca_kalınlığı As Double
Public part As Object
Public gys_1, gys_2, gys_3, gys_4, gys_5, gys_6, gys_7, gys_9, gys_10 As Double
Public gys_11, gys_12, gys_13, gys_14, gys_15, gys_16, gys_17 As Double
Public r1, r2, r3, r4, r5, r6 As Double
Public l1, pi, teta1, x1, y1, x2, y2, x3, y3, x4, y4, gys_merk_x, gys_merk_y As Double
Public r1_csac, r2_csac, xp1, yp1, xp2, yp2, tet1, tet2, tet3, tet4, R As Double
Public bs As Long
Public inp1, inp2, inp3, inp4, inp5, inp6, inp7, inp8, inp9, inp10, inp11, inp12, inp13, inp14, inp15, inp16, inp17, inp18, inp19,
inp20, inp21, inp22, inp23, inp24 As String
'alt plaka ile ilgili degiskenler
Public ap_1, ap_parca_kalınlığı, ap_3, ap_4, ap_5, ap_6, ap_7, ap_8, ap_parca_genisligi, ap_r1, ap_r2,
ap_buyuk_delik_derinligi, ap_vida_yaricapi, ap_vida_x_koord, ap_vida_y_koord, pt_parca_kalınlığı, pt_boyu As Double
'sol sag plakaler arası degiskenler
Public min, max, mesafe As Double
'c_sacı ile ilgili degiskenler
Public cs_1, cs_2, cs_3, cs_8, cs_10, cs_11, cs_12, cs_parca_kalınlığı, cs_r1, cs_r2, cs_r3, cs_r4, cs_r5, cs_r6 As Double
Public cs_merk_x, cs_merk_y, Öteleme_x, Öteleme_y, pt_mesafe As Double
'ayak sacı parametreleri
Public as_1, as_2, as_3, as_4, as_5, as_6, as_r, as_parca_kalınlığı, as_parca_genisligi As Double
'ön pana parametreleri
Public ön_p_1, ön_p_2, ön_p_r, ön_p_parca_kalınlığı As Double
Public parca_sayac, parca_adedi As Long
'pye gelen sac parametreleri
Public pygs_parca_kalınlığı, m, pygs_k, beta1, alfa1, alfa, beta, pygs_y, pygs_mesafe, PRx, PRy As Double
'kızak sacı parametreler
Public ks_parca_kalınlığı, eksen_1, No_1_parca_kalınlığı, No_2_parca_kalınlığı, No_2_3_parca_kalınlığı,
No_3_parca_kalınlığı As Double
'ön_yatak parametreler
Public ön_yatak_parca_kalınlığı, ön_yatak_r, eksen_3 As Double
'ön_yatak_mesafe_sacı parametreler
Public öyms_1, öyms_2, öyms_r, öyms_parca_kalınlığı As Double
Public e18_b_x, e18_b_y, e18_a_x, e18_a_y As Double

```



```

'c_takviyesi parametreler
Public ct_x, cL, cR, cB, cXx, cY, ct_tet1, ct_tet2, ct_alfa, ct_tet3, ct_parca_boyu, ct_parca_kalınligı As Double
'C'ler arası muhafaza sacı parametreler
Public ms_1, ms_3, ms_r, ms_parca_kalınligı, ms_x, C_Muhafaza_Sacı_KM As Double
'cye gelen sac parametreler
Public cygs_r, cygs_p, cygs_k, cygs_m, cygs_z, cygs_b, cygs_e, cygs_f, cygs_x, cygs_y, cygs_KM, cygs_ms_boyu, cygs_beta,
cygs_gama, cygs_parca_kalınligı As Double
'No 5 Sacı parametreler
Public no_5s_parca_kalınligı, no_5s_y, eksen_2, no5s_m As Double
'Yan Kapak parametreler
Public yk_1, yk_2, yk_3, yk_4, yk_parca_kalınligı, girinti, üst_girinti As Double
'UST BOLGE KRANK YATAKLARI ARASI PARAMETRELER
Public X_1, X_2, X_3, X_4, uzaklık, X_5, X_6, X_7, Y_1 As Double
'Krank Orta Yatak parametreler
Public oy_r1, oy_r2, oy_boy As Double
'Krank Arka Yatak parametreler
Public ay_r1, ay_r2, ay_boy As Double
'Ön Yatak Destek Sacı parametreler
Public öyds_r1, öyds_r2, öyds_boy As Double
'Arka Yatak Destek Sacı parametreler
Public ayds_r1, ayds_r2, ayds_boy As Double
'No 4 Sacı parametreler
Public no_4s_parca_kalınligı, no_4s_y, no4s_m, no4s_alfa As Double
'Kızaklar Parametreler
Public karşı_kızak_parca_kalınligı, yan_kızak_parca_kalınligı, karşı_kızak_x, yan_kızak_x, kızak_y, No1_y,
No1_parca_kalınligı As Double
'Burc Federi ve Takviyesi Parametreler
Public bt_1, bt_2, bt_3, bt_parca_kalınligı, bf_1, bf_2, bf_3, bf_4, bf_parca_kalınligı, bt_uzaklık As Double
'Kaynak parametreleri
Public c_kaynak_dikis_boyu, c_kaynak_dikis_boyutu, i, j As Double
Public degiskenler() As String
Public par_adet, deg_adet As Integer
'Kaynaklı Bölüm Parametreleri
Public parcalar_arasi_bosluk, Kaynak_Dikisi_Boyutu, Kaynak_Dikisi_Boyuy, Kullanilan_Parca_Kalınligı, x_factor As Double
'montaj ortamı parametreler
Public m_o_öteleme_x, m_o_öteleme_y, m_o_öteleme_z, bt_mesafe As Double

```

```
Sub VeriOkuma()
```

```

'Dim mesafe_msg
mesafe = Val(Montaj.Text4.Text) / 1000
eksen_1 = Val(Montaj.Text2.Text) / 1000
eksen_2 = Val(Montaj.Text3.Text) / 1000
eksen_3 = Val(Montaj.Text12.Text) / 1000
X_1 = Val(Montaj.Text1.Text) / 1000
X_2 = Val(Montaj.Text5.Text) / 1000
X_3 = Val(Montaj.Text6.Text) / 1000
X_4 = Val(Montaj.Text7.Text) / 1000
X_5 = Val(Montaj.Text9.Text) / 1000
X_6 = Val(Montaj.Text10.Text) / 1000
X_7 = Val(Montaj.Text11.Text) / 1000
Y_1 = Val(Montaj.Text8.Text) / 1000

```

```
parcalar_arasi_bosluk = 0.0001
```

```

Call gys_sol_parametre_okutma
Call gys_sag_parametre_okutma
Call c_sacı_parametre_okutma
Call alt_plaka_parametre_okutma
Call ön_pano_parametre_okutma
Call ayak_sacı_parametre_okutma
Call kızak_sacı_parametre_okutma
'mesafe_msg = MsgBox(" Mesafe Değişkenine Bağlı Olarak Tekrar Oluşturulması Gereken Parçalar; Alt Plaka, Ayak Sacı, Ön
Pano, P'ye Gelen Sac ", vbExclamation + vbOKOnly, "Tekrar Modellenmesi Gereken Parçalar")
Call ön_yatak_parametre_okutma
Call öyms_parametre_okutma
Call c_takviyesi_parametre_okutma
Call ms_parametre_okutma

```

```

Call cygs_parametre_okutma
Call no_5s_parametre_okutma
Call yk_parametre_okutma
Call orta_yatak_parametre_okutma
Call arka_yatak_parametre_okutma
Call öyds_parametre_okutma
Call ayds_parametre_okutma
Call no_4s_parametre_okutma
Call bt_parametre_okutma
Call bf_parametre_okutma
Call e18_a_b_parametre_okutma
Call kızaklar_parametre_okutma

m_o_öteleme_x = -gys_1 / 2
m_o_öteleme_y = -gys_10 / 2
m_o_öteleme_z = (parca_kalınlığı / 2 - parcalar_arasi_bosluk / 2)

pygs_y = gys_4 - 0.02 ' 0.02m alt plaka ust yuzeyi ile p'YE GElen sacın ust yuzey arası istenilen sabit mesafedir
No_1_parca_kalınlığı = 0.016
No1_parca_kalınlığı = 0.016
No1_y = 0.15
No_2_parca_kalınlığı = 0.016
No_2_3_parca_kalınlığı = 0.016
No_3_parca_kalınlığı = 0.016
uzaklık = gys_9 - öyms_parca_kalınlığı - (X_1 + No_1_parca_kalınlığı + X_3 + No_2_parca_kalınlığı + No_3_parca_kalınlığı
+ X_5 + no_4s_parca_kalınlığı + X_6 + no_5s_parca_kalınlığı + X_7)
girinti = 0.02 ' kapak lar için yan dikmelerin montajında kullanılmaktadır, gys, sırtından içeri 20mm
üst_girinti = 0.01 ' üstteki no1,no2 no 3v.s sacclarının gys üstünden içeri doğru girinti mesafesi
pt_mesafe = 0.06 ' plaka takviyesi mesafe: bu mesafe plaka takviyesi arka yuzu ile alt plaka arka yuzu arası mesafedir.

End Sub

Sub parcaları_cagir()
'Bulunması Gereken Satırlar, SolidWorks Montaj ortamı Aktif hale getirilir.
Set swApp = GetObject( "sldworks.application")
Set asmb1 = swApp.NewDocument(flsw + "assem.asmdot", 0, 0#, 0#)
Set asmb1 = swApp.ActiveDoc: nft = 1

Call snap

For i = 1 To n
  fl = parca(i) ' Burada motaj için kullanılacak parcalar listelendikten sonra numara sırasına göre çağırılır ve 50 nolu satırdaki
  GoSub 50 ' prosedür uygulanır.
Next i

asmb1.ViewZoomtofit2: bs = asmb1.EditRebuild3
Exit Sub
' Asagıda ,parcalar tek tek acılıp montaj ortamına atıldıktan sonra tek tek tekrar kapatılıyor ...
50 Set part = swApp.OpenDoc6(fl0 + fl, nft, 0, "", n1, n1)
  asmb1.AddComponent fl0 + fl, 0, 0, 0
  'Form1.Print fl0 + fl
  Set part = swApp.ActivateDoc2(fl, False, n1): Set part = Nothing ' Yandaki komut ile açık olan ve montaj ortamına tasınan
  swApp.CloseDoc fl: Return ' part. dökümaları yani parçalar kapatılır. Sadece Assembly
End Sub ' dökümanı açık kalır.

Sub parcaların_lokasyonu()
Set swApp = GetObject( "sldworks.application")
Set asmb1 = swApp.ActiveDoc

Call snap

'Form1.Cls
Dim prc As String

prc = parca(1)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "fix"
'Form1.Print flasmb
xt = 0 / 1000: yt = 0 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 180, tz)

```

```

prc = parca(2)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = -1800 / 1000: yt = -1500 / 1000: zt = 150 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(3)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1300 / 1000: yt = 500 / 1000: zt = -250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(4)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = -1800 / 1000: yt = -1500 / 1000: zt = 450 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(5)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = -1800 / 1000: yt = -1500 / 1000: zt = -900 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(6)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = -1800 / 1000: yt = -1200 / 1000: zt = -150 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(7)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1000 / 1000: yt = -1200 / 1000: zt = -150 / 1000
Call loc_comp(xt, yt, zt, tx, 270, tz)

```

```

prc = parca(8)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = -250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(9)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = -250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(10)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-3@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = -250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(11)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-4@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = -250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(12)

```

```

cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = 250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(13)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = 250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(14)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-3@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = 250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(15)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-4@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = 250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(16)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-5@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = 250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(17)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-6@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = -1200 / 1000: zt = 250 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(18)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = -1000 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(19)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -1200 / 1000: yt = -1000 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(20)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = 900 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(21)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = 1400 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(22)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"

```

```

Form1.Print flasmb
xt = 1400 / 1000: yt = 1400 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

prc = parca(23)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1400 / 1000: yt = 1400 / 1000: zt = -250 / 1000
Call loc_comp(xt, yt, zt, tx, 90, 90)

prc = parca(24)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1400 / 1000: yt = 1400 / 1000: zt = 500 / 1000
Call loc_comp(xt, yt, zt, tx, 90, 90)

prc = parca(25)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1400 / 1000: yt = 1500 / 1000: zt = -250 / 1000
Call loc_comp(xt, yt, zt, tx, 90, 90)

prc = parca(26)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1400 / 1000: yt = 1500 / 1000: zt = 500 / 1000
Call loc_comp(xt, yt, zt, tx, 90, 90)

prc = parca(27)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1400 / 1000: yt = 1200 / 1000: zt = 500 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

prc = parca(28)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 1400 / 1000: yt = 750 / 1000: zt = 500 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

prc = parca(29)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = -1100 / 1000: yt = -250 / 1000: zt = -800 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

prc = parca(30)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = 0 / 1000: yt = -500 / 1000: zt = -800 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

prc = parca(31)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
Form1.Print flasmb
xt = -1500 / 1000: yt = -500 / 1000: zt = -800 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

prc = parca(32)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
Form1.Print flasmb

```

```

xt = -1500 / 1000: yt = -500 / 1000: zt = 1000 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(33)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 0 / 1000: yt = 1500 / 1000: zt = -800 / 1000
Call loc_comp(xt, yt, zt, tx, ty, 90)

```

```

prc = parca(34)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 0 / 1000: yt = 1500 / 1000: zt = 1000 / 1000
Call loc_comp(xt, yt, zt, tx, ty, 90)

```

```

prc = parca(35)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -1500 / 1000: yt = -300 / 1000: zt = -800 / 1000
Call loc_comp(xt, yt, zt, tx, ty, 90)

```

```

prc = parca(36)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -1500 / 1000: yt = -300 / 1000: zt = 1000 / 1000
Call loc_comp(xt, yt, zt, tx, ty, 90)

```

```

prc = parca(37)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -1500 / 1000: yt = -400 / 1000: zt = -800 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(38)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -1500 / 1000: yt = -400 / 1000: zt = 1000 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(39)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -1500 / 1000: yt = -100 / 1000: zt = -800 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(40)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -1500 / 1000: yt = -100 / 1000: zt = 1000 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(41)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 450 / 1000: yt = 1200 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(42)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 400 / 1000: yt = 1200 / 1000: zt = 305 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(43)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -150 / 1000: yt = 1200 / 1000: zt = 305 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(44)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 0 / 1000: yt = 1200 / 1000: zt = 305 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(45)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -650 / 1000: yt = 1200 / 1000: zt = 305 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(46)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 150 / 1000: yt = 1200 / 1000: zt = 15 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(47)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -250 / 1000: yt = 750 / 1000: zt = 15 / 1000
Call loc_comp(xt, yt, zt, tx, 90, 90)

```

```

prc = parca(48)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -100 / 1000: yt = 1200 / 1000: zt = 15 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(49)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -700 / 1000: yt = -800 / 1000: zt = 15 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

```

```

prc = parca(50)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1500 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(51)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1500 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(52)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1500 / 1000: zt = -500 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(53)

```

```

cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1500 / 1000: zt = -500 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(54)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1300 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(55)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1300 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(56)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1600 / 1000: yt = 1300 / 1000: zt = 0 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(57)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1200 / 1000: yt = 1300 / 1000: zt = 200 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(58)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 0 / 1000: yt = 1100 / 1000: zt = 15 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(59)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = -100 / 1000: yt = 1200 / 1000: zt = 15 / 1000
Call loc_comp(xt, yt, zt, tx, 90, tz)

```

```

prc = parca(60)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1400 / 1000: zt = -150 / 1000
Call loc_comp(xt, yt, zt, 180, 90, tz)

```

```

prc = parca(61)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 1400 / 1000: zt = 500 / 1000
Call loc_comp(xt, yt, zt, 180, 90, tz)

```

```

prc = parca(62)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 1400 / 1000: yt = 500 / 1000: zt = 200 / 1000
Call loc_comp(xt, yt, zt, tx, 90, 180)
asmb1.ViewZoomtofit2: bs = asmb1.EditRebuild3

```

```

prc = parca(63)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-2@" + flasmb: cfix = "unfix"

```



```

'Form1.Print flasmb
xt = 1400 / 1000: yt = 500 / 1000: zt = 200 / 1000
Call loc_comp(xt, yt, zt, tx, 90, 180)

prc = parca(64)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 400 / 1000: yt = -800 / 1000: zt = 15 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

prc = parca(65)
cmp = Left(prc, Len(prc) - 7)
comp = cmp + "-1@" + flasmb: cfix = "unfix"
'Form1.Print flasmb
xt = 400 / 1000: yt = -800 / 1000: zt = 600 / 1000
Call loc_comp(xt, yt, zt, tx, ty, tz)

End Sub
Sub loc_comp(xtr, ytr, ztr, thx, thy, thz)
Dim vtr(15) As Double
bs = asmb.Extension.SelectByID2(comp, "COMPONENT", 0, 0, 0, False, 0, Nothing, 0)
Set swcomp = asmb.SelectionManager.GetSelectedObjectsComponent3(1, 0) 'This method gets the component of the-
' selected object in assembly mode.
If cfix = "fix" Then asmb.FixComponent: cfix = "": Exit Sub
If cfix = "unfix" Then cfix = "": asmb.UnfixComponent

'Orientasyon hazırlanıyor
pi = 3.141592654
cX = Cos(thx * pi / 180): sx = Sin(thx * pi / 180)
cY = Cos(thy * pi / 180): sy = Sin(thy * pi / 180)
cz = Cos(thz * pi / 180): sz = Sin(thz * pi / 180)

vtr(0) = cY * cz: vtr(1) = -cY * sz: vtr(2) = sy
vtr(3) = sx * sy * cz + cX * sz: vtr(4) = -sx * sy * sz + cX * cz: vtr(5) = -sx * cY
vtr(6) = -cX * sy * cz + sx * sz: vtr(7) = cX * sy * sz + sx * cz: vtr(8) = cX * cY
vtr(9) = xtr: vtr(10) = ytr: vtr(11) = ztr
vtr(12) = 1: vtr(13) = 0: vtr(14) = 0: vtr(15) = 0
Set trans1 = swApp.GetMathUtility.CreateTransform((vtr)) 'oryantasyon matrisi oluşturuldu, This method creates-
'orientation tamamlandı new MathTransform object.

swcomp.Transform2 = trans1: asmb.ClearSelection2 (All) ' This property gets or sets the component transform. It affects-
the underlying model geometry and the display of the component.

End Sub
Sub kaydet()
'Set swapp = Application.SldWorks
'Set Part = swapp.ActiveDoc
'Set SelMgr = Part.SelectionManager
'swapp.ActiveDoc.ActiveView.FrameState = 1
'Part.ShowNamedView2 "*Isometric", 7

asmb.SaveAs2 "D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres.SLDASM", 0, False, False

End Sub
Sub part_olarak_kaydet()

asmb.SaveAs2 "D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres.SLDPRT", 0, True, False

End Sub
Sub Assembly_Gövdeyi_Kapat()

Set asmb = Nothing
swApp.CloseDoc "Dirinler_Pres"

End Sub
Sub Part_Gövdeyi_Cagır()

Dim longstatus As Long, longwarnings As Long
'Set asmb = swApp.OpenDoc6("D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres.SLDPRT", 1, 0, "", longstatus, longwarnings)

'YUKARIDA longstatus VE longwarnings Long DEĞİŞKEN OLARAK TANIMLANMIŞTIR, AŞAĞIDA BUNLARIN
YERİNE

```

```

' "n1" YAZILSA DA OLUR CUNKU "n1" de yukarıda Long Değişken olarak tanımlanmıştır.....Loc_Parts Sub 'nda n1
Kullanılmıştır.

Set part = swApp.ActiveDoc
' Set part = swApp.NewDocument(flsw + "part.prtdot", 0, 0#, 0#)
Set part = swApp.OpenDoc6("D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres.SLDPRt", 1, 0, "", longstatus, longwarnings)
Set part = swApp.ActivateDoc2("Dirinler_Pres", False, longstatus)

End Sub
Sub newtonR()

pi = 3.141592654
r1_csac = Sqr(xp1 ^ 2 + yp1 ^ 2)
' 0/0 durumu belirsiz olduğu için ve bu durum prog.da hataya yol açtığı için aşağıdaki if döngüsünü kullanırız
If xp1 = 0 Then
    tet1 = 0
Else
    tet1 = Atn(yp1 / xp1)
End If
r2_csac = Sqr(xp2 ^ 2 + yp2 ^ 2)
If xp2 = 0 Then
    tet2 = 0
Else
    tet2 = Atn(yp2 / xp2)
End If

' Initial values of Teta3 (x) and Teta4 (y)
X = 300 * pi / 180: Y = 30 * pi / 180
h = 0.00000001

donaded = 0

Dim j(2, 2), jek(2, 2)

Do

K1 = X
K2 = Y

F1 = r1_csac * Cos(tet1) + R * Cos(X) - r2_csac * Cos(tet2) - R * Cos(Y)
F2 = r1_csac * Sin(tet1) + R * Sin(X) - r2_csac * Sin(tet2) - R * Sin(Y)

j(1, 1) = -R * Sin(X)
j(1, 2) = R * Sin(Y)
j(2, 1) = R * Cos(X)
j(2, 2) = -R * Cos(Y)

v1 = -F1
v2 = -F2

Rem X=INV(J)*V

For Z = 1 To 2
    For k = 1 To 2
        jek(Z, k) = j(Z, k)
    Next k
Next Z

j(1, 1) = jek(2, 2)
j(2, 2) = jek(1, 1)
j(1, 2) = -jek(1, 2)
j(2, 1) = -jek(2, 1)

detj = jek(1, 1) * jek(2, 2) - jek(1, 2) * jek(2, 1)

For Z = 1 To 2
    For k = 1 To 2
        j(Z, k) = j(Z, k) / detj
    Next k
Next Z

Rem INV(J) hesaplandı

```

```

vv1 = j(1, 1) * v1 + j(1, 2) * v2
vv2 = j(2, 1) * v1 + j(2, 2) * v2

X = X + vv1
Y = Y + vv2

If Abs(K1 - X) <= h And Abs(K2 - Y) <= h Then Exit Do
donaded = donaded + 1

DoEvents

Loop

tet3 = X
tet4 = Y

End Sub
Sub newtonR_Pyegelensac()
Rem Bu alt program artık kullanılmamaktadır.
Dim AC, ptet3, ptet4, ptet1, PRx, PRy As Double
PRx = (((gys_1 - gys_3 - r1 - as_parca_kalınlığı) - cs_8) + cs_merk_x)
PRy = (((gys_2 - cs_10) + cs_merk_y) - as_3)

pi = 3.141592654
AC = Sqr(PRx ^ 2 + PRy ^ 2)
ptet3 = Atn(PRy / PRx)
ptet1 = Atn((pygs_y - as_3) / (gys_1 - gys_3 - r1 - as_parca_kalınlığı))

'P_ye_gelen_sac.Print PRx, PRy, AC, ptet3 * 180 / pi, ptet1 * 180 / pi

' Initial values of Teta3 (x) and Teta4 (y)
X = 0.5: Y = 20 * pi / 180
h = 0.00000001

donaded = 0

Dim j(2, 2), jek(2, 2)

Do

K1 = X
K2 = Y

F1 = X * Cos(ptet1) + cs_r1 * Cos(Y) - AC * Cos(ptet3)
F2 = X * Sin(ptet1) + cs_r1 * Sin(Y) - AC * Sin(ptet3)

j(1, 1) = Cos(ptet1)
j(1, 2) = -cs_r1 * Sin(Y)
j(2, 1) = Sin(ptet1)
j(2, 2) = cs_r1 * Cos(Y)

v1 = -F1
v2 = -F2

Rem X=INV(J)*V

For Z = 1 To 2
For k = 1 To 2
jek(Z, k) = j(Z, k)
Next k
Next Z

j(1, 1) = jek(2, 2)
j(2, 2) = jek(1, 1)
j(1, 2) = -jek(1, 2)
j(2, 1) = -jek(2, 1)

detj = jek(1, 1) * jek(2, 2) - jek(1, 2) * jek(2, 1)

For Z = 1 To 2

```

```

For k = 1 To 2
  j(Z, k) = j(Z, k) / detj
Next k
Next Z

Rem INV(J) hesaplandı

vv1 = j(1, 1) * v1 + j(1, 2) * v2
vv2 = j(2, 1) * v1 + j(2, 2) * v2

X = X + vv1
Y = Y + vv2

If Abs(K1 - X) <= h And Abs(K2 - Y) <= h Then Exit Do
donaded = donaded + 1

DoEvents

Loop

pygs_mesafe = X
ptet4 = Y
End Sub
Sub Cyegelensac()
'Dim cL, cR, cB, cXx, cY, ct_tet1, ct_tet2, c_tet3 As Double
pi = 3.141592654

Öteleme_x = ((gys_1 - gys_3 - r1) - cs_8)
Öteleme_y = (gys_2 - cs_10)
C_Muhafaza_Sacı_KM = 10 / 1000
Call gys_sag_parametre_okutma

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)

cXx = ((cs_2 + cs_3 + Öteleme_x) - ks_parca_kalinligı - ct_parca_boyu / 2)
cY = gys_5
cB = Sqr((gys_merk_x - cXx) ^ 2 + (cY - gys_merk_y) ^ 2)
cR = r2
'asagıdaki C_Muhafaza_Sacı_KM degeri iki c arası muhafaza sacı nın c egrisinden dik uzaklık.. kaynak dikisi icin bırakılan
bosluktur.
cL = Sqr((cB ^ 2 - (cR + C_Muhafaza_Sacı_KM) ^ 2))

ct_tet1 = pi - Atn((cY - gys_merk_y) / (gys_merk_x - cXx))
ct_tet3 = Atn(cL / (cR + C_Muhafaza_Sacı_KM)) + ct_tet1

'Muhafaza_Sacı.Print cL, (cR + C_Muhafaza_Sacı_KM), cB

ct_x = 1.5 'Initial value          ct_x=teta2
h = 0.000001 'Acceptable error value

Do
K1 = ct_x
'F = cL * Cos(ct_x) + cB * Cos(ct_tet1) - (cR + C_Muhafaza_Sacı_KM) * Cos(ct_tet3)
'F = cL * Cos(ct_x) - cB * Cos(ct_tet1) + (cR + C_Muhafaza_Sacı_KM) * Cos(ct_tet3)
fussu = -cL * Sin(ct_x)
ct_x = ct_x - F / fussu
If Abs(K1 - ct_x) <= h Then Exit Do
donaded = donaded + 1
If donaded > 700000 Then
Exit Do

```

End If

Loop

'ana_form.Print ct_x * 180 / pi

If ct_x < (pi / 2) Then

' 16mm parca kalınlığı

ct_alfa = (pi / 2) - ct_x

Else

ct_alfa = ct_x - (pi / 2)

End If

Tan(ct_alfa) = ms_x / 0.016

ms_x = Tan(ct_alfa) * ms_parca_kalınlığı

'ana_form.Print "tet2="; ct_x, ms_x

End Sub

' Snap alt programı sketch relationsları kapatır ve otomatik hizalamaların önüne geçer.

Sub snap()

Set swApp = Application.SldWorks

swApp.SetUserPreferenceToggle swSketchSnapsNearest, False

swApp.SetUserPreferenceToggle swSketchSnapsHVLines, False

swApp.SetUserPreferenceToggle swSketchSnapsHVPoints, False

swApp.SetUserPreferenceToggle swSketchAutomaticRelations, False

swApp.SetUserPreferenceToggle swSketchSnapsPerpendicular, False

End Sub

Sub parcalistesi()

ana_form.List1.Clear

n = 65 'Parça adedi

ReDim parca(n) As String

parca(1) = "Govde Yan Sacı Sag_M.sldprt"

parca(2) = "Govde Yan Sacı Sol_M.sldprt"

parca(3) = "Alt Plaka.sldprt"

parca(4) = "C_Sacı.sldprt"

parca(5) = "C_Sacı.sldprt"

parca(6) = "Ayak Sacı.sldprt"

parca(7) = "Ön Pano.sldprt"

parca(8) = "Ayak.sldprt"

parca(9) = "Ayak.sldprt"

parca(10) = "Ayak.sldprt"

parca(11) = "Ayak.sldprt"

parca(12) = "Ayak Federi.sldprt"

parca(13) = "Ayak Federi.sldprt"

parca(14) = "Ayak Federi.sldprt"

parca(15) = "Ayak Federi.sldprt"

parca(16) = "Ayak Federi.sldprt"

parca(17) = "Ayak Federi.sldprt"

parca(18) = "Plaka Takviyesi.sldprt"

parca(19) = "P ye Gelen Sac.sldprt"

parca(20) = "Kızak Sacı.sldprt"

parca(21) = "Ön Yatak.sldprt"

parca(22) = "Ön Yatak Mesafe Sacı.sldprt"

parca(23) = "E18-B.sldprt"

parca(24) = "E18-B.sldprt"

parca(25) = "E18-A.sldprt"

parca(26) = "E18-A.sldprt"

parca(27) = "C_Takviyesi.sldprt"

parca(28) = "İki C Arası Muhafaza Sacı.sldprt"

parca(29) = "C ye Gelen Sac.sldprt"

parca(30) = "No 5 Sacı.sldprt"

parca(31) = "Yan Kapak.sldprt"

parca(32) = "Yan Kapak.sldprt"

parca(33) = "Yan Dikme_1.sldprt"

parca(34) = "Yan Dikme_1.sldprt"

parca(35) = "Yan Dikme_2.sldprt"

parca(36) = "Yan Dikme_2.sldprt"

```

parca(37) = "Yan Dikme_3.sldprt"
parca(38) = "Yan Dikme_3.sldprt"
parca(39) = "Yan Dikme_4.sldprt"
parca(40) = "Yan Dikme_4.sldprt"
parca(41) = "No 1 Alın Sacı.sldprt"
parca(42) = "Krank Orta Yatak.sldprt"
parca(43) = "Krank Arka Yatak.sldprt"
parca(44) = "Ön Yatak Destek Sacı.sldprt"
parca(45) = "Arka Yatak Destek Sacı.sldprt"
parca(46) = "No 2 Sacı.sldprt"
parca(47) = "2_3 Nolu Ara Sac.sldprt"
parca(48) = "No 3 Sacı.sldprt"
parca(49) = "No 4 Sacı.sldprt"
parca(50) = "Karsı Kızak.sldprt"
parca(51) = "Karsı Kızak.sldprt"
parca(52) = "Yan Kızak.sldprt"
parca(53) = "Yan Kızak.sldprt"
parca(54) = "No 1.sldprt"
parca(55) = "No 1.sldprt"
parca(56) = "1_2 Ara Kapama Sacı.sldprt"
parca(57) = "3_4 Ara Kapama Sacı.sldprt"
parca(58) = "Arka Ara Kapama Sacı.sldprt"
parca(59) = "Yan Kapak Destek Sacı.sldprt"
parca(60) = "Borc Takviyesi.sldprt"
parca(61) = "Borc Takviyesi.sldprt"
parca(62) = "Borc Federi.sldprt"
parca(63) = "Borc Federi.sldprt"
parca(64) = "C Kaynak Dikisi_sag.sldprt"
parca(65) = "C Kaynak Dikisi_sol.sldprt"

```

```

For i = 1 To n
  ana_form.List1.AddItem (f10 + parca(i))
Next i
End Sub

```

```

Sub gys_sol_parametre_okutma()

```

```

Open f10 + "gys_sol_parametrik_degerler.txt" For Input As 1
Input #1, msg
Line Input #1, inp1: gys_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: gys_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: gys_3 = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: gys_4 = Val(Left(inp4, 8)) / 1000
Line Input #1, inp5: gys_5 = Val(Left(inp5, 8)) / 1000
Line Input #1, inp6: gys_6 = Val(Left(inp6, 8)) / 1000
Line Input #1, inp7: gys_7 = Val(Left(inp7, 8)) / 1000
Line Input #1, inp8: gys_8 = Val(Left(inp8, 8)) / 1000
Line Input #1, inp9: gys_9 = Val(Left(inp9, 8)) / 1000
Line Input #1, inp10: gys_10 = Val(Left(inp10, 8)) / 1000
Line Input #1, inp11: gys_11 = Val(Left(inp11, 8)) / 1000
Line Input #1, inp12: gys_12 = Val(Left(inp12, 8)) / 1000
Line Input #1, inp13: gys_13 = Val(Left(inp13, 8)) / 1000
Line Input #1, inp14: gys_14 = Val(Left(inp14, 8)) / 1000
Line Input #1, inp15: gys_15 = Val(Left(inp15, 8)) / 1000
Line Input #1, inp16: gys_16 = Val(Left(inp16, 8)) / 1000
Line Input #1, inp17: gys_17 = Val(Left(inp17, 8)) / 1000
Line Input #1, inp18: r1 = Val(Left(inp18, 8)) / 1000
Line Input #1, inp19: r2 = Val(Left(inp19, 8)) / 1000
Line Input #1, inp20: r3 = Val(Left(inp20, 8)) / 1000
Line Input #1, inp21: r4 = Val(Left(inp21, 8)) / 1000
Line Input #1, inp22: r5 = Val(Left(inp22, 8)) / 1000
Line Input #1, inp23: r6 = Val(Left(inp23, 8)) / 1000
Line Input #1, inp24: parca_kalinligi = Val(Left(inp24, 8)) / 1000

```

```

Close #1
End Sub

```

```

Sub gys_sag_parametre_okutma()
Dim inp1, inp2, inp3, inp4, inp5, inp6, inp7, inp8, inp9, inp10, inp11, inp12, inp13, inp14, inp15, inp16, inp17, inp18, inp19,
inp20, inp21, inp22, inp23, inp24 As String

```

```

Open fl0 + "gys_sag_parametrik_degerler.txt" For Input As 1
Input #1, msg
Line Input #1, inp1: gys_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: gys_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: gys_3 = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: gys_4 = Val(Left(inp4, 8)) / 1000
Line Input #1, inp5: gys_5 = Val(Left(inp5, 8)) / 1000
Line Input #1, inp6: gys_6 = Val(Left(inp6, 8)) / 1000
Line Input #1, inp7: gys_7 = Val(Left(inp7, 8)) / 1000
Line Input #1, inp8: gys_8 = Val(Left(inp8, 8)) / 1000
Line Input #1, inp9: gys_9 = Val(Left(inp9, 8)) / 1000
Line Input #1, inp10: gys_10 = Val(Left(inp10, 8)) / 1000
Line Input #1, inp11: gys_11 = Val(Left(inp11, 8)) / 1000
Line Input #1, inp12: gys_12 = Val(Left(inp12, 8)) / 1000
Line Input #1, inp13: gys_13 = Val(Left(inp13, 8)) / 1000
Line Input #1, inp14: gys_14 = Val(Left(inp14, 8)) / 1000
Line Input #1, inp15: gys_15 = Val(Left(inp15, 8)) / 1000
Line Input #1, inp16: gys_16 = Val(Left(inp16, 8)) / 1000
Line Input #1, inp17: gys_17 = Val(Left(inp17, 8)) / 1000
Line Input #1, inp18: r1 = Val(Left(inp18, 8)) / 1000
Line Input #1, inp19: r2 = Val(Left(inp19, 8)) / 1000

```

```

Line Input #1, inp20: r3 = Val(Left(inp20, 8)) / 1000
Line Input #1, inp21: r4 = Val(Left(inp21, 8)) / 1000
Line Input #1, inp22: r5 = Val(Left(inp22, 8)) / 1000
Line Input #1, inp23: r6 = Val(Left(inp23, 8)) / 1000
Line Input #1, inp24: parca_kalinligi = Val(Left(inp24, 8)) / 1000

```

```
Close #1
```

```
End Sub
```

```
Sub alt_plaka_parametre_okutma()
```

```
Dim inp1, inp2, inp3, inp4, inp5, inp6, inp7, inp8, inp9, inp10, inp11, inp12, inp13, inp14, inp15, inp16, inp17 As String
```

```
Open fl0 + "ap_parametrik_degerler.txt" For Input As 1
```

```
Input #1, msg
```

```
Line Input #1, inp1: ap_1 = Val(Left(inp1, 8)) / 1000
```

```
Line Input #1, inp2: ap_parca_kalinligi = Val(Left(inp2, 8)) / 1000
```

```
Line Input #1, inp3: ap_3 = Val(Left(inp3, 8)) / 1000
```

```
Line Input #1, inp4: ap_4 = Val(Left(inp4, 8)) / 1000
```

```
Line Input #1, inp5: ap_5 = Val(Left(inp5, 8)) / 1000
```

```
Line Input #1, inp6: ap_6 = Val(Left(inp6, 8)) / 1000
```

```
Line Input #1, inp7: ap_7 = Val(Left(inp7, 8)) / 1000
```

```
Line Input #1, inp8: ap_8 = Val(Left(inp8, 8)) / 1000
```

```
Line Input #1, inp9: ap_parca_genisligi = Val(Left(inp9, 8)) / 1000
```

```
Line Input #1, inp10: ap_r1 = Val(Left(inp10, 8)) / 1000
```

```
Line Input #1, inp11: ap_r2 = Val(Left(inp11, 8)) / 1000
```

```
Line Input #1, inp12: ap_buyuk_delik_derinligi = Val(Left(inp12, 8)) / 1000
```

```
Line Input #1, inp13: ap_vida_yaricapi = Val(Left(inp13, 8)) / 1000
```

```
Line Input #1, inp14: ap_vida_x_koord = Val(Left(inp14, 8)) / 1000
```

```
Line Input #1, inp15: ap_vida_y_koord = Val(Left(inp15, 8)) / 1000
```

```
Line Input #1, inp16: pt_parca_kalinligi = Val(Left(inp16, 8)) / 1000
```

```
Line Input #1, inp17: pt_boyu = Val(Left(inp17, 8)) / 1000
```

```
Close #1
```

```
End Sub
```

```
Sub c_saci_parametre_okutma()
```

```
Dim msg As String
```

```
Dim inp1, inp2, inp3, inp4, inp5, inp6, inp7, inp8, inp9, inp10, inp11, inp12, inp13, inp14, inp15, inp16, inp17, inp18, inp19 As String
```

```
Open fl0 + "c_saci_parametrik_degerler.txt" For Input As 1
```

```
Input #1, msg
```

```
Line Input #1, inp1: cs_1 = Val(Left(inp1, 8)) / 1000
```

```
Line Input #1, inp2: cs_2 = Val(Left(inp2, 8)) / 1000
```

```
Line Input #1, inp3: cs_3 = Val(Left(inp3, 8)) / 1000
```

```
Line Input #1, inp8: cs_8 = Val(Left(inp8, 8)) / 1000
```

```
Line Input #1, inp10: cs_10 = Val(Left(inp10, 8)) / 1000
```

```

Line Input #1, inp11: cs_11 = Val(Left(inp11, 8)) / 1000
Line Input #1, inp12: cs_12 = Val(Left(inp12, 8)) / 1000
Line Input #1, inp13: cs_parca_kalnlığı = Val(Left(inp13, 8)) / 1000
Line Input #1, inp14: cs_r1 = Val(Left(inp14, 8)) / 1000
Line Input #1, inp15: cs_r2 = Val(Left(inp15, 8)) / 1000
Line Input #1, inp16: cs_r3 = Val(Left(inp16, 8)) / 1000
Line Input #1, inp18: cs_r5 = Val(Left(inp18, 8)) / 1000
Line Input #1, inp19: cs_r6 = Val(Left(inp19, 8)) / 1000

Close #1
End Sub
Sub ayak_sacı_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4, inp5, inp6, inp7, inp8 As String

Open fl0 + "ayak_sacı_parametrik_degerler.txt" For Input As 1

Input #1, msg
Line Input #1, inp1: as_parca_kalnlığı = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: as_1 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: as_2 = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: as_parca_genisligi = Val(Left(inp4, 8)) / 1000
Line Input #1, inp5: as_3 = Val(Left(inp5, 8)) / 1000
Line Input #1, inp6: as_4 = Val(Left(inp6, 8)) / 1000
Line Input #1, inp7: as_5 = Val(Left(inp7, 8)) / 1000
Line Input #1, inp8: as_r = Val(Left(inp8, 8)) / 1000

Close #1
End Sub

Sub ön_pano_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4 As String

Open fl0 + "ön_pano_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: ön_p_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: ön_p_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: ön_p_r = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: ön_p_parca_kalnlığı = Val(Left(inp4, 8)) / 1000

Close #1
End Sub

Sub kızak_sacı_parametre_okutma()
Dim msg As String
Dim inp1, inp2 As String

Open fl0 + "ks_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: ks_parca_kalnlığı = Val(Left(inp1, 8)) / 1000
Line Input #1, inp3: eksen_1 = Val(Left(inp3, 8)) / 1000

Close #1
End Sub

Sub ön_yatak_parametre_okutma()
Dim msg As String
Dim inp1, inp2 As String

Open fl0 + "ön_yatak_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: ön_yatak_parca_kalnlığı = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: ön_yatak_r = Val(Left(inp2, 8)) / 1000

```



```

Close #1
End Sub
Sub öyms_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4 As String

Open fl0 + "öyms_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: öyms_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: öyms_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: öyms_r = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: öyms_parca_kalınlığı = Val(Left(inp4, 8)) / 1000

Close #1
End Sub

Sub c_takviyesi_parametre_okutma()
Dim msg As String
Dim inp1, inp2 As String

Open fl0 + "ct_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: ct_parca_boyu = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: ct_parca_kalınlığı = Val(Left(inp2, 8)) / 1000

Close #1
End Sub

Sub ms_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4, inp5 As String

Open fl0 + "ms_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: ms_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: ms_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: ms_3 = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: ms_r = Val(Left(inp4, 8)) / 1000
Line Input #1, inp5: ms_parca_kalınlığı = Val(Left(inp5, 8)) / 1000

Close #1
End Sub

Sub cygs_parametre_okutma()
Dim msg As String
Dim inp1, inp2 As String

Open fl0 + "cygs_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: cygs_KM = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: cygs_parca_kalınlığı = Val(Left(inp2, 8)) / 1000

Close #1
End Sub

Sub no_5s_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3 As String

Open fl0 + "no_5s_parametrik_degerler.txt" For Input As 1

Input #1, msg

```

```
Line Input #1, inp1: no_5s_parca_kalnlığı = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: no_5s_1 = Val(Left(inp2, 8)) / 1000
```

```
Close #1
End Sub
```

```
Sub yk_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4, inp5 As String
```

```
Open fl0 + "yk_parametrik_degerler.txt" For Input As 1
```

```
Input #1, msg
```

```
Line Input #1, inp1: yk_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: yk_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: yk_3 = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: yk_4 = Val(Left(inp4, 8)) / 1000
Line Input #1, inp5: yk_parca_kalnlığı = Val(Left(inp5, 8)) / 1000
Close #1
```

```
End Sub
Sub orta_yatak_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3 As String
```

```
Open fl0 + "orta_yatak_parametrik_degerler.txt" For Input As 1
```

```
Input #1, msg
```

```
Line Input #1, inp1: oy_r1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: oy_r2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: oy_boy = Val(Left(inp3, 8)) / 1000
```

```
Close #1
End Sub
Sub arka_yatak_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3 As String
```

```
Open fl0 + "arka_yatak_parametrik_degerler.txt" For Input As 1
```

```
Input #1, msg
```

```
Line Input #1, inp1: ay_r1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: ay_r2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: ay_boy = Val(Left(inp3, 8)) / 1000
```

```
Close #1
End Sub
Sub öyds_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3 As String
```

```
Open fl0 + "öyds_parametrik_degerler.txt" For Input As 1
```

```
Input #1, msg
```

```
Line Input #1, inp1: öyds_r1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: öyds_r2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: öyds_boy = Val(Left(inp3, 8)) / 1000
```

```
Close #1
End Sub
Sub ayds_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3 As String
```

```
Open fl0 + "ayds_parametrik_degerler.txt" For Input As 1
```

```
Input #1, msg
```

```
Line Input #1, inp1: ayds_r1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: ayds_r2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: ayds_boy = Val(Left(inp3, 8)) / 1000
```

```

Close #1
End Sub
Sub no_4s_parametre_okutma()
Dim msg As String
Dim inp1 As String

Open fl0 + "no_4s_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: no_4s_parca_kalinligi = Val(Left(inp1, 8)) / 1000

Close #1
End Sub

Sub bt_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4, inp5 As String

Open fl0 + "bt_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: bt_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: bt_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: bt_3 = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: bt_parca_kalinligi = Val(Left(inp4, 8)) / 1000
Line Input #1, inp5: bt_uzaklik = Val(Left(inp5, 8)) / 1000

Close #1
End Sub

Sub bf_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4, inp5 As String

Open fl0 + "bf_parametrik_degerler.txt" For Input As 1

Input #1, msg

Line Input #1, inp1: bf_1 = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: bf_2 = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: bf_3 = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: bf_4 = Val(Left(inp4, 8)) / 1000
Line Input #1, inp5: bf_parca_kalinligi = Val(Left(inp5, 8)) / 1000
Close #1
End Sub

Sub e18_a_b_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4 As String

Open fl0 + "e18_a_b_parametrik_degerler.txt" For Input As 1

Input #1, msg
' mid komutu ---> bastan basla 8 satir oku demek
Line Input #1, inp1: e18_b_x = Val(Mid(inp1, 1, 8)) / 1000
Line Input #1, inp2: e18_b_y = Val(Mid(inp2, 1, 8)) / 1000
Line Input #1, inp3: e18_a_x = Val(Mid(inp3, 1, 8)) / 1000
Line Input #1, inp4: e18_a_y = Val(Mid(inp4, 1, 8)) / 1000

Close #1

'ana_form.Print e18_b_x, e18_b_y, e18_a_x, e18_a_y, gys_1, bf_1
End Sub
Sub kizaklar_parametre_okutma()
Dim msg As String
Dim inp1, inp2, inp3, inp4 As String

Open fl0 + "kizaklar_parametrik_degerler.txt" For Input As 1

Input #1, msg

```

```

Line Input #1, inp1: karsı_kızak_parca_kalınlığı = Val(Left(inp1, 8)) / 1000
Line Input #1, inp2: yan_kızak_parca_kalınlığı = Val(Left(inp2, 8)) / 1000
Line Input #1, inp3: karsı_kızak_x = Val(Left(inp3, 8)) / 1000
Line Input #1, inp4: yan_kızak_x = Val(Left(inp4, 8)) / 1000

Close #1
End Sub

Sub C_yegelen_Rutin()

Call VeriOkuma

"*****GYS ve C SACI Olusumundan Buraya Yansıyacak Degerler İçin*****"

xp1 = gys_6
yp1 = gys_5
xp2 = gys_1 - gys_3 - r1
yp2 = gys_4
R = r2

Call newtonR

gys_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
gys_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)
.....
Öteleme_x = ((gys_1 - gys_3 - r1) - cs_8)
Öteleme_y = (gys_2 - cs_10)

xp1 = cs_2
yp1 = cs_1
xp2 = 0
yp2 = 0
R = cs_r1

Call newtonR

r1_csac degismiyor,yukarıdaki xp1 ve yp1,R degerleri degisiyor

cs_merk_x = r1_csac * Cos(tet1) + R * Cos(tet3)
cs_merk_y = r1_csac * Sin(tet1) + R * Sin(tet3)

.....

Call Cyegelensac

Tan(ct_alfa)=cygs_p/cygs_r
'cygs_p^2+cygs_r^2=cygs_KM^2

'(Tan(ct_alfa)*cygs_r)^2+cygs_r^2 =cygs_KM^2
'cygs_KM = 0.003
'cygs_parca_kalınlığı = 0.016
cygs_r = Sqr(cygs_KM ^ 2 / (Tan(ct_alfa) * Tan(ct_alfa) + 1))
cygs_beta = Atn(as_parca_kalınlığı / (as_1 - as_2))
'Tan(cygs_beta)=cygs_parca_kalınlığı/cygs_k

cygs_k = cygs_parca_kalınlığı / (Tan(cygs_beta))

cygs_m = Sqr(cygs_k ^ 2 + cygs_parca_kalınlığı ^ 2)
.....
If ct_x < (pi / 2) Then
cygs_gama = cygs_beta - ct_alfa
Else
cygs_gama = cygs_beta + ct_alfa
End If

Tan(cygs_gama)=cygs_parca_kalınlığı/cygs_z
'C_ye_gelen_sac.Print cygs_beta, cygs_k, cygs_m

```

```

C_ye_gelen_sac.Print cygs_r, cygs_gama

cygs_z = cygs_parca_kalınligı / (Tan(cygs_gama))
cygs_b = Sqr(cygs_parca_kalınligı ^ 2 + cygs_z ^ 2)
cygs_e = (cygs_b + cygs_KM) * Sin(ct_alfa)
cygs_f = (cygs_b + cygs_KM) * Cos(ct_alfa)
cygs_y = (cygs_parca_kalınligı ^ 2 - cygs_k ^ 2 + cygs_m ^ 2) / (2 * cygs_m)
cygs_x = Sqr(cygs_parca_kalınligı ^ 2 - cygs_y ^ 2)
cygs_ms_boyu = 0.5 ' Hayali Muhafaza sacı boyu aşağıda kesim yaparken kullanılan boy , bu boy ne kadar uzun olursa sonuc
okadar yakın oluyor, cunku ct_x acısına göre çizgi çizip kesiyoruz sonucun tam olması için bu boyu uzun tutuyoruz.

```

```
End Sub
```

```

Sub pygs_data()
pygs_y = gys_4 - 0.02
pygs_parca_kalınligı = 0.016 ' 16 mm
'0.02 = 20 mm sabit olacak olan gys_4 mesafesinden saca uzaklık
alfa1 = Atn((pygs_y - as_3) / (gys_1 - gys_3 - r1 - as_parca_kalınligı))
'P_ye_gelen_sac.Print "r1=", r1, (gys_1 - gys_3 - r1 - as_parca_kalınligı), (pygs_y)
'alfa1 = Atn(217.2 / 802)
alfa = alfa1 * 180 / 3.141592654
beta = 90 - alfa
'alfa = 15.15, beta = 74.85 derece
'P_ye_gelen_sac.Print alfa, beta

```

```
' küçük üçgen için
```

```
beta1 = Tan(3.141592654 / 2 - alfa1)
'beta1=(16/k)
```

```

pygs_k = (pygs_parca_kalınligı / beta1)
'P_ye_gelen_sac.Print beta1, pygs_k
'ayak sacına yapışacak olan yüzeyin yandan görünen hattının uzunluğu m olsun
m = Sqr(pygs_k ^ 2 + pygs_parca_kalınligı ^ 2)
'P_ye_gelen_sac.Print m, ((pygs_y - as_3) / 2), (pygs_y - as_3), (((pygs_y - as_3) / 2) + m)
End Sub

```

```
Option Explicit
```

```
Public OKUD, OKAD, AKOD, AKAD, N4SD, CGSD, PGSD, GYSN3N4AD, CSKSAD As Double
```

```
Public kaynak_counter As Integer
```

```
Sub Kaynak()
```

```
' x_factor = kaynak dikisi işleminde tek taraftan kaynak yapılıyorsa 0.7 , Cift taraflı kaynak yapılıyorsa 0.5 alınır.
Kaynak_Dikisi_Boyutu = Kullanilan_Parca_Kalınligı * x_factor
```

```
Kaynak_Dikis_Boyutu = Kaynak_Dikisi_Boyutu * Sqr(2)
```

```
End Sub
```

```
Sub Kaynak_Dikisi_Atama()
```

```

.....
' ..... KAYNAK DİKİŞİ ATAMA .....
.....
' ..... KAYNAK NUMARALARINA DİKKAT ET .....

```

```
Kullanilan_Parca_Kalınligı = parca_kalınligı * 3 / 4 ' çok kalın gözüktüğü için bu işlem yapıldı
```

```
x_factor = 0.5
```

```
Call Kaynak
```

```
'1
```

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 * 8 / 9, -m_o_öteleme_z, False, 2, Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", gys_1 + m_o_öteleme_x, gys_2 / 9 * 8 + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
```

```
' ..... AssemblyDoc.InsertWeld ( type, shape, topDelta, bottomDelta, radius, part) .....

```

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyutu, "D:\Dirinler_Makina A.Ş\GövDe\bead1.sldprt")
```

```
asubl.ClearSelection2 True
```

```
Call asubl.EditRebuild3
```


8

```

Call asubl.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1 + m_o_oteleme_x, gys_2 - (pt_boyu / 2) + m_o_oteleme_y,
m_o_oteleme_z + cs_parca_kalinligi + parcalar_arasi_bosluk, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1 + pt_mesafe + pt_parca_kalinligi + m_o_oteleme_x, gys_2 -
(pt_boyu / 2) + m_o_oteleme_y, mesafe / 2, True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1 + pt_mesafe + (pt_parca_kalinligi / 2) + m_o_oteleme_x,
gys_2 - pt_boyu + m_o_oteleme_y, mesafe / 2, True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1 + pt_mesafe + m_o_oteleme_x, gys_2 - (pt_boyu / 2) +
m_o_oteleme_y, mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead8.sldprt")
asubl.ClearSelection2 True

```

'9 c iç ve alt plaka alt ve iç tarafı

```

Call asubl.Extension.SelectByID2("", "FACE", (gys_1 - gys_3) + m_o_oteleme_x, gys_2 - pt_boyu + m_o_oteleme_y,
m_o_oteleme_z + cs_parca_kalinligi + parcalar_arasi_bosluk, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_1 - (gys_3 / 2)) + m_o_oteleme_x, gys_2 + m_o_oteleme_y,
m_o_oteleme_z + (mesafe / 2 - ap_r2), True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_1 - gys_3 - r1) + m_o_oteleme_x, gys_2 + (ap_parca_kalinligi / 2) +
m_o_oteleme_y, -m_o_oteleme_z * 4, True, 2, Nothing, 0)
' bu yüzey secimi iptal sebebi 3 nolu kaynak ta yazıyor..
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead9.sldprt")
asubl.ClearSelection2 True

```

'10 c takviyesi alt plaka (ön)

```

Call asubl.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1 + pt_mesafe + pt_parca_kalinligi + m_o_oteleme_x, gys_2 -
(pt_boyu / 2) + m_o_oteleme_y, mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_1 - (gys_3 / 2)) + m_o_oteleme_x, gys_2 + m_o_oteleme_y,
m_o_oteleme_z + (mesafe / 2 - ap_r2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead10.sldprt")
asubl.ClearSelection2 True

```

'11 c takviyesi alt plaka

```

Call asubl.Extension.SelectByID2("", "FACE", gys_1 - gys_3 - r1 + pt_mesafe + m_o_oteleme_x, gys_2 - (pt_boyu / 2) +
m_o_oteleme_y, mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_1 - (gys_3 / 2)) + m_o_oteleme_x, gys_2 + m_o_oteleme_y,
m_o_oteleme_z + (mesafe / 2 - ap_r2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead11.sldprt")
asubl.ClearSelection2 True

```

```

asubl.ViewZoomtofit2
asubl.EditRebuild3

```

Call pygs_data

```

Kullanilan_Parca_Kalinligi = pygs_parca_kalinligi * 3 / 4 ' Pygs ile a.plakayı birbirine baglayan bu kaynak dikişi baska bir
radius degerini yapmıyor maks. 8,5 mm izin veriyor dolayısı ile burada parca kalınlığı olarak 12mm degeri kullanıldı 12/2 = 6 ,
6 *sqr(2) = 8.48 yapar...
x_factor = 0.5
Call Kaynak

```

'12 pygs alt

```

Call asubl.Extension.SelectByID2("", "FACE", ((gys_1 - gys_3 - r1 - as_parca_kalinligi)) / 2 + as_parca_kalinligi +
m_o_oteleme_x, ((as_3 - m) + (pygs_y - as_3) / 2) + m_o_oteleme_y, (mesafe / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_1 - gys_3 - r1) + m_o_oteleme_x, gys_2 + (ap_parca_kalinligi / 2) +
m_o_oteleme_y, -m_o_oteleme_z * 4, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead12.sldprt")
asubl.ClearSelection2 True

```

```

asubl.ViewZoomtofit2
asubl.EditRebuild3

```

Kullanilan_Parca_Kalinligi = pygs_parca_kalinligi

```

x_factor = 0.5
Call Kaynak

```

'13 pygs üst

```

asubl.ClearSelection2 True
Call asubl.Extension.SelectByID2("", "FACE", ((gys_1 - gys_3 - r1 - as_parca_kalinligi)) / 2 + as_parca_kalinligi +
m_o_oteleme_x, ((as_3) + (pygs_y - as_3) / 2) + m_o_oteleme_y, (mesafe / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_1 - gys_3 - r1) + m_o_oteleme_x, gys_2 + (ap_parca_kalinligi / 2) +
m_o_oteleme_y, -m_o_oteleme_z * 4, True, 2, Nothing, 0)

```

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead13.sldprt")
asubl.ClearSelection2 True
```

```
asubl.ViewZoomtofit2
asubl.EditRebuild3
```

Kullanilan_Parca_Kalinligi = pygs_parca_kalinligi * 9 / 10 ' yan tarafta yapılan işlemin sebebi daha büyük radiusu programın kabul etmemesidir..

```
'x_factor = 0.5
x_factor = PGSD
Call Kaynak
```

'14 pygs alt-gys

```
Call asubl.Extension.SelectByID2("", "FACE", ((gys_1 - gys_3 - r1 - as_parca_kalinligi)) / 2 + as_parca_kalinligi +
m_o_oteleme_x, ((as_3 - m) + (pygs_y - as_3) / 2) + m_o_oteleme_y, (mesafe / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead14.sldprt")
asubl.ClearSelection2 True
```

'Kullanilan_Parca_Kalinligi = cygs_parca_kalinligi ' pygs ile cygs parca kalınlığı aynıdır...

```
'x_factor = 0.5
Call Kaynak
```

'15 pygs üst-gys

Kullanilan_Parca_Kalinligi = pygs_parca_kalinligi * 9 / 10 ' yan tarafta yapılan işlemin sebebi daha büyük radiusu programın kabul etmemesidir..

```
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", ((gys_1 - gys_3 - r1 - as_parca_kalinligi)) / 2 + as_parca_kalinligi +
m_o_oteleme_x, ((as_3) + (pygs_y - as_3) / 2) + m_o_oteleme_y, (mesafe / 2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead15.sldprt")
asubl.ClearSelection2 True
```

```
asubl.ViewZoomtofit2
asubl.EditRebuild3
```

```
Kullanilan_Parca_Kalinligi = pygs_parca_kalinligi
x_factor = 0.5
Call Kaynak
```

' 16 pygs alt_ ayak sacı

```
Call asubl.Extension.SelectByID2("", "FACE", as_parca_kalinligi + m_o_oteleme_x, as_3 * 8 / 9 + m_o_oteleme_y, (mesafe /
2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", ((gys_1 - gys_3 - r1 - as_parca_kalinligi)) / 2 + as_parca_kalinligi +
m_o_oteleme_x, ((as_3 - m) + (pygs_y - as_3) / 2) + m_o_oteleme_y, (mesafe / 2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead16.sldprt")
asubl.ClearSelection2 True
```

' 17 ayaksacı_gys

```
Kullanilan_Parca_Kalinligi = as_parca_kalinligi
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", as_parca_kalinligi + m_o_oteleme_x, as_3 * 8 / 9 + m_o_oteleme_y, (mesafe /
2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead17.sldprt")
asubl.ClearSelection2 True
```

' 18 ayaksacı_cygs_alt

```
Call C_yegelen_Rutin
```

```
Kullanilan_Parca_Kalinligi = cygs_parca_kalinligi
x_factor = 0.5
Call Kaynak
```



```

Call asubl.Extension.SelectByID2("", "FACE", as_parca_kalnlığı + m_o_öteleme_x, as_3 * 8 / 9 + m_o_öteleme_y, (mesafe / 2), False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - cygs_y - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - cygs_y - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead18.sldprt")
asubl.ClearSelection2 True

```

'19 cygs_alt_gys

```

Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - cygs_y - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - cygs_y - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead19.sldprt")
asubl.ClearSelection2 True

```

'20 cygs_üst_gys

```

Kullanilan_Parca_Kalnlığı = cygs_parca_kalnlığı
'x_factor = 0.5
x_factor = CGSD
Call Kaynak

```

```

Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead20.sldprt")
asubl.ClearSelection2 True

```

```

Kullanilan_Parca_Kalnlığı = cygs_parca_kalnlığı
x_factor = 0.5
Call Kaynak

```

'21 cygs_üst_no_5_arka

'asagıda gys_14 den 10 mm asagıdaki yuzeyin y koordinati

```

Call asubl.Extension.SelectByID2("", "FACE", X_7 + m_o_öteleme_x, gys_14 - 0.01 + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu * 2, "D:\Dirinler_Makina A.Ş\GövDe\bead21.sldprt")
asubl.ClearSelection2 True

```

'22 cygs_üst_no_5_ön

'asagıda gys_14 den 10 mm asagıdaki yuzeyin y koordinati

```

Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalnlığı + m_o_öteleme_x, gys_14 - 0.01 + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If

```

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead22.sldprt")
asubl.ClearSelection2 True
```

'23 cygs_üst_no_4_arka

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalnlığı + X_6 + m_o_öteleme_x, gys_14 +
m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) +
m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) +
m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu * 2, "D:\Dirinler_Makina A.Ş\GövDe\bead23.sldprt")
asubl.ClearSelection2 True
```

'24 cygs_üst_no_4_ön

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalnlığı + X_6 + no_4s_parca_kalnlığı +
m_o_öteleme_x, gys_14 + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) +
m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) +
m_o_öteleme_x, ((gys_5 - cygs_r - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead24.sldprt")
asubl.ClearSelection2 True
```

```
Kullanilan_Parca_Kalinligi = no_5s_parca_kalnlığı
x_factor = 0.5
Call Kaynak
```

'25 gys_no_5_arka

```
'asagıda gys_14 den 10 mm asagıdaki yuzeyin y koordinati
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + m_o_öteleme_x, gys_14 - 0.01 + m_o_öteleme_y, m_o_öteleme_z +
mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead25.sldprt")
asubl.ClearSelection2 True
```

'26 gys_no_5_ön

```
'asagıda gys_14 den 10 mm asagıdaki yuzeyin y koordinati
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalnlığı + m_o_öteleme_x, gys_14 - 0.01 +
m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead26.sldprt")
asubl.ClearSelection2 True
```

```
Kullanilan_Parca_Kalinligi = no_4s_parca_kalnlığı
x_factor = 0.5
x_factor = N4SD
Call Kaynak
```

'27 gys_no_4_arka

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalnlığı + X_6 + m_o_öteleme_x, gys_14 +
m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead27.sldprt")
asubl.ClearSelection2 True
```

'28 gys_no_4_ön

```
Kullanilan_Parca_Kalinligi = no_4s_parca_kalnlığı
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, False, 2, Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + no_4s_parca_kalinligi +
m_o_oteleme_x, gys_14 + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead28.sldprt")
asubl.ClearSelection2 True
```

'29 arka yatak destek sacı

```
Kullanilan_Parca_Kalinligi = ayds_boy
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + m_o_oteleme_x, gys_14 - 0.01 + m_o_oteleme_y, m_o_oteleme_z +
mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 - (ayds_boy / 2) + m_o_oteleme_x, ((eksen_1 + gys_2 +
ap_parca_kalinligi) - eksen_2) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + ayds_r2, True, 2,
Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead29.sldprt")
asubl.ClearSelection2 True
```

'30 ön yatak destek sacı

```
Kullanilan_Parca_Kalinligi = öyds_boy
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + m_o_oteleme_x, gys_14 +
m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 - (öyds_boy / 2) + m_o_oteleme_x,
((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi)) - eksen_2 + üst_girinti) + m_o_oteleme_y,
m_o_oteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + öyds_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead30.sldprt")
asubl.ClearSelection2 True
```

'31 arka yatak_arka

```
Kullanilan_Parca_Kalinligi = (ay_boy - X_4 - X_5 - no_4s_parca_kalinligi - No_3_parca_kalinligi) / 3
x_factor = 0.5
x_factor = AKAD
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + m_o_oteleme_x, gys_14 +
m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + no_4s_parca_kalinligi + 0.01 +
m_o_oteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + üst_girinti))) + m_o_oteleme_y,
m_o_oteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + ay_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead31.sldprt")
asubl.ClearSelection2 True
```

'32 arka yatak_ön

```
Kullanilan_Parca_Kalinligi = X_4
x_factor = 0.5
x_factor = AKOD
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + no_4s_parca_kalinligi + X_5 +
No_3_parca_kalinligi + m_o_oteleme_x, gys_10 - 0.05 + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + no_4s_parca_kalinligi + 0.01 +
m_o_oteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + üst_girinti))) + m_o_oteleme_y,
m_o_oteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + ay_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead32.sldprt")
asubl.ClearSelection2 True
```

'33 orta yatak_arka

```
Kullanilan_Parca_Kalinligi = (oy_boy - X_3 - X_2 - No_1_parca_kalinligi - No_2_parca_kalinligi)
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - X_3 -
No_2_parca_kalinligi) + m_o_oteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi +
üst_girinti))) + oy_r2 + 2 * No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2, Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınligı - X_1 + (X_2 / 2)) + m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınligı + üst_girinti))) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + oy_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead33.sldprt")
asubl.ClearSelection2 True
```

'34 orta yatak_ön

```
Kullanilan_Parca_Kalınligı = X_2 / 2
'x_factor = 0.5
x_factor = OKUD
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınligı - X_1) + m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınligı + üst_girinti))) + oy_r2 + 2 * No_1_parca_kalınligı + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınligı - X_1 + (X_2 / 2)) + m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınligı + üst_girinti))) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + oy_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead34.sldprt")
asubl.ClearSelection2 True
```

'35 Yan kapak_destek_sacı arka_ara_kapama_sacı

```
Kullanilan_Parca_Kalınligı = No1_parca_kalınligı
x_factor = 0.35 'bu deger verildi cunku 0.5 buyuk geliyor BU DEGERI DEGIŞTİRME
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalınligı + X_6 - ((X_6 + X_7 + no_5s_parca_kalınligı) - (gys_11 + girinti)) + No1_parca_kalınligı + m_o_öteleme_x, gys_10 + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalınligı + X_6 - 0.005 + m_o_öteleme_x, (gys_10 - üst_girinti) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead35.sldprt")
asubl.ClearSelection2 True
```

```
asubl.ViewZoomtofit2
asubl.EditRebuild3
```

Call asubl.SetUserPreferenceToggle(197, False) ' kaynak işaretlerini kaldırıyoruz bu komut ile

'36 no_3--gys

```
Kullanilan_Parca_Kalınligı = No_3_parca_kalınligı
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalınligı + X_6 + no_4s_parca_kalınligı + X_5 + No_3_parca_kalınligı + m_o_öteleme_x, gys_10 - 0.05 + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead36.sldprt")
asubl.ClearSelection2 True
```

'37 no_3--2_3_ara_sacı

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalınligı + X_6 + no_4s_parca_kalınligı + X_5 + No_3_parca_kalınligı + m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınligı + üst_girinti))) + oy_r2 + 2 * No_1_parca_kalınligı + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalınligı + X_6 + no_4s_parca_kalınligı + X_5 + No_3_parca_kalınligı + 0.05 + m_o_öteleme_x, (eksen_1 + gys_2 + ap_parca_kalınligı - Y_1) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead37.sldprt")
asubl.ClearSelection2 True
```

'38 2_3_ara_sacı--gys

```
Kullanilan_Parca_Kalınligı = No_2_3_parca_kalınligı
'x_factor = 0.5
x_factor = GYSN3N4AD
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, False, 2, Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + no_4s_parca_kalinligi + X_5 +
No_3_parca_kalinligi + 0.05 + m_o_oteleme_x, (eksen_1 + gys_2 + ap_parca_kalinligi - Y_1) + m_o_oteleme_y,
m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead38.sldprt")
asubl.ClearSelection2 True
```

```
'39 2_3 ara_saci--no_2
Kullanilan_Parca_Kalinligi = No_2_3_parca_kalinligi
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - X_3 -
No_2_parca_kalinligi) + m_o_oteleme_x, gys_10 - 0.05 + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", X_7 + no_5s_parca_kalinligi + X_6 + no_4s_parca_kalinligi + X_5 +
No_3_parca_kalinligi + 0.05 + m_o_oteleme_x, (eksen_1 + gys_2 + ap_parca_kalinligi - Y_1) + m_o_oteleme_y,
m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead39.sldprt")
asubl.ClearSelection2 True
```

```
Kullanilan_Parca_Kalinligi = No_2_parca_kalinligi
x_factor = 0.5
x_factor = GYSN3N4AD
Call Kaynak
```

```
'40 no_2--gys_arca
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - X_3 -
No_2_parca_kalinligi) + m_o_oteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi +
üst_girinti))) + oy_r2 + 2 * No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead40.sldprt")
asubl.ClearSelection2 True
```

```
'41 no_2--gys_ön
Kullanilan_Parca_Kalinligi = No_2_parca_kalinligi
x_factor = 0.5
Call Kaynak
```

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - X_3) +
m_o_oteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + üst_girinti))) + oy_r2 + 2 *
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead41.sldprt")
asubl.ClearSelection2 True
```

```
Kullanilan_Parca_Kalinligi = No_1_parca_kalinligi
x_factor = 0.5
Call Kaynak
```

```
'42 no_1alın--gys_arca
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalinligi - X_1 - No_1_parca_kalinligi) +
m_o_oteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + üst_girinti))) + oy_r2 + 2 *
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead42.sldprt")
asubl.ClearSelection2 True
```

```
'43 no_1alın--gys_ön
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalinligi - X_1) + m_o_oteleme_x, ((gys_10 -
üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + üst_girinti))) + oy_r2 + 2 * No_1_parca_kalinligi +
m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead43.sldprt")
asubl.ClearSelection2 True
```

```
'44 burc takviyesi_orta yatak_üst
Kullanilan_Parca_Kalinligi = bt_parca_kalinligi * 2.9 / 4 ' Bu işlemi yapmamızın sebebi kaynak dikişinin max. 8.3 mm radius
kabul ediyor olmasıdır.
x_factor = 0.5
Call Kaynak
```

```

Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 +
m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + bt_uzaklık + bt_parca_kalınlgı, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + m_o_öteleme_y,
m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + oy_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead44.sldprt")
asubl.ClearSelection2 True

```

```

'45 burc takviyesi_orta yatak_alt
Kullanilan_Parca_Kalınlgı = bt_parca_kalınlgı
x_factor = 0.5
Call Kaynak

```

```

bt_mesafe = Sqr((oy_r2) ^ 2 - ((mesafe / 2) - (bt_uzaklık + bt_parca_kalınlgı)) ^ 2)

```

```

Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + bt_mesafe +
m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + bt_uzaklık + (bt_parca_kalınlgı / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + m_o_öteleme_y,
m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + oy_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead45.sldprt")
asubl.ClearSelection2 True

```

```

'46 burc takviyesi_sol_1
Kullanilan_Parca_Kalınlgı = bt_parca_kalınlgı
x_factor = 0.5
Call Kaynak

```

```

Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 +
m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + bt_uzaklık + bt_parca_kalınlgı, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 + 2 *
No_1_parca_kalınlgı + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead46.sldprt")
asubl.ClearSelection2 True

```

```

'47 burc takviyesi_sol_2

```

```

Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 +
m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + bt_uzaklık + bt_parca_kalınlgı, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - X_3) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 + 2 *
No_1_parca_kalınlgı + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead47.sldprt")
asubl.ClearSelection2 True

```

```

'48 burc takviyesi_sag_1

```

```

Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 +
m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + bt_uzaklık, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 + 2 *
No_1_parca_kalınlgı + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead48.sldprt")
asubl.ClearSelection2 True

```

```

'49 burc takviyesi_sag_2

```

```

Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - (X_3 / 2)) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 +
m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + bt_uzaklık, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlgı - X_1 - No_1_parca_kalınlgı - X_3) +
m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlgı + üst_girinti))) + oy_r2 + 2 *
No_1_parca_kalınlgı + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead49.sldprt")
asubl.ClearSelection2 True

```

'50 no1--gys

Kullanilan_Parca_Kalinligi = No1_parca_kalinligi

x_factor = 0.5

Call Kaynak

```
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - (X_3 / 2)) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) +
No1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + (No1_y / 2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead50.sldprt")
asubl.ClearSelection2 True
```

'51 no1--no_1alın

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) + oy_r2 + 2 *
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - (X_3 / 2)) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) +
No1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + (No1_y / 2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead51.sldprt")
asubl.ClearSelection2 True
```

'52 no1--no_2

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - X_3) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) + oy_r2 + 2 *
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - (X_3 / 2)) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) +
No1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + (No1_y / 2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead52.sldprt")
asubl.ClearSelection2 True
```

'53 1_2 ara_kapama--no1

Kullanilan_Parca_Kalinligi = No_1_parca_kalinligi

x_factor = 0.5

Call Kaynak

bt_mesafe = Sqr((oy_r2) ^ 2 - ((mesafe / 2) - (bt_uzaklik + bt_parca_kalinligi)) ^ 2)

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - (X_3 / 2)) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) + bt_mesafe + bt_1 +
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) + oy_r2 + 2 *
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead53.sldprt")
asubl.ClearSelection2 True
```

'54 1_2 ara_kapama--gys

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - (X_3 / 2)) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) + bt_mesafe + bt_1 +
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_oteleme_y, m_o_oteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead54.sldprt")
asubl.ClearSelection2 True
```

'55 1_2 ara_kapama--no_2

```
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - (X_3 / 2)) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) + bt_mesafe + bt_1 +
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - oyms_parca_kalinligi - X_1 - No_1_parca_kalinligi - X_3) +
m_o_oteleme_x, ((gys_10 - ust_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalinligi + ust_girinti))) + oy_r2 + 2 *
No_1_parca_kalinligi + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead55.sldprt")
asubl.ClearSelection2 True
```

'56 orta_yatak--kızak sacı

Kullanilan_Parca_Kalinligi = X_2 / 2

'x_factor = 0.5

x_factor = OKAD
Call Kaynak

Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + Öteleme_x + m_o_öteleme_x, gys_2 + ap_parca_kalınlığı + eksen_1 - (oy_r2 * 2) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlığı - X_1 - No_1_parca_kalınlığı - (X_3 / 2)) + m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlığı + üst_girinti))) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2) + oy_r2, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead56.sldprt")
asubl.ClearSelection2 True

'57 kızak sacı--gys
Kullanilan_Parca_Kalinligi = ks_parca_kalınlığı
x_factor = 0.5
Call Kaynak

Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + Öteleme_x + m_o_öteleme_x, gys_2 + ap_parca_kalınlığı + eksen_1 - (oy_r2 * 2) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead57.sldprt")
asubl.ClearSelection2 True

'58 kızak sacı--no1
No1_y = 0.15
Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + Öteleme_x + m_o_öteleme_x, gys_2 + ap_parca_kalınlığı + eksen_1 - (oy_r2 * 2) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlığı - X_1 - No_1_parca_kalınlığı - (X_3 / 2)) + m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlığı + üst_girinti))) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (No1_y / 2), True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead58.sldprt")
asubl.ClearSelection2 True

'59 gys--no1
Kullanilan_Parca_Kalinligi = No1_parca_kalınlığı
x_factor = 0.5
Call Kaynak

Call asubl.Extension.SelectByID2("", "FACE", (gys_9 - öyms_parca_kalınlığı - X_1 - No_1_parca_kalınlığı - (X_3 / 2)) + m_o_öteleme_x, ((gys_10 - üst_girinti) - (gys_10 - (eksen_1 + gys_2 + ap_parca_kalınlığı + üst_girinti))) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (No1_y / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead59.sldprt")
asubl.ClearSelection2 True

asubl.ViewZoomtofit2
asubl.EditRebuild3

Call asubl.SetUserPreferenceToggle(197, False) ' kaynak işaretlerini kaldırıyoruz bu komut ile

'60 kızak_sacı--karsı_kızak
Kullanilan_Parca_Kalinligi = ks_parca_kalınlığı
x_factor = 0.5
Call Kaynak

kızak_y = (gys_2 + ap_parca_kalınlığı + eksen_1) - gys_5 - eksen_3 * 5

Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + Öteleme_x + m_o_öteleme_x, gys_2 + ap_parca_kalınlığı + eksen_1 - (oy_r2 * 2) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + (karsı_kızak_parca_kalınlığı / 2) + Öteleme_x + m_o_öteleme_x, gys_5 + kızak_y + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (karsı_kızak_x / 2), True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + (karsı_kızak_parca_kalınlığı / 2) + Öteleme_x + m_o_öteleme_x, gys_5 + (kızak_y / 2) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + karsı_kızak_x, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead60.sldprt")
asubl.ClearSelection2 True

'61 karsı kızak--gys


```

Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + (karsi_kizak_parca_kalinligi * 9 / 10) + Öteleme_x +
m_o_öteleme_x, gys_5 + kizak_y + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (karsi_kizak_x / 2), False, 2,
Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead61.sldprt")
asubl.ClearSelection2 True

```

```

'62 karsi kizak--yan kizak
Kullanilan_Parca_Kalinligi = yan_kizak_parca_kalinligi
x_factor = 0.5
Call Kaynak

```

```

Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + karsi_kizak_parca_kalinligi + Öteleme_x + m_o_öteleme_x,
gys_5 + (kizak_y / 2) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (karsi_kizak_x / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + karsi_kizak_parca_kalinligi + (yan_kizak_x / 2) + Öteleme_x +
m_o_öteleme_x, gys_5 + (kizak_y / 2) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk +
yan_kizak_parca_kalinligi, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead62.sldprt")
asubl.ClearSelection2 True

```

```

'63 yan kizak --gys
' asagidaki kaynak esitligi kullanildi cunku her iki parca da 30 mm ve bu deger cok buyuk dikis demektir.
Kullanilan_Parca_Kalinligi = ks_parca_kalinligi
x_factor = 0.5
Call Kaynak

```

```

Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + karsi_kizak_parca_kalinligi + yan_kizak_x + Öteleme_x +
m_o_öteleme_x, gys_5 + (kizak_y / 2) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk +
(yan_kizak_parca_kalinligi / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", cs_2 + cs_3 + karsi_kizak_parca_kalinligi + (yan_kizak_x / 2) + Öteleme_x +
m_o_öteleme_x, gys_5 + kizak_y + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (yan_kizak_parca_kalinligi /
2), True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead63.sldprt")
asubl.ClearSelection2 True

```

```

'64 E18-A--gys

```

```

Kullanilan_Parca_Kalinligi = e18_a_y
x_factor = 0.5
Call Kaynak

```

```

Call asubl.Extension.SelectByID2("", "FACE", gys_7 - e18_b_y - e18_a_x + m_o_öteleme_x, gys_5 + ((gys_2 +
ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3) / 2) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (e18_a_y /
2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", gys_7 - e18_b_y - (e18_a_x / 2) + m_o_öteleme_x, gys_5 + (gys_2 +
ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (e18_a_y / 2),
True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead64.sldprt")
asubl.ClearSelection2 True

```

```

'65 ön yatak - -gys
Kullanilan_Parca_Kalinligi = parca_kalinligi
x_factor = 0.5
Call Kaynak

```

```

Call asubl.Extension.SelectByID2("", "FACE", gys_9 - öyms_parca_kalinligi - ön_yatak_parca_kalinligi + m_o_öteleme_x,
gys_10 - 0.03 + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + mesafe / 2, False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, gys_2 / 10 + m_o_öteleme_y, m_o_öteleme_z, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead65.sldprt")
asubl.ClearSelection2 True

```

```

'66 E18A--E18B
Kullanilan_Parca_Kalinligi = e18_a_y
x_factor = 0.5
Call Kaynak

```

```

Call asubl.Extension.SelectByID2("", "FACE", gys_7 - e18_b_y - (e18_a_x / 2) + m_o_öteleme_x, gys_5 + ((gys_2 +
ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3) / 2) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + e18_a_y,
False, 2, Nothing, 0)

```

```
Call asubl.Extension.SelectByID2("", "FACE", gys_7 - e18_b_y + m_o_oteleme_x, gys_5 + ((gys_2 + ap_parca_kalinligi +
eksen_1 - gys_5 - eksen_3) / 2) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + e18_a_y + 0.01, True, 2,
Nothing, 0)
```

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead66.sldprt")
asubl.ClearSelection2 True
```

'67 E18B---Ön yatak

```
Call asubl.Extension.SelectByID2("", "FACE", gys_7 - (e18_b_y / 2) + m_o_oteleme_x, gys_5 + ((gys_2 + ap_parca_kalinligi
+ eksen_1 - gys_5 - eksen_3) / 2) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + e18_a_y + e18_b_x, False, 2,
Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", gys_7 - e18_b_y + m_o_oteleme_x, gys_5 + ((gys_2 + ap_parca_kalinligi +
eksen_1 - gys_5 - eksen_3) / 2) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + e18_b_x - 0.01, True, 2, Nothing,
0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", gys_9 - öyms_parca_kalinligi - (ön_yatak_parca_kalinligi / 2) +
m_o_oteleme_x, gys_5 + ((gys_2 + ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3)) + m_o_oteleme_y, m_o_oteleme_z +
parcalar_arasi_bosluk + (e18_a_y * 2), True, 2, Nothing, 0)
```

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead67.sldprt")
asubl.ClearSelection2 True
```

'68 Önyatak-- E18A

```
Call asubl.Extension.SelectByID2("", "FACE", gys_7 - e18_b_y - (e18_a_x / 2) + m_o_oteleme_x, gys_5 + ((gys_2 +
ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3) / 2) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + e18_a_y,
False, 2, Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", gys_9 - öyms_parca_kalinligi - (ön_yatak_parca_kalinligi / 2) +
m_o_oteleme_x, gys_5 + ((gys_2 + ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3)) + m_o_oteleme_y, m_o_oteleme_z +
parcalar_arasi_bosluk + (e18_a_y + e18_b_x), True, 2, Nothing, 0)
```

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead68.sldprt")
asubl.ClearSelection2 True
```

'69 E18B---öN_Yatak_mesafe_sacı

```
Call asubl.Extension.SelectByID2("", "FACE", gys_7 - (e18_b_y / 2) + m_o_oteleme_x, gys_5 + ((gys_2 + ap_parca_kalinligi
+ eksen_1 - gys_5 - eksen_3) / 2) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + e18_b_x, False, 2, Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", gys_7 + m_o_oteleme_x, gys_5 + ((gys_2 + ap_parca_kalinligi + eksen_1 -
gys_5 - eksen_3) / 2) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + (e18_b_x / 2), True, 2, Nothing, 0)
```

```
Call asubl.Extension.SelectByID2("", "FACE", gys_9 - (öyms_parca_kalinligi / 2) + m_o_oteleme_x, gys_5 + ((gys_2 +
ap_parca_kalinligi + eksen_1 - gys_5 - eksen_3)) + m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + (e18_a_y * 2),
True, 2, Nothing, 0)
```

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead69.sldprt")
asubl.ClearSelection2 True
```

'70 muhafaza sacı----c_takviyesi -arka...

Call C_yegelen_Rutin

```
Call asubl.Extension.SelectByID2("", "FACE", cXx + (ct_parca_boyu / 4) + m_o_oteleme_x, gys_5 + parcalar_arasi_bosluk +
m_o_oteleme_y, m_o_oteleme_z + parcalar_arasi_bosluk + (mesafe / 2), False, 2, Nothing, 0)
```

```
If ct_x < (pi / 2) Then
```

```
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalinligi ^ 2) - cygs_r * Tan(ct_alfa)) - 0.25
* Cos(ct_x)) + m_o_oteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2,
Nothing, 0)
```

```
Else
```

```
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalinligi ^ 2) + cygs_r * Tan(ct_alfa)) - 0.25
* Cos(ct_x)) + m_o_oteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, True, 2,
Nothing, 0)
```

```
End If
```

'asagıda girilen kaynak dikis degeri 2 mm max. degerdir diger degerler cok buyuk gelmektedir mevcut geometriye

```
Call asubl.InsertWeld("FILL", "FLT", 0, 0, 0.002, "D:\Dirinler_Makina A.Ş\GövDe\bead70.sldprt")
```

```
asubl.ClearSelection2 True
```

'71 pygs alt_--muhafaza_sacı

Kullanilan_Parca_Kalinligi = ms_parca_kalinligi * 0.98 ' Kaynak dikişinin atlabilmesi için carpan kullanılmıştır.

x_factor = 0.5

Call Kaynak

```
If ct_x < (pi / 2) Then
```

```
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalinligi ^ 2) - cygs_r * Tan(ct_alfa)) - 0.25
* Cos(ct_x)) + m_o_oteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_oteleme_y, m_o_oteleme_z + mesafe / 2, False, 2,
Nothing, 0)
```

```
Else
```

```
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
End If
```

```
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - cygs_y - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) / 2) + m_o_öteleme_x, ((gys_5 - cygs_r - cygs_y - as_2) / 2 + as_2) + m_o_öteleme_y, mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead71.sldprt")
asubl.ClearSelection2 True
```

' 72 muhafaza sacı---c_takviyesi --ön...

```
Call asubl.Extension.SelectByID2("", "FACE", cXx + (ct_parca_boyu / 4) + m_o_öteleme_x, gys_5 + parcalar_arasi_bosluk + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2), False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead72.sldprt")
asubl.ClearSelection2 True
```

' 73 c_takviyesi---c_sacı....

```
Call asubl.Extension.SelectByID2("", "FACE", cXx + (ct_parca_boyu / 4) + m_o_öteleme_x, gys_5 + parcalar_arasi_bosluk + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2), False, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", cXx - (ct_parca_boyu / 2) + m_o_öteleme_x, gys_5 + parcalar_arasi_bosluk + (ct_parca_kalnlığı / 2) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2), True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", cXx + (ct_parca_boyu / 4) + m_o_öteleme_x, gys_5 + parcalar_arasi_bosluk + ct_parca_kalnlığı + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2), True, 2, Nothing, 0)
Call asubl.Extension.SelectByID2("", "FACE", 0, 0, m_o_öteleme_z + cs_parca_kalnlığı, True, 2, Nothing, 0)
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead73.sldprt")
asubl.ClearSelection2 True
```

' 74 muhafaza_sacı---c_sacı---arka

```
Call C_yegelen_Rutin
Call asubl.Extension.SelectByID2("", "FACE", 0, 0, m_o_öteleme_z + cs_parca_kalnlığı, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead74.sldprt")
asubl.ClearSelection2 True
```

' 75 muhafaza_sacı---c_sacı---ön

```
Call C_yegelen_Rutin
Call asubl.Extension.SelectByID2("", "FACE", 0, 0, m_o_öteleme_z + cs_parca_kalnlığı, False, 2, Nothing, 0)
If ct_x < (pi / 2) Then
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) - cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
Else
Call asubl.Extension.SelectByID2("", "FACE", ((cXx - Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + cygs_r * Tan(ct_alfa)) - 0.25 * Cos(ct_x)) + Sqr(ms_x ^ 2 + ms_parca_kalnlığı ^ 2) + m_o_öteleme_x, gys_5 - cygs_r - 0.25 * Sin(ct_x) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
End If
Call asubl.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead75.sldprt")
```

```
asmb1.ClearSelection2 True
```

```
'76 c_ takviyesi_kızak sacı
```

```
Kullanılan_Parca_Kalınligi = ks_parca_kalınligi
```

```
x_factor = 0.5
```

```
Call Kaynak
```

```
Call asmb1.Extension.SelectByID2("", "FACE", cXx + (ct_parca_boyu / 4) + m_o_öteleme_x, gys_5 + parcalar_arasi_bosluk + ct_parca_kalınligi + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + (mesafe / 2), False, 2, Nothing, 0)
```

```
Call asmb1.Extension.SelectByID2("", "FACE", cs_2 + cs_3 - ks_parca_kalınligi + Öteleme_x + m_o_öteleme_x, gys_2 +
```

```
ap_parca_kalınligi + eksen_1 - (oy_r2 * 2) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, True, 2, Nothing, 0)
```

```
Call asmb1.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead76.sldprt")
```

```
asmb1.ClearSelection2 True
```

```
Kullanılan_Parca_Kalınligi = ks_parca_kalınligi
```

```
'x_factor = 0.5
```

```
x_factor = CSKSAD
```

```
Call Kaynak
```

```
'77 c_sacı--kızak sacı iç taraf_arka..
```

```
Call asmb1.Extension.SelectByID2("", "FACE", cs_2 + cs_3 - ks_parca_kalınligi + Öteleme_x + m_o_öteleme_x, gys_2 +
```

```
ap_parca_kalınligi + eksen_1 - (oy_r2 * 2) + m_o_öteleme_y, m_o_öteleme_z + mesafe / 2, False, 2, Nothing, 0)
```

```
Call asmb1.Extension.SelectByID2("", "FACE", 0, 0, m_o_öteleme_z + cs_parca_kalınligi, True, 2, Nothing, 0)
```

```
Call asmb1.Extension.SelectByID2("", "FACE", cs_2 + cs_3 / 2 + Öteleme_x + m_o_öteleme_x, cs_1 + Öteleme_y +
```

```
m_o_öteleme_y, m_o_öteleme_z + cs_parca_kalınligi / 2, True, 2, Nothing, 0)
```

```
Call asmb1.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead77.sldprt")
```

```
asmb1.ClearSelection2 True
```

```
asmb1.ViewZoomtofit2
```

```
asmb1.EditRebuild3
```

```
Call asmb1.SetUserPreferenceToggle(197, False) ' kaynak işaretlerini kaldırıyoruz bu komut ile
```

```
' Ön_Yatak--Ön_Yatak_Mesafe_Sacı
```

```
' aşağıda ilk egrisel yüzey secilemediği için iptal edildi, feature part olarak kaynak dikisi atıldı...
```

```
'Call asmb1.Extension.SelectByID2("", "FACE", gys_9 - öyms_parca_kalınligi + m_o_öteleme_x, (gys_5 + (gys_2 + ap_parca_kalınligi + eksen_1 - gys_5 - eksen_3) + ön_yatak_r) + m_o_öteleme_y, m_o_öteleme_z + ((mesafe - öyms_1) / 2) + 0.015, False, 2, Nothing, 0)
```

```
'Call asmb1.Extension.SelectByID2("", "FACE", (gys_9 - (öyms_parca_kalınligi / 2) + m_o_öteleme_x), (gys_5 + (gys_2 + ap_parca_kalınligi + eksen_1 - gys_5 - eksen_3) + (öyms_2 / 3) + m_o_öteleme_y), (m_o_öteleme_z + ((mesafe - öyms_1) / 2)), True, 2, Nothing, 0)
```

```
'Call asmb1.Extension.SelectByID2("", "FACE", (gys_9 - (öyms_parca_kalınligi / 2) + m_o_öteleme_x), (gys_5 + (gys_2 + ap_parca_kalınligi + eksen_1 - gys_5 - eksen_3) + (öyms_2 - öyms_r) + (öyms_r / (Sqr(2)))) + m_o_öteleme_y), (parcalar_arasi_bosluk + m_o_öteleme_z + ((mesafe - öyms_1) / 2) + (öyms_r - (öyms_r / (Sqr(2))))), True, 2, Nothing, 0) ' ,,,,,,2.191355339,+0.13336446609
```

```
'Call asmb1.Extension.SelectByID2("", "FACE", gys_9 - (öyms_parca_kalınligi / 2) + m_o_öteleme_x, gys_5 + (gys_2 + ap_parca_kalınligi + eksen_1 - gys_5 - eksen_3) + öyms_2 + m_o_öteleme_y, m_o_öteleme_z + (mesafe / 2) + parcalar_arasi_bosluk, True, 2, Nothing, 0)
```

```
'Call asmb1.Extension.SelectByID2("", "FACE", gys_9 - (öyms_parca_kalınligi / 2) + m_o_öteleme_x, gys_5 + (gys_2 + ap_parca_kalınligi + eksen_1 - gys_5 - eksen_3) + (öyms_2 - öyms_r) + (öyms_r / (Sqr(2)))) + m_o_öteleme_y, m_o_öteleme_z + parcalar_arasi_bosluk + ((mesafe - öyms_1) / 2) + öyms_1 - (öyms_r - (öyms_r / (Sqr(2))))), True, 2, Nothing, 0)
```

```
'Call asmb1.Extension.SelectByID2("", "FACE", gys_9 - (öyms_parca_kalınligi / 2) + m_o_öteleme_x, gys_5 + (gys_2 + ap_parca_kalınligi + eksen_1 - gys_5 - eksen_3) + (öyms_2 / 3) + m_o_öteleme_y, m_o_öteleme_z + ((mesafe - öyms_1) / 2) + öyms_1, True, 2, Nothing, 0)
```

```
'Call asmb1.InsertWeld("FILL", "FLT", 0, 0, Kaynak_Dikis_Boyu, "D:\Dirinler_Makina A.Ş\GövDe\bead70.sldprt")
```

```
'asmb1.ClearSelection2 True
```

```
'ana_form.Print (gys_9 - (öyms_parca_kalınligi / 2) + m_o_öteleme_x), (gys_5 + (gys_2 + ap_parca_kalınligi + eksen_1 - gys_5 - eksen_3) + (öyms_2 - öyms_r) + (öyms_r / (Sqr(2)))) + m_o_öteleme_y), (parcalar_arasi_bosluk + m_o_öteleme_z + ((mesafe - öyms_1) / 2) + (öyms_r - (öyms_r / (Sqr(2))))))
```

```
'part.ClearSelection2 True
```

```
asmb1.ViewZoomtofit2
```

```
asmb1.EditRebuild3
```

```
End Sub
```

```
Sub Birlestirme()
```



```

boolstatus = part.Extension.SelectByID2("Cut-Extrude1[26]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[27]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[28]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[29]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[30]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[31]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[32]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[33]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[34]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[35]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[36]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[37]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[38]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[39]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[40]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[41]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[42]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[43]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[44]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[45]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[46]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[47]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[48]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[49]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[50]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[51]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[52]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Cut-Extrude1[53]", "SOLIDBODY", 0, 0, 0, True, 2, Nothing, 0)
part.FeatureManager.InsertCombineFeature 15903, Nothing, Empty
End Sub

```

```
Sub Kesme()
```

```

m_o_öteleme_x = -gys_1 / 2
m_o_öteleme_y = -gys_10 / 2
m_o_öteleme_z = (parca_kalınlığı / 2 - parcalar_arasi_bosluk / 2)

```

```

boolstatus = part.Extension.SelectByID2("Right Plane", "PLANE", 0, 0, 0, True, 0, Nothing, 0)
part.CreatePlaneAtOffset3 gys_9, False, True
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.SketchManager.InsertSketch True

```

```

part.SketchRectangle -(m_o_öteleme_z + (mesafe / 2) + parcalar_arasi_bosluk), -gys_10, 0, -(mesafe * 2 + m_o_öteleme_z),
gys_10, 0, 1
part.ClearSelection2 True
boolstatus = part.Extension.SelectByID2("Line2", "SKETCHSEGMENT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line1", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line4", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
boolstatus = part.Extension.SelectByID2("Line3", "SKETCHSEGMENT", 0, 0, 0, True, 0, Nothing, 0)
part.FeatureManager.FeatureCut True, False, False, 0, 0, gys_9 * 2, 0, False, False, False, False, 0, 0, False, False, False, False,
0, 1, 1
part.SelectionManager.EnableContourSelection = 0

```

```

boolstatus = part.Extension.SelectByID2("Plane1", "PLANE", 0, 0, 0, False, 0, Nothing, 0)
part.BlankRefGeom

```

```

part.Save2 False
part.SaveAs2 "D:\Dirinler_Makina A.Ş\GövDe\Dirinler_Pres_Part.SLDPRТ", 0, True, False

```

```
End Sub
```

```
Sub Simetrik_Mirror_ile_Tekrar_Birlestirme()
```

' Asagıda mirror yapılırken secilen yüzey, ayak sacının sol alt kosesidir. Bu yüzeyin ve kullanılan koordinatlar, her
'''Modelde, her farklı modelde yer alacağı düşünüldüğü için kullanılmıştır.

```

m_o_öteleme_x = -gys_1 / 2
m_o_öteleme_y = -gys_10 / 2
m_o_öteleme_z = (parca_kalınlığı / 2 - parcalar_arasi_bosluk / 2)

```

```
part.ClearSelection2 True
```



```
boolstatus = part.Extension.SelectByID2("", "FACE", m_o_öteleme_x + 0.01, m_o_öteleme_y + 0.01, m_o_öteleme_z +  
mesafe / 2, False, 2, Nothing, 0)  
boolstatus = part.Extension.SelectByID2("Combine1", "SOLIDBODY", 0, 0, 0, True, 256, Nothing, 0)  
part.FeatureManager.InsertMirrorFeature True, False, True, True
```

```
part.Save2 False
```

```
End Sub
```