ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES

MSc THESIS

Osman Oktay AKÇALI

PARAMETRIC DESIGN of AUTOMOTIVE BALL JOINT USING COMPUTER ASSISTED 3D MODELLING

DEPARTMENT OF MECHANICAL ENGINEERING

ADANA, 2015

ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES

PARAMETRIC DESIGN of AUTOMOTIVE BALL JOINT USING COMPUTER ASSISTED 3D MODELLING

Osman Oktay AKÇALI

MSc THESIS

DEPARTMENT of MECHANICAL ENGINEERING

We certify that the thesis titled above was reviewed and approved for the award of degree of the Master of Science by the board of jury on ..../..../2015.

……………………….	………………………………..	……………………………….
Prof. Dr. Necdet GEREN	Prof. Dr. Melih BAYRAMOĞLU	Asst. Prof. Dr. Çetin CANPOLAT
SUPERVISOR	MEMBER	MEMBER

This MSc Thesis is written at the Department of Institute of Natural And Applied Sciences of Çukurova University.
Registration Number:

Prof. Dr. Mustafa GÖK
Director
Institute of Natural and Applied Sciences

ABSTRACT

MSc THESIS

PARAMETRIC DESIGN of AUTOMOTIVE BALL JOINT USING COMPUTER ASSISTED 3D MODELING

Osman Oktay AKÇALI

ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES
DEPARTMENT of MECHANICAL ENGINEERING

Supervisor : Prof. Dr. Necdet GEREN
Year: 2015, Pages: 296
Jury : Prof. Dr. Necdet GEREN
: Prof. Dr. Melih BAYRAMOĞLU
: Asst. Prof. Dr. Çetin CANPOLAT

Ball joints which are used in steering system of vehicles have hundreds of different types and configurations. In this study, a platform called as "*parametric design platform*" has been developed for the parametric design of automotive ball joint using 3D modelling to reduce design time and cost. The developed platform can be used for part and assembly design with top-down design approach. In the development of the platform original *"flow diagram"* and *"decomposition technique"* has been used. The major advantage of the proposed system is that the system can parametrically change assembly, part, part material, feature, geometry and dimensions in a programmable environment. This provides a wide range of alternative solutions to design every parts of ball joints systematically. Whereas parametric systems, which is not programmable, provide change in dimensions only. After completing the development of *"parametric design platform"* tests have been applied to validate design and the results demonstrate the practicability and validity of the parametric system.

Key Words: Parametric Design, Ball Joint, CAD Modeling, Automotive Steering System

ÖZ

YÜKSEK LİSANS TEZİ

BİLGİSAYAR DESTEKLİ 3 BOYUTLU MODELLEME KULLANARAK MOTORLU ARAÇ ROT BAŞLARININ PARAMETRİK TASARIMI

Osman Oktay AKÇALI

ÇUKUROVA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ
MAKİNE MÜHENDİSLİĞİ ANABİLİM DALI

Danışman : Prof. Dr. Necdet GEREN
Yıl: 2015, Sayfa: 296
Jüri : Prof. Dr. Necdet GEREN
: Prof. Dr. Melih BAYRAMOĞLU
: Yrd. Doç. Dr. Çetin CANPOLAT

Araçların direksiyon sistemlerinde kullanılan rotbaşları yüzlerce çeşit ve konfigürasyona sahiptir. Bu çalışmada, tasarım zamanı ve maliyetini düşürmek için motorlu taşıt rotbaşlarının parametrik tasarımı amacıyla 3 boyutlu modelleme kullanılarak *"parametrik tasarım platformu"* adında bir platform geliştirilmiştir. Geliştirilen platform yukarıdan aşağıya tasarım yaklaşımı kullanarak parça ve montaj tasarımında kullanılabilir. Platformun geliştirilmesinde için orijinal *"akış diyagramı"* ve *"ayrıştırma tekniği"* kullanılmıştır. Önerilen sistemin en büyük avantajı sistem programlanabilir bir ortamda montaj, parça, parça malzemesi, modelleme özelliği, geometri ve ölçüleri değiştirebilir. Bu rotbaşlarının her parçasının tasarımı için geniş alternatif çözümler sağlar. Programlama özelliği olmayan parametrik sistemler sadece ölçüleri değiştirebilir. Parametrik sistem tamamlandıktan sonra doğrulama testleri yapılmıştır ve sonuçlar parametrik sistemin geçerliliğini ve uygulanabilirliğini göstermiştir.

Anahtar Kelimeler: Parametrik tasarım, Rot başı, Bilgisayar destekli modelleme, Direksiyon Sistemi

ACKNOWLEDGEMENTS

I would like to special thank to my advisor Prof. Dr. Necdet GEREN, for his advises, supports, guidance and especially endless encouragements during my MSc Thesis study.

I would like to thank to my family; my wife Elif İZGİ AKÇALI, and my son Yiğit AKÇALI, for their supports and patience.

Finally, I am forever indebted to my parents Mehmet and Hayriye AKÇALI for their supports and encouragements.

CONTENTS **PAGE**

LIST OF TABLES **PAGE**

LIST OF FIGURES **PAGE**

1. INTRODUCTION

People do design. Planning and implementing change in the world around us is one of the key things that make us human. Language is what we say. Design and making is what we do. Computers are simply a new medium for this ancient enterprise. True, they are truly active medium. As general symbol processors computers can present almost limitless kinds of tools. With craft and care we can program them to do much of what we call design. But it is not all. Designers continue to amaze us with new function and form. Sometimes new work embodies wisdom, a precious commodity in a finite world. To the human enterprise of design, parametric systems bring fresh and needed new capabilities in adapting to context and contingency and exploring the possibilities inherent in an idea (Woodbury, 2010).

1.1. Parametric Design

Parametric design has been incorporated into various main CAD/CAM systems, such as Unigraphics, Pro/Engineer and SolidWorks, etc. Parametric design deals with variable dimensions as control parameters and allows the designers to modify the existing designs by changing the parameter values. Such capability increases the efficiency in the design of part families whose members differ only in dimensions (Wu, Fuh, & Lee, 2007).

Usually, when the designer makes a decision on the design, the related parameters must be calculated accurately and, then the design is to be verified to meet the process conditions. This, first of all, requires that the designer is to have a good expertise to meet the requirements of the job. Secondly, the time spent for the job increases the designer's workload to some extent. The evaluation module could help to predict related process parameters in advance, assess the result of the design and provide guidance for modification (Wu, Fuh, & Lee, 2007).

With the increasing demand of flexible tools for Computer Aided Design (CAD), Parametric Modeling is becoming a mainstream of Computer Aided Design

(CAD) software, in order to make variations in the design process less difficult. This is traditionally called Parametric Design. Until recently, parametric design was understood as highly sophisticated and expensive software made exclusively for manufacturing in aerospace, shipping and automobile industries. However, designer's demands for flexibility to make changes without deleting or redrawing in a computer has pushed the incorporation of parametric modeling as standard tools in traditional CAD programs (Wu, Fuh, & Lee, 2007).

Variations in design are a fundamental part of the design process in the search for solutions to design problems. Design variations support improvement of design which in turn improves the quality of designed artifacts. Designers constantly go back and forth between different alternatives in the universe of possible solutions, working in a particular part at a given time, or looking back at the whole from a broader perspective. This is a continuous and iterative search process of variations of a design idea, and it is very likely to revisit a previously abandoned solution to rework it. As a result, designers demand flexible tools that allow variations in the design process until a solution is established for further development (Hernandez, 2006).

Parametric Design is the process of designing in environment where design variations are effortless, thus replacing singularity with multiplicity in the design process. Parametric design is done with the aid of Parametric Models. A parametric model is a computer representation of a design constructed with geometrical entities that have attributes (properties) that are fixed and others that can vary. The variable attributes are also called parameters and the fixed attributes are said to be constrained. The designer changes the parameters in the parametric model to search for different alternative solutions to the problem at hand. The parametric model responds to the changes by adapting or reconfiguring to the new values of the parameters without erasing or redrawing. In parametric design, designers use declared parameters to define a form. This requires rigorous thinking in order to build a sophisticated geometrical structure embedded in a complex model that is flexible enough for doing variations. Therefore, the designer must anticipate which kinds of variations he wants to explore in order to determine the kinds of transformations the parametric model should do. This is a very difficult task due of the unpredictable nature of the design process (Hernandez, 2006).

Commercialized 3D CAD (computer-aided design) systems provide algorithms that might describe a master model that defines the correlation of each feature with the parametric structure. The model has shape information that satisfies design rules and attributes that represent character information. The attributes of parts should be managed from the commencement of design (Mok, Kim, & Kim, 2011).

Parametric system involves three phases (Badiru A. B., 1992);

Definition: A problem definition phase; the problem associated with engineering design is presented.

Reasoning: A problem solving phase; symbolic processing is done while requirements and constraints are being evaluated until a solution is reached.

Presentation: A recommendation phase; the recommended solution is graphically presented to the user.

1.2. Ball Joints

Ball Joints (or universal joints) are especially used in steering system of vehicles in automotive industry. According to shape and using place in the steering system, ball joints are called as rod end, axial joint, pitman arm, guide joint, wishbone etc. Steering and chassis components are safety parts, which are designed, made and assembled with particular care (Fig. 1.1 and Fig. 1.2). They must provide an ultimate level of safety in all driving situations of the vehicle, which can occur, whether influenced by the driver or not.

The upper and lower shell of ball joints can be made of steel, plastic or combination of both materials. Plastic shells are made of polyethylene, for instance. The friction pairing results from the shell material and the ball surface. The moments (torque, motive torque, initial break away torque) and the compliance, the ball play (better: the stroke) derives from the friction pairing. Steel shell construction provides an extremely low compliance both in radial and axial direction. High-grade plastic

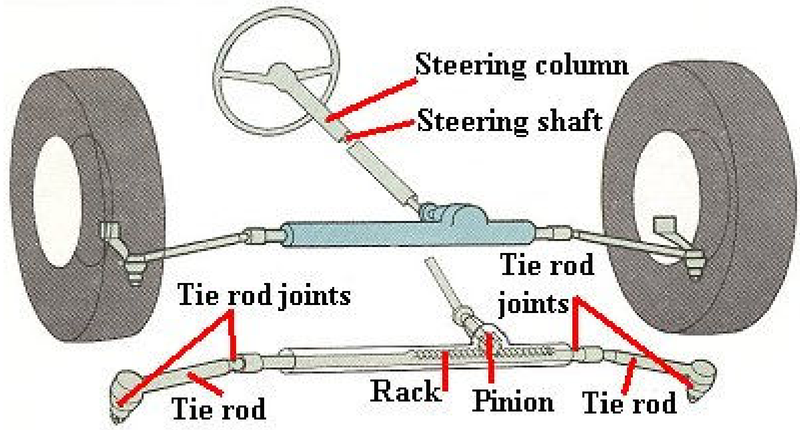


Figure 1.1. Steering system of a vehicle (http://www2.mae.ufl.edu/designlab/Class%20Projects/Background%20Information/Steering.htm)

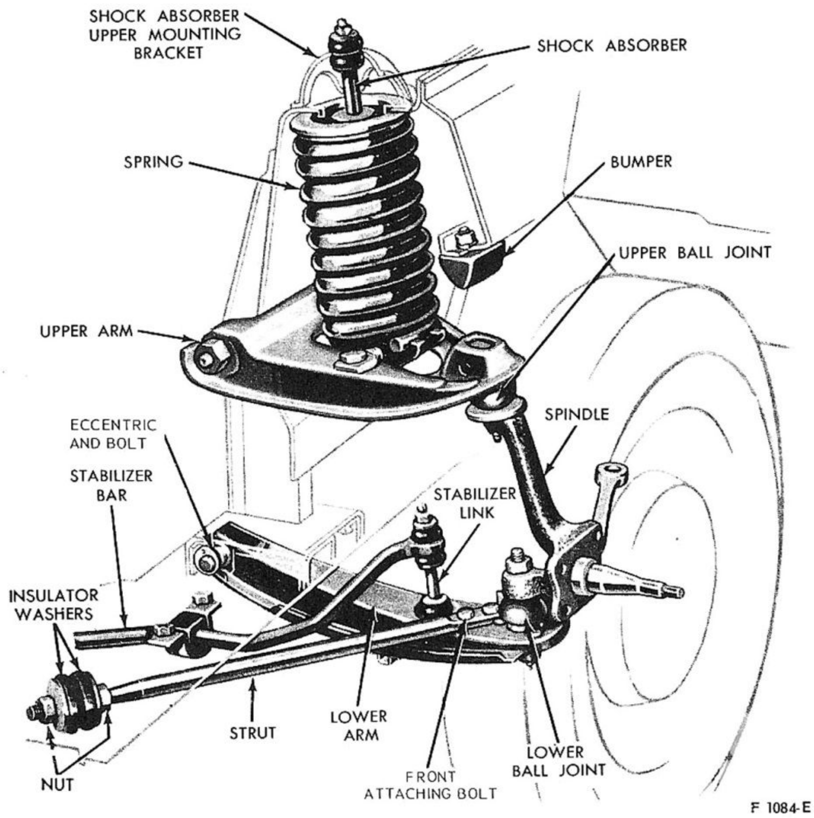


Figure 1.2. Wheel and steering system components of a vehicle (http://www.frontendalignmentcost.com/front-end-alignment-tools/)

shell materials also provide a low compliance couplet with excellent anti-friction characteristics and excellent damping (www.zf.com).

Ball Joints consists basically of;

- housing
- bearing
- ball pin
- end cap
- boot support
- dust boot
- spring
- clamping ring

The joint connections can be made of many different designs. The most important of all, there are at least 6 ball joints in one model of any vehicle. Considering that many model changes, updates, redesign changes takes place, continues changes of the ball joints are needed continuously. Spare part manufacturers usually deals with many brands and different models of each brand at the same time. As a result of this, hundreds of different types and configurations arise. The high numbers of different ball joint designs is to be carried out with minimum time and effort as there is high pressures on the cost and ever increasing demands on urgent supplies coming from original equipment vehicle manufacturers. This demand can be met if the design is carried using parametric design of the ball joints based on the computer assisted 3D modeling technique.

1.2.1. Classification of Ball Joints

In Turkish standards TS 9444 and 5476, ball joints used in vehicles are classified according to their bearings, manufacturing type and connections.

1. According to bearings, ball joints are divided into two type (Fig. 1.3);

    - plastic bearing

- steel bearing

2. According to connection, ball joints are divided into two type (Fig. 1.4);

- castle-nut connection
- self-locking nut connection

3. According to housing connection , ball joints divided into two type (Fig. 1.5);

- outer thread
- inner thread

4. According to housing type, ball joints are divided into two type (Fig. 1.6);

- elliptical
- oval

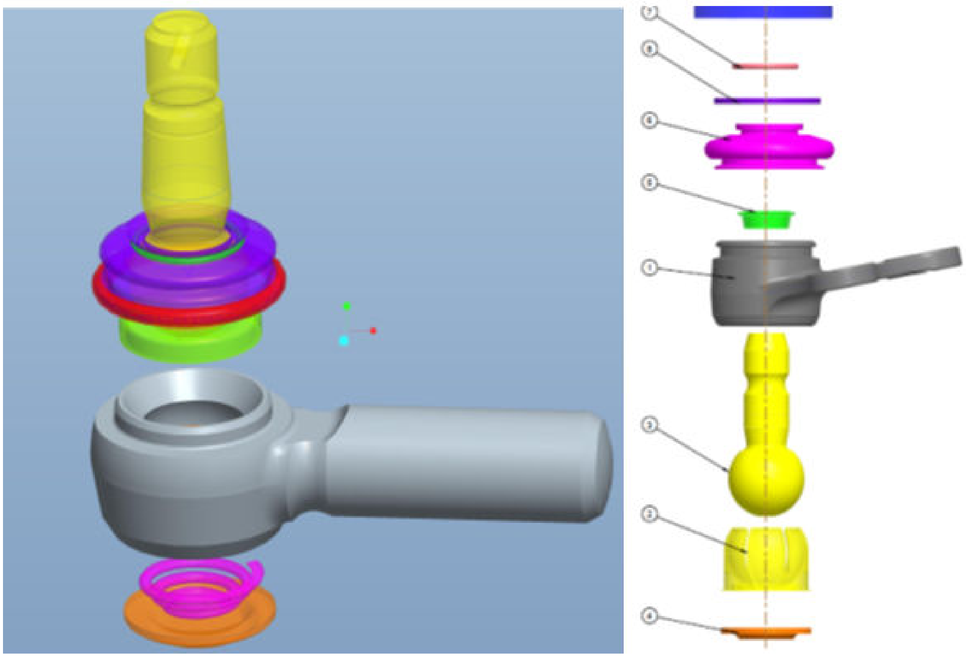


Figure 1.3. On the left, spring and steel bearing ball joint, on the right plastic and without spring ball joint

(a) (b)

Figure 1.4 (a) Castle nut connection (b) Self-locking nut connection

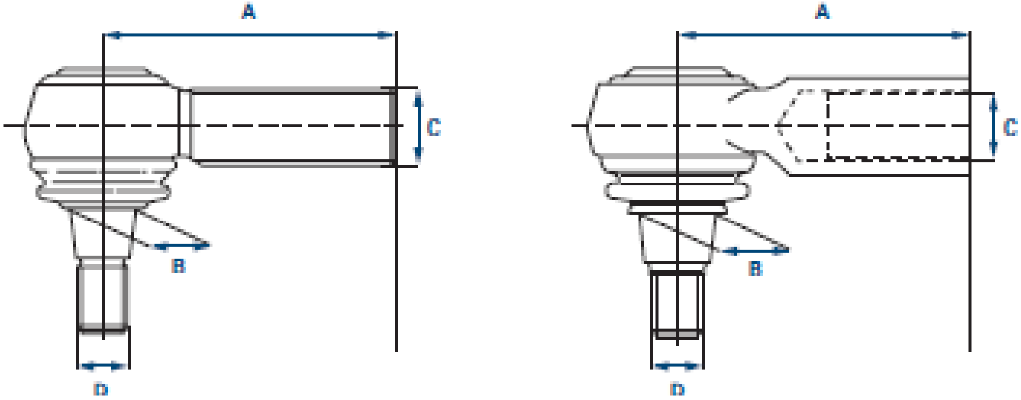
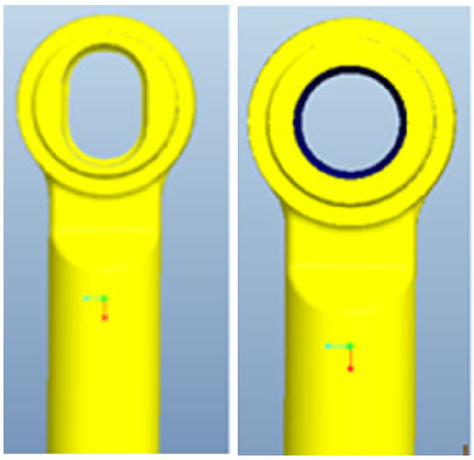


Figure 1.5. On the left, outer thread ball joint, on the right inner thread ball joint (www.zf.com)



(a) (b)

Figure 1.6 (a) Elliptical housing design (b) Oval housing design

1.2.2. Test Applied Ball Joints

Overall dimensions of ball joints and the related tests are defined in Turkish standards TS 9444 and TS 5476. The overall dimensions are subjected to change by auto manufacturers considering safety and design issues of each new vehicle design. Thus, some dimensions may be different from manufacturer to manufacturer. The tests provided below must be applied to the ball joint, the test results must be convenient or meet the standards, and be safe under dynamic loading conditions of vehicles.

List of tests applied to ball joints (Fig. 1.7) are listed down;

- Static load
- Fatigue strength
- Elasticity
- Static moment (torque)
- Pull out load
- Push out load
- Wearing
- Sealing

1.3. Cad Systems

CAD has evolved from simple drafting and analysis tools. Our present systems automate small and often isolated tasks within the overall engineering process. The CAD systems have been extended by programmatic means to assist the engineer in localized application areas and in the optimization of specialist part objects. To go further, we must take a holistic view of the design. To create a design system, the design process should be modeled. Our design systems should be flexible, and not constrain the designer to work in an unnatural manner. To automate design or to increase the role of CAD systems we must increase the level of natural interaction and intelligence. The CAD system should play the part of a consultant, a valued team member, making decisions based on the design constraints, interacting with the free flow of the design process, suggesting alternative strategies and assisting in an optimized or compromised design solution.

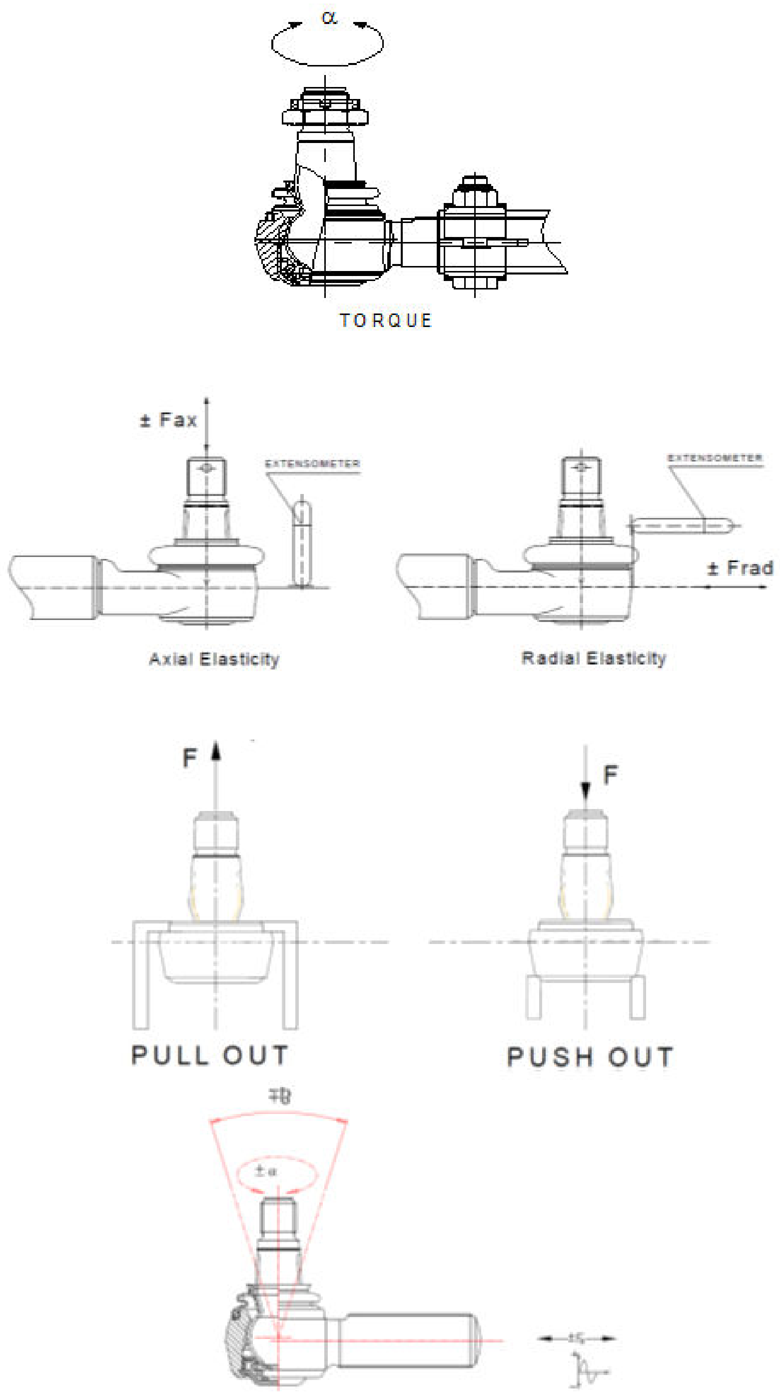


Figure 1.7. Tests applied on ball joints (TS 9444&TS 5476)

The system should allow for quicker design solutions, giving time to the human team member to do what they do best, be creative and search for new scenarios. The system should give back the time to be an engineer. A true CAD system should be able to draw from a company's natural knowledge base, the accumulated experience of the workforce. The proposed system will demonstrate the ability to utilize knowledge from various disciplines at the design stage and present this in an intuitive working environment. It is interesting to note that Quinn suggests that, "In the post-industrial era, the success of a corporation lies more in its intellectual and systems capabilities than in its physical assets. The capability to manage human intellect, and convert it into useful products and services is fast becoming the critical executive skill of this age" (Chapman & Pinfold, 1999).

The traditional way to create a model was to take a functional specification and give it to a design engineer who would draw the model in a CAD system using their experience to interpret the specification. As happens all too often within industry design models are duplicated, not due to integration issues, but due to the fact that the different engineering disciplines have a different yet valid view of the world. The analyst normally needs to remodel the initial design. This will either be done by editing the designer's CAD data or by creating a completely new model within the analysis software. Progressive CAD and analysis software companies have recognized the need for integration and have either integrated with other products or have modeling and analysis software within one system environment. This, however, does not go far enough and may well lead to rework. A design system must have within it the ability to represent a single product model as various alternative engineering views. There is a great deal of confusion in this area. As CAD systems create a geometric model and through data transfer or direct integration pass the model to other systems, the CAD vendors suggest that the design/analysis modeling route is automated. From understanding the design/analysis process we can see that there is a stage hardly discussed that of simplification or model transformation. This step is critical for the engineering process and often creates extra models that have no associatively (Chapman & Pinfold, 1999).

The CAD systems available in the market all have different modeling and functional capabilities. The commercial brands of domain CAD systems in the market are PTC Pro/Engineer Wildfire (now Creo Parametric), CATIA, Unigraphics (now Siemens NX), SolidWorks, Inventor, Rhino. A CAD system must have advantages about easy design of user interface, ease of programming, ease of modeling and assembly functions, applicability for CAE and CAM for developing parametric system efficiently.

Pro/Engineer, CATIA, Unigraphics and Rhino are compared in Table 1.1 for advantages of CAD system's functions as bad, not bad, good and very good.

Table 1.1. Comparison of CAD systems for efficient parametric system design

Function	**Pro/E**	**CATIA**	**Unigraphics**	**Rhino**
Ease of design interface	good	bad	bad	very good
Ease of programming	very good	bad	bad	very good
Ease of modeling and assembly functions	very good	very good	very good	very good
Applicability for CAE and CAM	very good	very good	very good	Very good

Pro/Engineer has a module called as Pro/Program to develop all functions needed for parametric design. The disadvantage of Pro/Program is that the user interface is not visual.

CATIA and Unigraphics need a high level programming knowledge like C or visual basic to develop user interface and parametric programming.

Although Rhino has excellent visual interface and programming properties with a package programming called as Grasshopper, the system can be applicable to CAE and CAM systems with an additional program called as RhinoCAM.

1.4. Aims of the Study

There are at least 3 ball joints at each steered wheel. This means that at least six ball joints are used in any conventional vehicle (see Fig. 1.8). Considering that many model changes, updates, redesign changes takes place, continues changes of the ball joints are needed continuously. This is because the weight of vehicles and their load carrying capacities change from vehicle to vehicle and even in a new model of a previous one. Therefore, original equipment manufacturers (OEMs), and other spare part manufacturers usually deals with designing and manufacturing of ball joints for many brands and different models of each brand. As a result of this, hundreds of different types and configurations arise. The high numbers of different ball joint designs is to be carried out with minimum time and effort as there is high pressures on the cost and ever increasing demands on urgent supplies coming from original equipment vehicle manufacturers. This demand can be met more efficiently if the design is carried using parametric design of the ball joints based on the computer assisted 3D modeling technique.

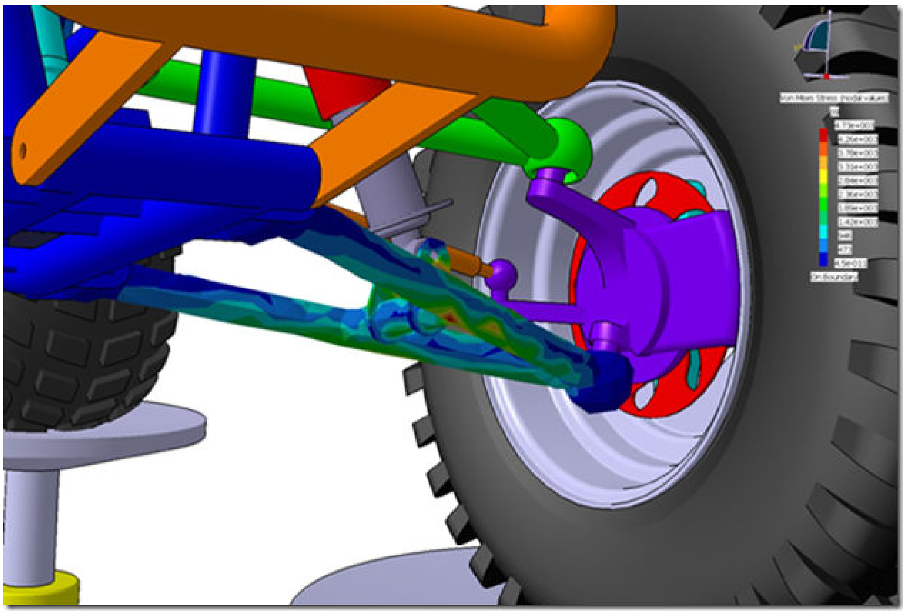


Figure 1.8. Ball joints used in a steering wheel
(http://www.mscsoftware.com/industry/motorsports)

Any work considering to develop a design platform for Parametric design of ball joints should consider designing of a new tie rod, redesigning of an existing one and generating solutions graphically in three dimensions (3D) using initial design parameters such as loads and moments on tie rod, size, tolerances material standards etc.

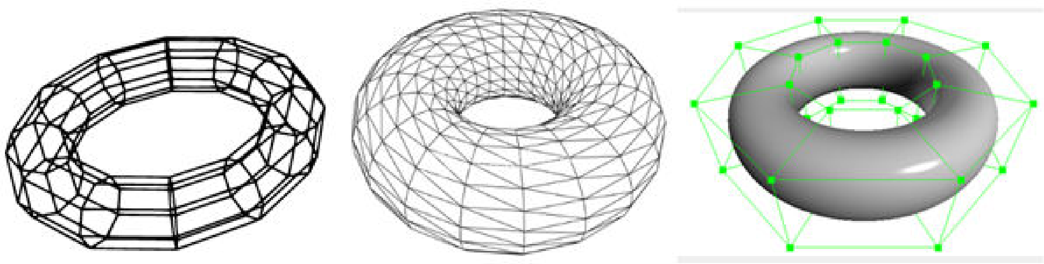
Hence, the aim of the study is to develop a Computer Aided Parametric Design platform using commercially available CAD systems in a programmable environment and using Knowledge-Based Engineering which can design and redesign all the new and existing parts of ball joints parametrically including ball pin, housing, clamping ring wire, clamping ring sheet, dust boot, end cap, spring, bearings and the tie rod. And it will also create 3D model and 2D layout of the all parts of the ball joint and parts assembly using a commercial CAD system. Briefly it proposes a new approach to parametric design system which supports not only parts but also assembly using a commercial CAD system and a new developed software.

The results of this study will not only allow decreasing the design cost and time of steering tie rod, it will also provide to collect knowledge of experienced designers.

2. PREVIOUS STUDIES

2.1. Evolution of CAD Modeling

The first methods and techniques were put into practice during the 1960s and included basic 2D primitives, as well as new entities like splines. The work of Bezier and De Casteljau goes back to this period. This was extended to 3D wireframes (Fig. 2.1a) and surfaces patches. New graphical methods are to be associated to the name of Sutherland and to the year 1963, in which his thesis was published. Polygonal meshes (Fig. 2.1b) were used at the end of the 1960s; soon there were techniques to visualize them by such methods as what is now currently known as flat-shading (Bouknight, 1970) or, better, Gouraud shading (1971) or, even better, Phong shading (1975). Free-form or sculptured surfaces were developed extensively during the 1970s. The most advanced method currently used, nurbs (non-uniform rational b-splines seen in Fig 2.1c) can be traced back to an article by A.R. Forrest in 1980. AutoCad has incorporated it, in 1995s (15 years after it was introduced), through an extra module (AutoSurf) that came together with version 13.



(a) (b) (c)

Figure 2.1.(a). Wireframe (b) Polygon mesh (c) Patchwork bicubic patches such as Bspline or NURBS (Monedero, 2000)

Solid modeling using a primitive form of CSG (Constructive Solid Geometry) was also born at the beginning of the 1960s (at the MAGI labs in USA), and evolved rather slowly until some complete products appeared in Europe and USA at the beginning of the 1970s. The first important commercial packages, like Romulus, commercialized by Evans and Sutherland in 1980, appeared at the end of the 1970s. A very much quoted article by Requicha that summarized the state of the art at that time, with five principal systems quoted was also published in 1980. At the present time, most systems currently used combine two systems, B-Reps and CSG or use B-Reps as a shell that allows multiple representations and favors the transit from one to the other. This is the case with ACIS (Alan, Charles and Ian System) used by AutoCad since version 13 after dropping AME, a CSG system that did not work as it should (Monedero, 2000).

2.2. Representation of CAD Models

Spatial database supports the applications that data types are characterized by spatial properties. Spatial attributes describe a space object with three factors: location, shape, size. These elements are suitable for graphic representation more than by the numeric value, string representation. The researches and developments in this area began in the 1990s (Tuan, 2013). Firstly, what is 3D GIS (Geography Information Systems). This type of system should be able to model, represent, manage, manipulate, analyze and support decisions based upon information associated with three-dimensional phenomena (Rahman, Zlatanova, & Pliouk, 2000).

Four main types to classify the models: 3D objects are represented by its boundaries, by voxel elements, by a combination of the 3D basic block and by a combination of the above methods (Tuan, 2013).

2.3. Evolution of Parametric Design Techniques

Besides the above-mentioned pioneering work of Ivan Sutherland, Hillyard and Braid proposed a system around 1978 that allowed the specification of geometric constraints between part co-ordinates in such a way that possible variations remain restricted to a range given by some particular tolerances. This proposal was not developed in the sense that could be expected from our present point-of-view. Gossardand Light mention this work as a basis for their own, which can be quoted as the primary reference for what can be called parametric design in a more mature sense. The work of Gossard and Light that will commented below as a basis for what is called variational geometry or variational design, was a major step as it provided geometrical representations with new mathematical and geometrical tools that opened the way to the generalization of a model.

Around the end of the 1980s, when the main techniques of geometrical modeling, free-form surfaces and solid modeling were already assimilated, there was a growing sense that modeling techniques should advance in the direction of an increasing interactivity and ability to modify a model after it had been sketched. There were a number of important articles and books already published and, also, a few articles by researchers directly involved in the development of this field that attempted to resume the state-of-the-art. It is clear that there are still, at the present time, two big groups, one that is becoming obsolete and the other that attracts a growing number of researchers:

1. What we can call, as Roller does, variants programming or static generation of alternative models by means of a programming procedure. These systems can rely on current internal representations of models.
2. Graphic generation or interactive methods by means of more elaborated systems that allow the modification of dimension and constraints after the model has been created. These systems imply a modification or an extension of the internal representation of the model.

The main disadvantage of the first group is that it cannot do what the second group does, that is, to change some of the characteristics of a model in an interactive way. On the other hand, it is a mode of work that can adapt to current CAD programs if the user has some knowledge of simple programming techniques. The main disadvantage of the second group is that we will have to wait a few years until a consistent parametric modeler, based on some of the different alternatives still under research enumerated below, is integrated in some of the programs currently used by CAD/CAM program users (Monedero, 2000).

It is now almost fifty years after Sketchpad and computers have replaced the drawing boards they once imitated. Many new ways of generating parametric models have been developed: from history-based modelers (CATIA, SolidWorks, and Pro/Engineer), to visual scripts (Grasshopper, Generative Components, and Houdini) and textual programming environments (the scripting interfaces to most CAD programs). The commonality of all these parametric modeling environments is the ability for designers to modify parameters and relationships that trigger the transformation of related model parts. This is now a popular way to create and modify digital models. Fifty years since Sketchpad, Robert Aish and Robert Woodbury (2005) say the hope still remains that parametric modeling will "reduce the time and effort required for change and reuse" (Davis, 2013).

A time when computers were becoming affordable enough for some people to own a *personal* computer, AutoCAD was released and quickly rose to dominate the fledgling computer-aided design industry (Weisberg, 2008). Gone were the curves, the artificial intelligence, and the self-replicating geometries. It was replaced in AutoCAD with commands enabling the designer to explicitly draft two-dimensional lines on screen using a keyboard rather than a pen. Eighteen versions later, in AutoCAD 2010, parametric functionality was introduced (forty-three years after Sketchpad) and pronounced in the press release, "a groundbreaking new capability". Sometimes it takes a while to realize the impact concepts like parametric design will have on practice.

AutoCAD2010's ground breaking new parametric modeling features were present in software decades ago. In 1985, the former mathematics professor Samuel Geisberg founded Parametric Technology Corporation. They shipped what would become the first commercially successful parametric software, Pro/ENGINEER, in 1988. Like with Sketchpad, users could associate parts of the Pro/ENGINEER geometry together using various parametric equations. Unlike Sketchpad, the geometry was three-dimensional rather than two-dimensional and changes could propagate over many different drawings created by many different users. During an interview with *Industry Week* in 1993, Geisberg succinctly expressed the original motivations of Pro/ENGINEER and captured, to a large extent, the motivations of parametric modeling: The goal is to create a system that would be flexible enough to encourage the engineer to easily consider a variety of designs. And the cost of making design changes ought to be as close to zero as possible. In addition, the traditional CAD/CAM software of the time unrealistically restricted low-cost changes to only the very front end of the design-engineering process.

Geisberg makes two salient points. The first is that parametric modeling should enable designers to explore “a variety of designs”. This is made possible in Pro/ENGINEER both through the manipulation of parameters and through the manipulation of the model’s underlying relationships. His second point is that parametric models allow choices to be made later in the design process (Davis, 2013).

2.4. Constraints

A fundamental problem in CAD is how to make explicit some intuitive knowledge we have about something in such a way that a machine can interpret and treat it in an automatic way. This problem reveals its magnitude as soon as we try to formulate what is comfortably referred to as ‘‘common sense’’. From an architectural point-of-view, this is like knowing that floors ‘‘shall always be’’ horizontal or that windows ‘‘belong to’’ a wall and trying to formulate this knowledge in such a way that a machine could not violate such an obvious rule. This is dealt with by means of constraints. Constraints appeared in CAD as early as 1963, in the pioneer work of Sutherland. As it happens with the very notion of parametric design, the notion of constraint is present, in a basic way, in any CAD system. A polyline, for example, can be understood as a collection of curves with vertices constrained to remain attached. But, in general, the notion of constraint implies a model with an extended database. A constraint is a relation that limits the behavior of an entity or a group of entities. Examples of constraints are: a group of lines constrained to be parallel or perpendicular or collinear, a line constrained to be tangent to an arc, two cylinders constrained to be concentric, a dimension constrained to be less than a particular magnitude or equal to a multiple of a particular magnitude (Monedero, 2000).

A geometric constraint network consists of geometry, constraints (e.g. parallel constraints, coincident constraints, tangent constraints, and perpendicular constraints), dimensions (e.g. linear dimensions, angular dimensions, and radial dimensions), and engineering relations that can be explicitly expressed as numerical formulations to fulfill design intent as illustrated in Fig. 2.2

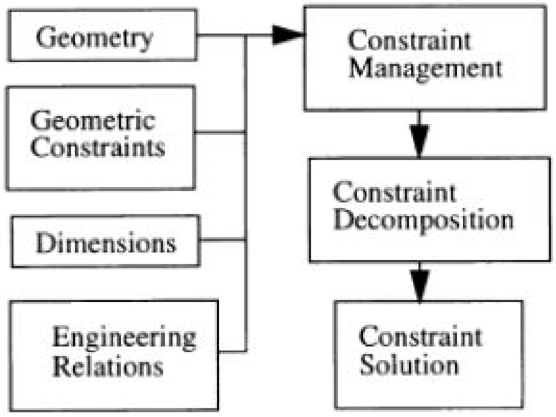


Figure 2.2. Overall Constraint diagram (Chung, Hwang, Wu, & Jiang, 2000)

There are many attempts in the literature to provide a powerful yet efficient method for solving geometric constraint problem. Ge and others (1999) divide geometric constraints into the symbolic approach, propagation approach, graph analysis approach, numerical approach. According to Essert-Villard et al (2000) since I.E. Sutherland's Sketchpad, various ways of solving constraints have been considered. Two main classes can be distinguished: numerical approaches and formal methods. Many resolution methods have been proposed for solving systems of geometric constraints; Li et al (2002) distinguish three major approaches for solving declarative constraint systems: the numerical, symbolic and synthetic approach. Ait-Aoudia et al (2010) classify the resolution methods in four broad categories: symbolic, numerical, rule-oriented and graph-constructive solvers.

In the numerical approach, the geometric constraints are translated into a system of equations. The constraints on simple geometry such as distance, angle, incidence can be represented by polynomials. Therefore it can be restricted to algebraic equations and various numerical techniques are used to solve them (Ge, Chou, & Gao, 1999).

In symbolic methods, the constraints are translated into a system of equations. Methods such as Gröbner bases or elimination with resultants are applied to find symbolic expressions for the solutions. These methods are ‘‘extremely” time consuming. They are typically exponential in time and space. They can be used only for small systems (Ait-Aoudia & Foufou, 2010).

Graph-constructive solvers or graph analysis approach are stemming from graph theory. They are based on the analysis of the structure of the constraint graph. The graph constructive approach provides means for developing sound and efficient algorithms (Ait-Aoudia & Foufou, 2010).

The propagation approach solves the constraints system by deriving unknown variables or geometric objects from already known one using a set of predefining rules. Usually propagation techniques are implemented with expert systems or logic programming techniques (Ge, Chou, & Gao, 1999).

2.5. Knowledge-Based Engineering (KBE)

Knowledge Based Engineering (KBE) has been identified as a promising method to enable design automation. KBE defines a wide range of methods and processes and can be described in several ways, depending on the application focus. In the literature, various definitions can be found that try to highlight the multiple sides of KBE. Chapman and Pinfold (2001) refers to KBE as ‘‘an engineering method that represents a merging of object oriented programming (OOP), artificial intelligence (AI) techniques and computer-aided design technologies, giving benefit to customized or variant design automation solutions’’. According to Blount et al. (1995) KBE is a ‘‘true integrator throughout the Product Introduction Process (PIP) supporting the ideas of concurrent engineering’’. Furthermore, Verhagen et al. (2011) state that ‘‘one of the hallmarks of the KBE approach is to automate repetitive, non-creative design tasks’’, which can lead to ‘‘significant cost savings’’ and ‘‘free up time for creativity’’. Through KBE, geometric models are given the knowledge, i.e. rules and instructions, which controls the geometrical transformations required for design automation

A holistic product perspective by means of design reuse and automation is needed in order to effectively manage product complexity. In this field, KBE is believed to be a powerful tool. To accomplish automated design and reuse, the knowledge of the design process has to be captured, stored, and deployed. This process can be referred to as Knowledge-Based Engineering (KBE) (Amadori, Tarkian, Ölvander, & Krus, 2012).

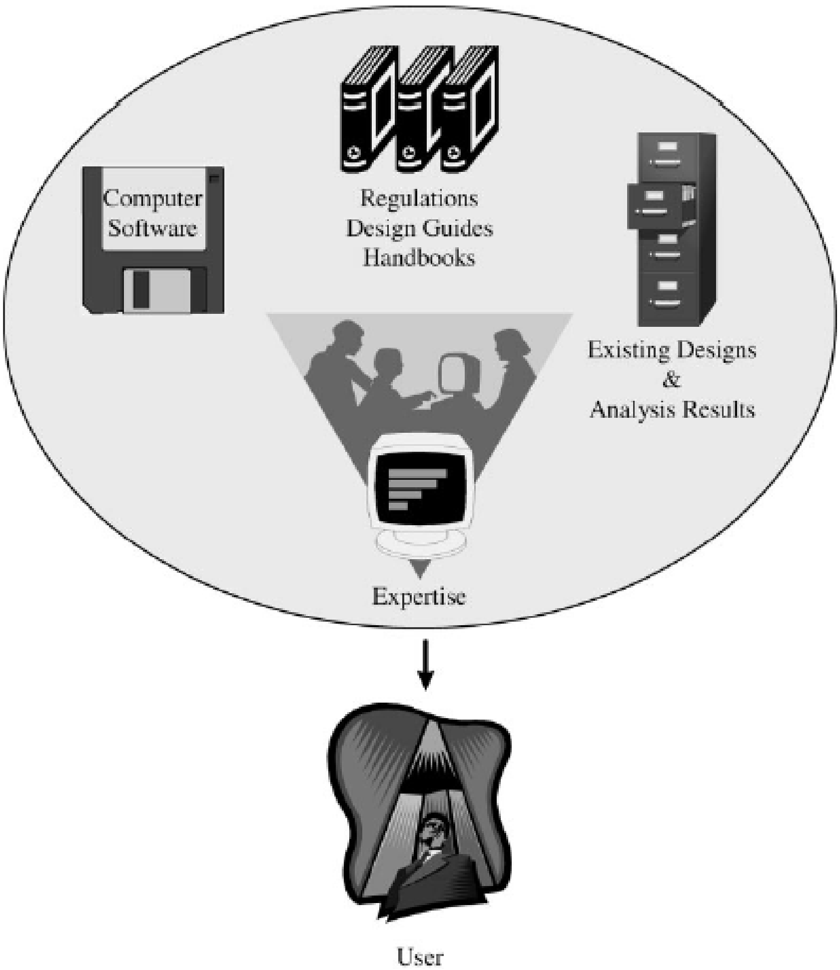


Figure 2.3. Corporate Knowledge Base (Chapman & Pinfold, 2001)

The KBE systems aim to capture product and process information in such a way as to allow businesses to model engineering design processes, and then use the model to automate all or part of the process. The emphasis is on providing, informational complete product representations, captured in a product model. The product model represents the engineering intent behind the product design, storing the how, why and what of a design. The product model is an internal computer representation of the product design process and can contain information on both the product and processes that go to create the part. Attributes can describe geometry, functional constraints, material type and processes such as the methods required to analyze, manufacture and cost a part. The KBE product model can also use information outside its product model environment such as databases and external company programs. The ultimate goal of the KBE system should be to capture the best design practices and engineering expertise into a corporate knowledge base (see Fig.2.3). The KBE methodology should provide an open framework for formally capturing and defining the process of design creation (Chapman & Pinfold, 2001).

2.6. Top-Down Design Approach

Although the top-down approach is a well-proven method, it has to be modified in order to enable topological automation of the geometry. In true design optimization of the geometry, the shape, placement and number of the CAD components should be modifiable. This in turn generates a new means of CAD modeling, referred to here as dynamic top-down modeling. When applying a dynamic top-down development process, the actual CAD models can be generated from pre-described High Level CAD templates (HLCt), see Fig. 2.4

The critical information on how the HLCt should be instantiated is stored in the knowledge base and used by the inference engine. The geometry model is divided into submodels that are linked to each other in a hierarchic relational structure. From an initial model, the user starts to identify the number of instances needed from each HLCt database through a user interface. Various components can be attached dynamically to the model and their shape altered by the inherited design variables. With reference to Fig. 2.5, the geometric groups represent subsets of the overall geometric model into which one or more HLCts are instantiated. Within dynamic top-down modeling, HLCts can either be placed in ‘‘geometric containers’’ in a part or in new parts within an assembly.

Topological parameterization requires reference geometries of both the HLCt and the surrounding context. The inference engine is then able to instantiate the selected HLCt into the CAD model and constrains it according to the predefined logic. In the simple example of Fig. 2.5, the references stored in the knowledge base are sought after by the inference engine following a logic resembling the pseudo code (Chapman & Pinfold, 1999).

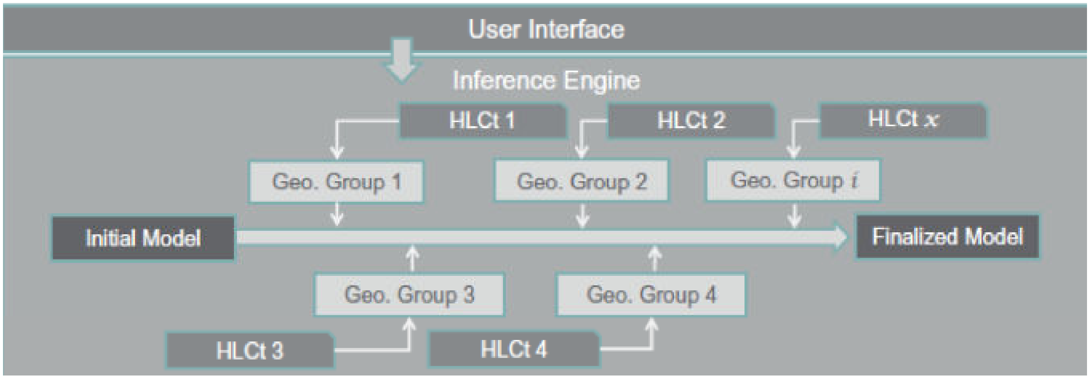


Figure 2.4. The geometric models are constructed by instantiating HLCts in the context defined in the inference engine (Chapman & Pinfold, 1999)

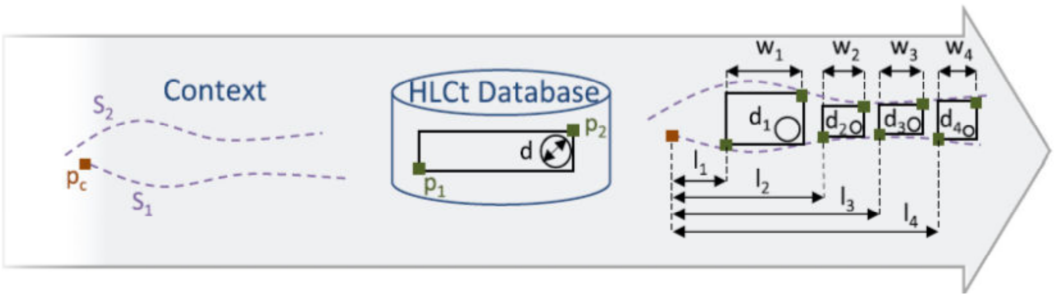


Figure 2.5. The topological instantiation process of a HLCt (Chapman & Pinfold, 1999)

2.7. Development Platform

Parameterized design making use of software not only demands model controlled by size parameters, but also have correlation among size parameters to keep them always relatively. In the physical design mode, the system allows the building of physical relationship between structural features, making the different characteristics of the entity associated, and the relationship created at this time becomes physical structure parameters relationship. In the physical assembly mode, the system also allows to create the parameter relationships between parts. In many 3D design software, the typical feature of Pro/E is parameterization, and the Pro/E software was selected through full consideration (Liu, 2011).

Top-down design is a product development process through which a design originates as a concept and gradually evolves into a complete product with individual parts and subassemblies modeled in Pro/ENGINEER. The designer treats the components as part of a system from the beginning, considering the interactions between components. The term “top-down design” refers to the method of placing critical information in a high-level location and then communicating that information to the lower levels of the product structure. As the design develops, more specific information becomes available and is incorporated into the design. By capturing the overall design information in one centralized location, it becomes easier to make significant design changes. Because the information is contained in one place—with all subcomponents looking to that location—if you change this information, the system updates all other components.

With the traditional assembly design approach, an engineer designs individual components independently of the assembly, using a manual approach to ensure that components fit properly and meet the design criteria. The designer places components in subassemblies and then brings those subassemblies together to develop the top-level assembly. Often, after creating the assemblies, a designer discovers that the models do not meet the design criteria. After detecting this problem, he or she must correct it by manually adjusting each model. As the assembly grows, detecting these inconsistencies and correcting them can consume a considerable amount of time. If significant changes occur that affect many components the designer must manually identify and modify each component to make the adjustment (www.ptc.com).

To successfully use the top-down design approach in Pro/ENGINEER, generally following steps are to be performed:

- Define the global product information in a layout.
- Generate an initial assembly structure. Create skeletons to define the product information in a 3-D form.
- Document design information using published geometry features.
- Communicate design information to the individual components using Copy Geometry features.
- Create model geometry using the central design information.
- Populate assemblies with solid models.
- Establish interchangeability to account for design variations.

As the design evolves and the designers are able to obtain more information about the design, they may need to further define the design intent, edit the skeletons, pass the critical data to other models and continue to populate the assembly. This is an iterative process—one in which the design becomes more detailed and specific throughout the project. You should, therefore, expect to perform the sequence of steps listed above more than once in order to complete the project.

Pro/PROGRAM is a simple automation tool provided with Pro/ENGINEER, which allows you to perform basic automation of design changes to parts and assemblies (www.ptc.com).

2.8. Studies about Parametric Design Using Computer Assisted 3D Modeling

2.8.1. Semi-automated Parametric Design of Gating Systems for Die-casting Die

The gating system is very critical to a die-casting die, but designing the gating system is an iterative process that can be very time-consuming and costly. A study was carried out to develop a design system that helps to realize automatic generation of the gating system's geometries by applying parametric design (Wu, Fuh, & Lee, 2007). Parameterized solid models of gating elements are pre-constructed and stored in the system database, then retrieved from the database after the desired parameters and locations are specified and finally joined to the die-casting part to form the die cavity using Boolean operations. This design approach was claimed to significantly reduce the time needed to construct the 3D geometries for such gating elements as overflow, gate, runner, sprue, etc., and it makes the modification of the existing design easier. A design evaluation module has been established to help predict the related parameters and verify the design. With such capability, die designers can incorporate their expertise into the design process at the early stage and make the initial gating system design more closely resemble their final design.

A prototype system for designing the gating system was developed on the commercial Unigraphics CAD system. Unigraphics was chosen as a platform because it allows the user to build a customized system using UG/open API and supports the use of user-defined features due to its parametric nature and the ability to extract design information from the feature.

As shown in Fig. 2.6, the proposed parametric design system is composed of system databases, design modules and design evaluation module. The system databases include a gating model database that stocks the parametric solid models of gating elements, a process database that contains the data of material and machine, and a knowledge base containing the rules for design evaluation. The design modules offer the tools for data initialization, flow path analysis, layout design and gating elements design, etc. The design and redesign tools are integrated in the design modules by the dialogue-based user interface. The design evaluation module, which is supported by the knowledge base and the process database, implements design evaluation and provides guidance for the designer during the design process.

As a conclusion, a prototype parametric system for designing the gating system of die-casting dies developed (Fig. 2.7). The proposed system is able to reduce the geometry construction time of gating elements significantly. The system also integrates the know-how on gating design so that it can predict and evaluate the parameters of a gating element to well meet the design requirements. With the ability to provide guidance for the designer in the streamlined procedure, the system makes the first-trial design of a gating system closer to the final design so that the time for modification and redesign can be shortened. However the prototype system needs further enhancement since gating system design varies from case to case.

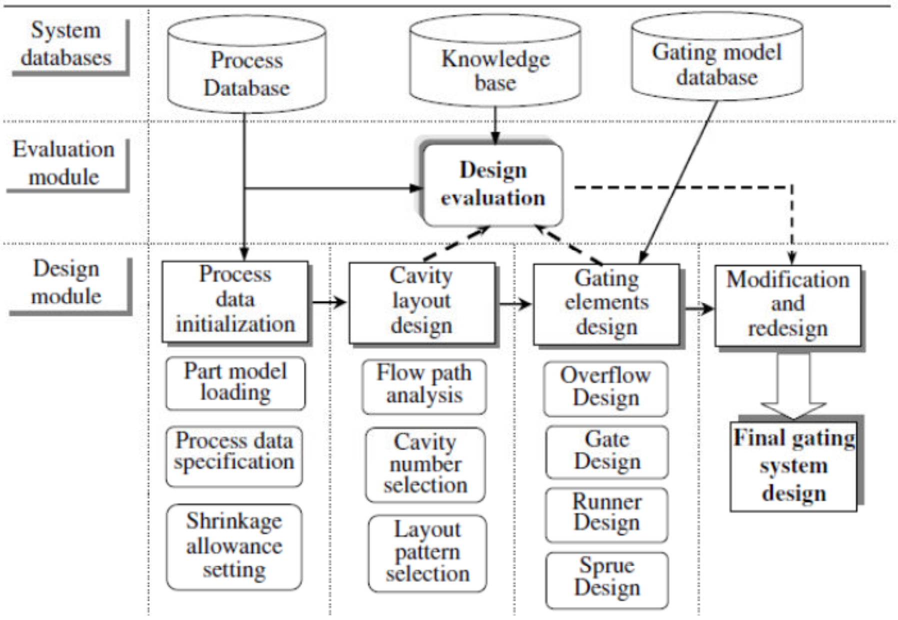


Figure 2.6. The overall framework of the design system (Wu, Fuh, & Lee, 2007)

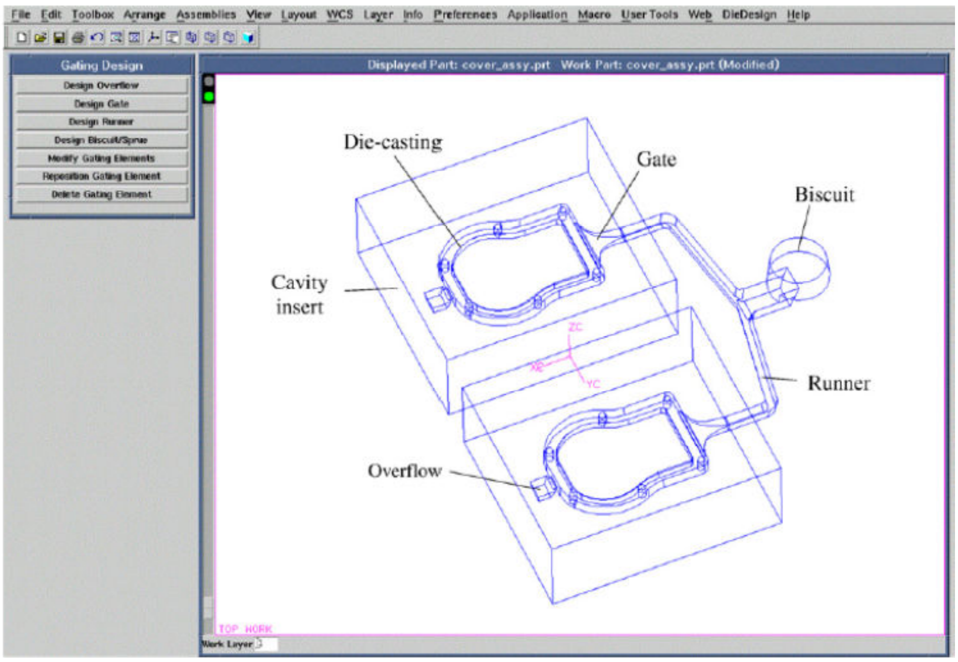


Figure 2.7. The complete gating system (Wu, Fuh, & Lee, 2007)

2.8.2. Parametric Design Implementation in Helicopter Gearbox Design

A study that was carried out by Eurocopter and Aix Marseille University (2011) describes a parametric design methodology implemented to reduce gearbox design phase. A helicopter gearbox is a complex mechanical system, in which parts are heavily loaded in order to save weight but that have to run for thousands of hours without failure. Such a system is strongly optimized therefore the smallest change in specifications or in geometrical environments can drastically jeopardize the scheduling. In order to speed up the gearbox design process and to strengthen its robustness regarding specifications variability, a parametric design methodology has been developed collaboration between the laboratories and Eurocopter (Mermoz, Lineras, & Bernard, 2011).

The role of the main gearbox (MGB) is to transform the mechanical power generated by the engines (from 0.3 to 4 MW depending on the size of the helicopter). The engine rotational speed (from 6000 RPM to 23,000 RPM) is reduced according the rotor needs to lower values (from 400 RPM to 160 RPM). A helicopter gearbox is a complex system based on several combinations of power gear trains with specific power bearings, installed into complex housing (Fig.2.8). The configuration of the gear train is the result of geometrical constrain given by the helicopter architecture together with power gears sizing. The role of the gearboxes on helicopter is not limited to mechanical power transformation and distribution. Indeed, gearboxes also provide a geometrical and mechanical links between many subsystems of the helicopter dynamic system such as engines, servo actuators, tail drive shafts and rotors systems.

The authors of the work chose to work with CATIA V5 to take advantages of powerful 3D representations together with its native parametric modeling capabilities. Another choice was to manage equations that drive the design parameters out of CATIA from Excel files. The target was to simplify the work of designers during the building of the model and also to ensure the protections of the Eurocopter knowledge that are linked with the choice of the parameters. During the development phase, the CAD model are indeed used for data exchanges between Eurocopter, engineering subcontractors, partners or manufacturing suppliers, so the model must not integrate specific know how.

Figure 2.8. Example of Eurocopter main gearbox CAD model (Mermoz, Lineras, & Bernard, 2011)

It's difficult to fix a complete methodology down to the lowest level of CAD design because the topology of the MGB (main gear box) housing is always different. An example of gearbox housing can be seen in Fig. 2.9.

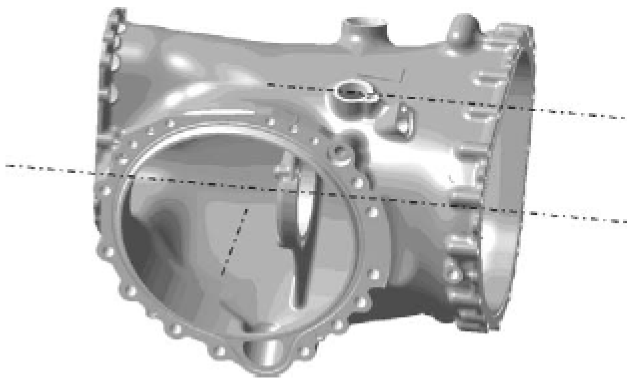


Figure 2.9. Gearbox housing (Mermoz, Lineras, & Bernard, 2011)

The interfaces and the gear trains axes are driven by external publications, but all other features which need to define bearing locations, fixation holes or wall thickness are managed locally by the designers. The parametric modeling of complex housings is still a field on which the design methodology has to be improved in order to easily generate a full 3D solid model.

The development of this parametric approach has brought many benefits in the lead-time reduction and in the quality of Eurocopter gearboxes conceptual design. Even if there is a huge benefit for MGB concept generations, a design choice is also highly linked with concept evaluations that will be mainly based on deflection and stress analyses. Therefore one of the next challenges for Eurocopter is the propagation of the design parameters not only in the CAD model, but also in finite elements models (FEM) (Fig.2.10).

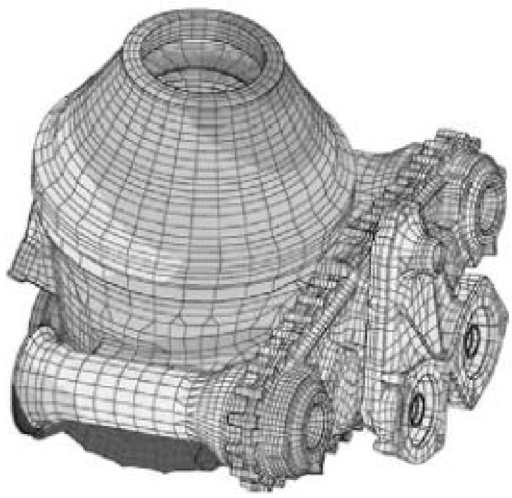


Figure 2.10. Example of Gearbox FEM analysis (Mermoz, Lineras, & Bernard, 2011)

Mastering a parametric approach that links the CAD model, the meshing, the loads and boundaries conditions are new challenges that can significantly contribute to mechanical design lead time reduction.

2.8.3. Computer Aided Parametric Design for 3D Tire Mold Production

A study presented by Chu et al (2006) used a parametric design system for 3D tire mold production. Tire grooves commonly used in the current industry are classified according to their modeling procedures, and the design parameters for each groove type are characterized (Fig 2.11). The result serves as a foundation for standardization of the tire mold design. The presented system simplifies the construction of 3D groove surfaces by reducing the number of interactive modeling operations. The resultant surface model is parameterized, and thus, allows for rapid creation of other grooves with simple design tables. In addition, a set of geometric algorithms is proposed that first detects undesired groove geometries arising in the design process, and then corrects them automatically. In this manner, 3D mold models are created with minimal user interactions. This work is implemented in an integrated CAD/CAM system for actual mold production. Test examples demonstrate that it provides an effective approach to reducing the time yet improving the quality of tire mold development (Chu, Song, & Luo, 2006).

This work has been implemented in a local company of tire mold manufacturing. First, international tire companies make new tire pattern designs with 2D CAD system (AutoCAD2002 in this case), and then send the pattern drawings in DWG format to the mold company via FTP and emails. The mold company manually checks the completeness of the data, including tire profiles, pitch arrangement, groove specifications, quality control instructions, and auxiliary information. Currently the 3D mold is modeled with the GSM module in CATIA V5 system. Surface model is adopted instead of solid model in this case, as the former facilitates the subsequent tool path planning. To demonstrate the feasibility of the design framework, a customized mold creation module has been developed in CATIA using Component Application Architecture (CAA), which allows an effective integration with the system. The proposed geometric algorithms are also implemented with CAAC++ API's and other programming resources (e.g. GUI's, memory management tools, and graphics) provided by CATIA V5 (Chu, Song, & Luo, 2006).

The authors of the work implemented a high-end CAD/CAM system for actual production. A commercial tire mold is used as a test example for demonstration of its practicality and effectiveness (Fig. 2.12). The results have indicated several advantages of introducing the parametric design system over the previous approach. First, the complexity of the mold design is hidden from the user. Modeling errors are reduced due to fewer user interactions resulted from the automatic groove construction by the design table. As a result, the quality of the mold design is enhanced. More importantly, the parameter values of the groove pattern in a first pitch can be utilized to accelerate the pattern creation in other pitches, saving a great amount of the time for the mold design.

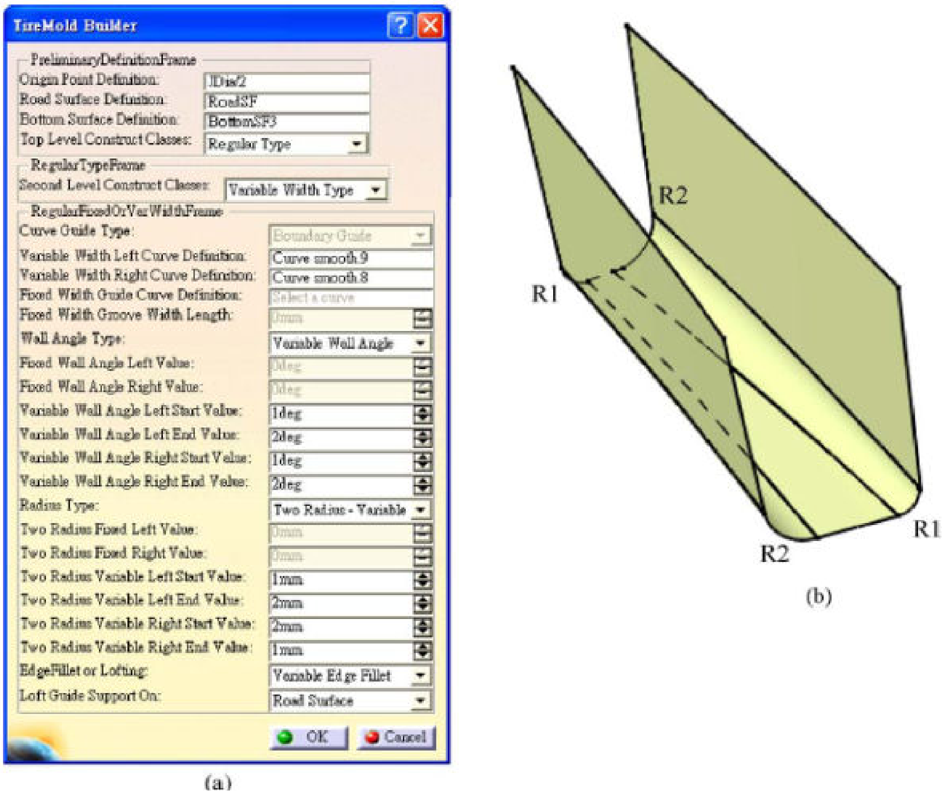


Figure 2.11. Groove design interface and result of groove design (Chu, Song, & Luo, 2006)

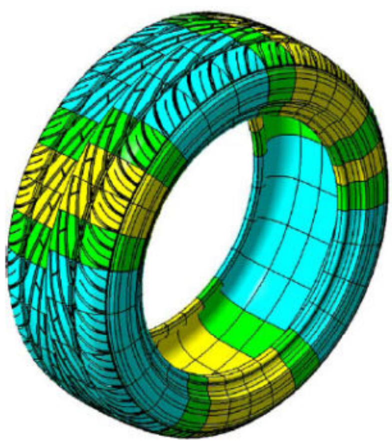


Figure 2.12. Complete 3D tire mold constructed with the proposed system (Chu, Song, & Luo, 2006)

The study concluded that it provided an effective approach to accelerating the tire mold development as well as the related new tire manufacturing. It is also claimed that theme's future research will be focused on parametric tool path planning with the mold models generated from the proposed system. In addition to that the authors claimed that an intelligent interface is also worth of studying for better integration between the tire design and them old modeling.

2.8.4. Parameterized Mockup Design of PDC (Polycrystalline Diamond Compact) Bits

With the development and spreading of computer-aided design (CAD) technology, it is the trend to use three dimensional design software for parameterized design. Based on Pro / E (Pro/ENGINEER) strong secondary development platform, with the matrix concave Polycrystalline Diamond Compact (PDC) bits commonly used in the coal industry as example, on the basis of systematically analyzing the correlation of structural features and main parameters of concave bits, the authors of the paper has established technology system of the related parameters for structural design of bits (Liu, 2011). Design model of PDC bits was set up by using the relationship of Pro / E, parameters and program modules. The design was carried out by using Ø65mm matrix PDC bits as real example. The results showed that the developed model can quickly and accurately draw three-dimensional and two-dimensional structural diagrams of bits, not only standardizing the designing flowchart, but also improving the quality and efficiency as well as reducing the error of drawings (Liu, 2011).

The cutting teeth of PDC bit is Polycrystalline Diamond Compact, and the structure of all PDC bits is similar, made up of PDC, bit crown, hydraulic system (water eye and waterway), and bit connecting part (Fig. 2.13). The crown shape can be divided into wing blade, inner concave, plane bottom and arc-shaped pillar, according to the strata performances, different crown shape was selected; according to the material of crown, the PDC bit can be divided into two kinds: matrix-body and steel-body. The crown part of matrix-body PDC bit was produced by sintering WC

(Tungsten Carbide) and solder, and it has many characteristics, such as structure variety, surface resistance to erosion, and excellent radius retention, but the cost is high, because the processing is complex. The crown part of steel-body PDC bit was produced by lathe, the cost is low, but it doesn't have above excellent characteristics. The matrix-body PDC bits having excellent performances were used widely in the field of mine and coal exploration (Liu, 2011).

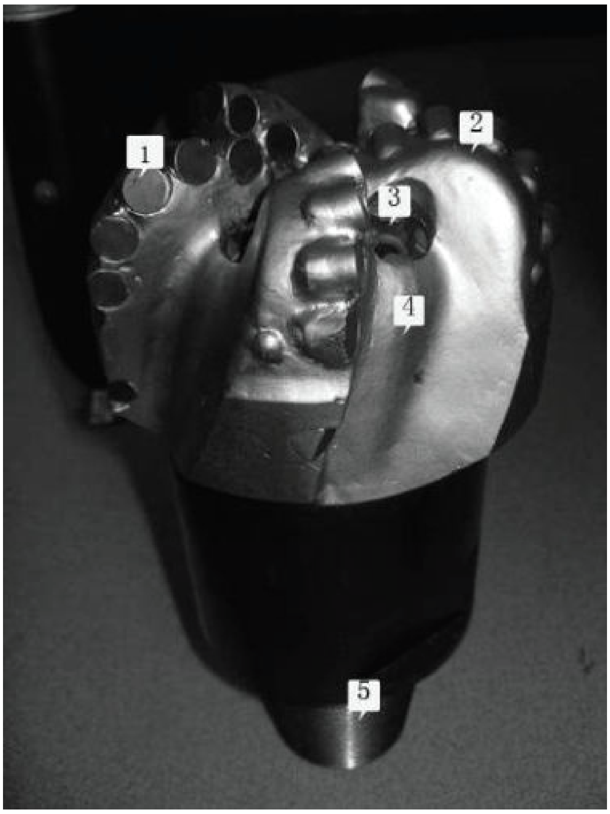


Figure 2.13. Structure of normal PDC bit (1-PDC 2-drilling bit crown 3-water eye 4-water way 5-drilling bit connecting part) (Liu, 2011)

The size parameter variation relationships were summarized about matrix-body concave PDC bit, and then these relationships were written in Pro/E software to complete drilling bit design. The 3D parameterized solid model of matrix-body PDC bit was run many times for the purpose of evaluating the design model (Fig. 2.14), and the Ø65mm matrix-body PDC bit was designed and manufactured according to the 3D parameterized solid model, which was claimed to satisfy user's demands. The results indicated in the paper claims that the design efficiency improve and the product development cycle was shortened by using 3D parameterized solid model of matrix-body PDC bit.

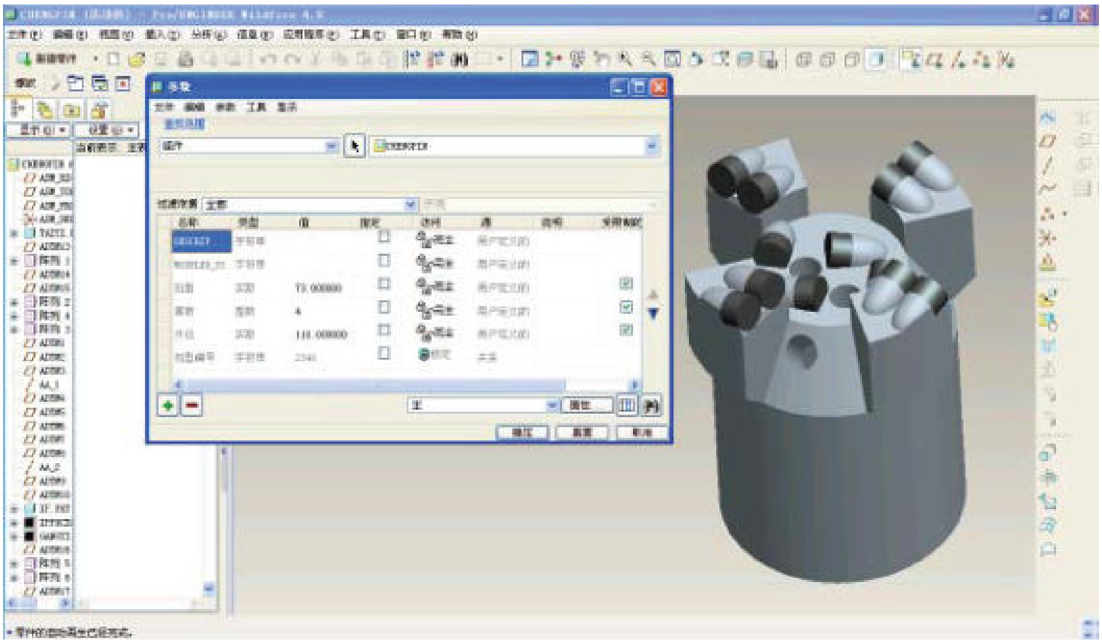


Figure 2.14. Interface of PDC bit 3D parameterized solid model in pro/e (Liu, 2011)

2.8.5. Case Based Parametric Design System for Test Turntable

Test turntable is a special and important equipment for inertia navigation system and inertia component's research. It is widely used in aerospace and aviation industries to examine and calibrate inertia navigation system or study the error models. Due to its particular functions and purposes, test turntables are mostly manufactured in a small volume with diverse product types. Conventionally, turntable design has always started with the sketch, which makes the whole design process iterative, time-consuming and costly. Moreover, the design work requires years of experience and a great deal of expertise, which is really lacking for novice or inexperienced designer. To change the current situation, an efficient and effective way needs to be proposed (Qiaosheng & Xi, 2011).

A study presented by Qiaosheng et al (2011) presents a case-based parametric design system for test turntable. The aim of this system is claimed to realize automatic and intelligent design of turntable. Utilizing the parametric design method, a knowledge-base composed of parameterized 3D product models is first constructed. All the corresponding design specifications or parameters are then saved in a data-base. Taking advantages of the CBR methodology, a most similar case can be easily retrieved from the pre-built design knowledge-base and case base when a new design task comes. Only with a few revisions, the modified design can mostly satisfy the new design requirements to fulfill a new design and can be further saved as a new case into the database. The design system is built as a module embedded in Pro/ENGINNER 3.0. It contains a turntable case base, a design knowledge-base, a CAD software and a graphic user interface. The system was put on a trial operation in a ship institute, then the result showed that it can enhance the design efficiency and reduce the design time and cost (Qiaosheng & Xi, 2011).

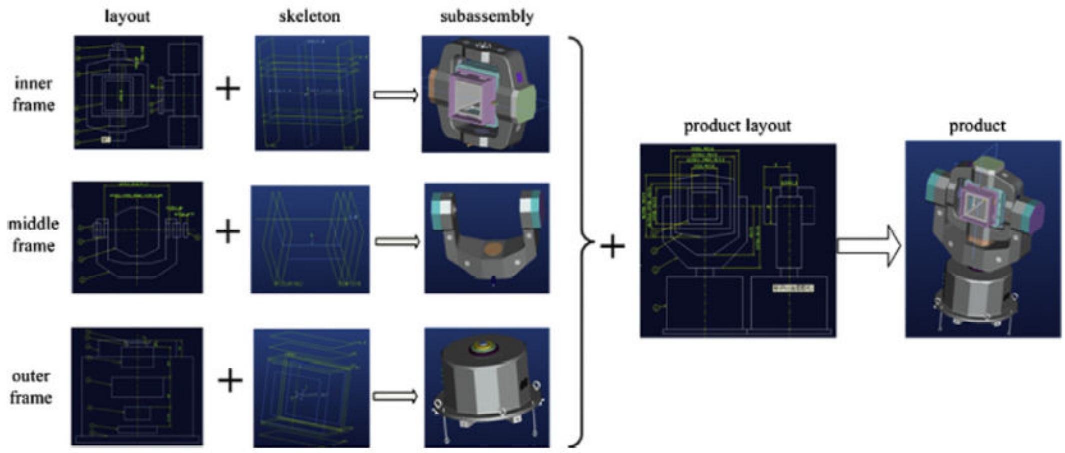


Figure 2.15. An example of top-down design (Qiaosheng & Xi, 2011)

Top–down design (Fig. 2.15) transfers the design data from layout to assembly structure and further to part level. Moreover, the design data also transfers between part and part. This method was claimed to protect the data- relevancy of the whole assembly structure, which makes the newly finished assembly highly qualify the layout of conceptual design. Based on top–down method, every part in a newly finished assembly can automatically change following the change of conceptual design rather than manually modify one by one. The CAD software – Pro/ENGINEER used in our expert system provides several tools to implement top–down design. After the creation of solid model, the whole model needs to be parameterized to achieve the goal of automatic generation. Parametric design deals with variable dimensions as control parameters and allows the designers to modify the existing designs by changing the key parameter values (Qiaosheng & Xi, 2011).

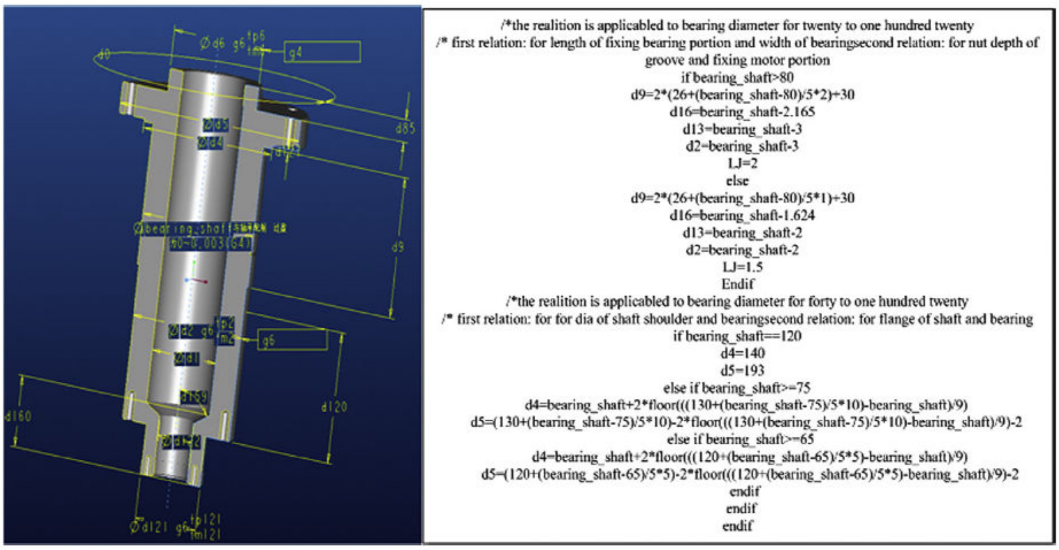


Figure 2.16. An example of part parameterization (Qiaosheng & Xi, 2011)

Briefly, the study presented a Case Based Reasoning (CBR) parametric system for designing test turntables. Parametric design method makes the design process automatic while CBR technology realizes the intelligent design (Fig. 2.16). Integrated both the parametric design and CBR technology, this system is able to speed the product design and reduce time and cost significantly. It was stated that young designers can easily handle the design task and reuse the knowledge or expertise conveniently. Moreover, it was claimed that a great deal of turntable design knowledge and expert experience has been saved in the knowledge base during model parameterization, which is significant in protecting the enterprise knowledge and expert experience. It was claimed that the time for redesigning an existing turntable is significantly reduced and the finished design is very closer to the final design. However, the authors concluded that the system still needs further enhancement such as improving the retrieve algorithm, enriching the case base and soon. It was also stated that there are more considerations to be taken to achieve a practical and accurate design system for test turntable (Qiaosheng & Xi, 2011).

2.8.6. Evolving Parametric Aircraft Models for Design Exploration and Optimization

Combining a design generation tool with an analysis software and an evolutionary algorithm provides a methodology for optimizing designs. This work combines NASA's parametric aircraft design tool (Open VSP) with a fluid dynamics solver (Open FOAM) to create and analyze aircraft. An evolutionary algorithm is then used to generate a range of aircraft that maximize lift and reduce drag while remaining within the framework of the original design. Approach in this work allows the designer to automatically optimize their chosen design and to generate models with improved aerodynamic efficiency. Different components on aircraft models are varied to highlight the ease and effectiveness of the parametric model optimization (Byrne, Cardiff, Brabazon, & O'Neill, 2014).

An important aspect of parametric design is that the user observes the effects caused by manipulating a variable in real time, allowing the user to treat the underlying algorithm as a black box. To do this, many parametric design systems, such as grasshopper, are implemented using a drag and drop interface, shown in Fig. 2.17. The user can then manipulate the input and evaluate the benefit of the component to the overall design. Parametric design tools have been introduced into main- stream design software. There is the Grasshopper parametric design tool plug-in for the Rhino modeling system (Byrne, Cardiff, Brabazon, & O'Neill, 2014).

The algorithm given in this work applied Cessna 182 and MIG-21 (see Fig. 2.18 and 2.19) At the end of the optimization, the lift is maximized and the drag is minimized. Experiment gained in this work shows that this approach could potentially be applied to any existing parametric design to generate optimized solutions, turning the computer into an active design tool in the conceptual design process.

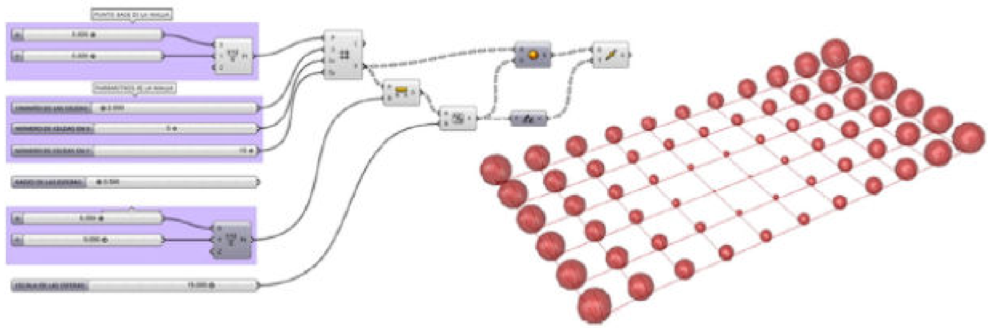


Figure 2.17. The GUI (graphical user interface) for the Grasshopper parametric design system (Byrne, Cardiff, Brabazon, & O'Neill, 2014)

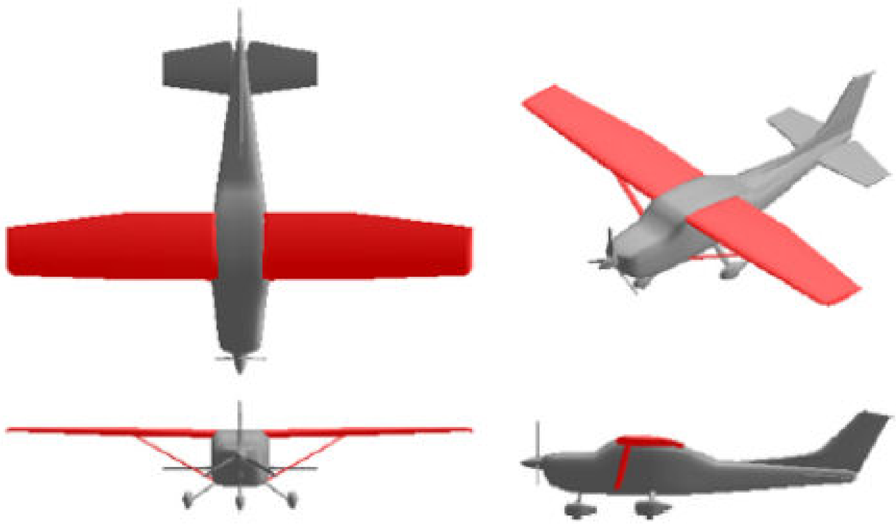


Figure 2.18. The Cessna Model (Byrne, Cardiff, Brabazon, & O'Neill, 2014)

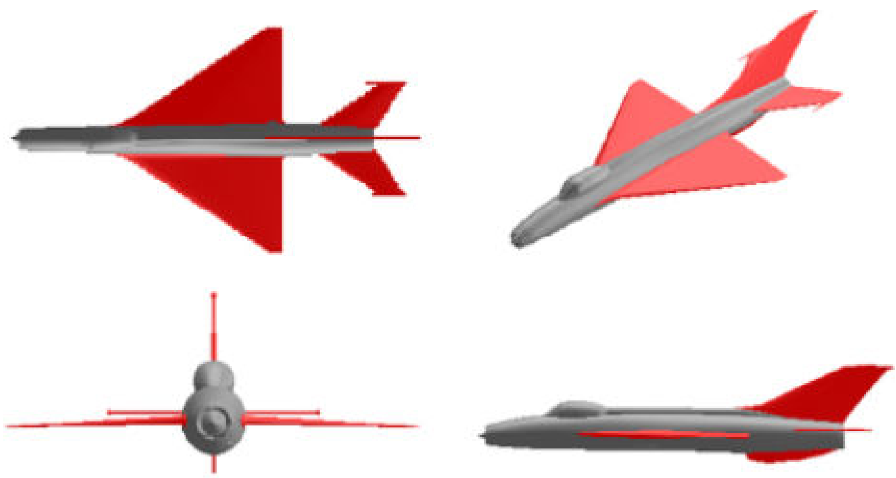


Figure 2.19. The MIG-21 Model (Byrne, Cardiff, Brabazon, & O'Neill, 2014)

2.9. Design of Ball Joints

Figure 2.20 shows most widely used hydraulic steering system used by Opel in the Vectra (1997). The oil pump is directly driven by the engine and constantly generates hydraulic power. The pressure oil required for steering boost is supplied direct to the steering valve (item 6) located in the pinion housing from vane pump 1 via the high pressure line 2 and the cooling circuit 3. From here, depending on the direction of rotation of the steering wheel and the corresponding counterforce on the wheels, distribution to the right or left cylinder line takes place (item 7 and 8). Both lead to the working cylinder which is integrated in the steering-gear housing 5. Ball joints (item 11) are used for the transmitting movement to wheels in desired direction (Reimpell, Stoll, & Betzler, 2002).

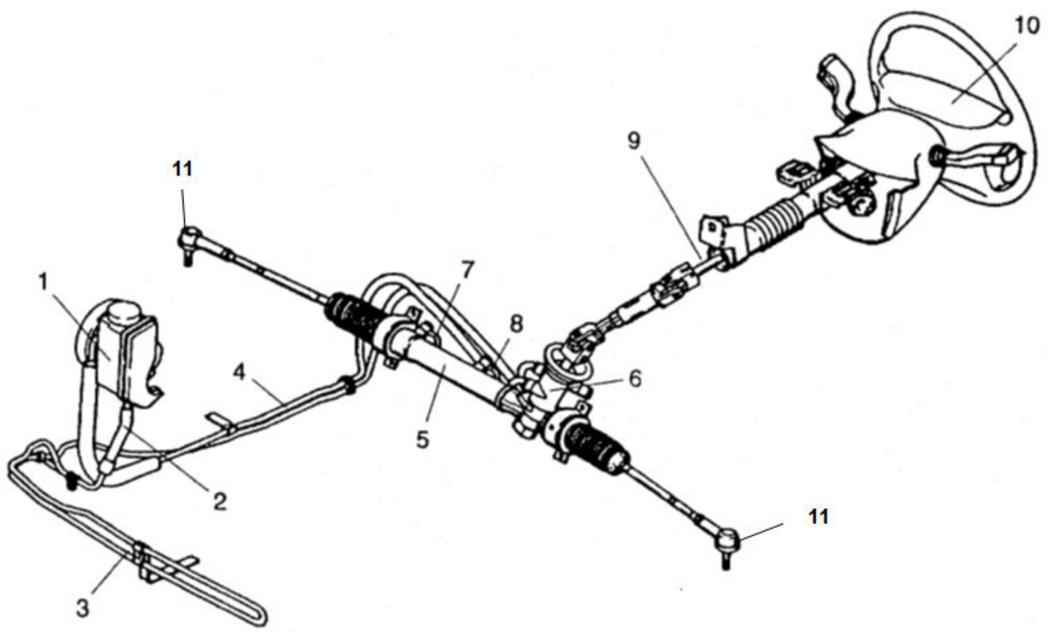


Figure 2.20. Hydraulic power steering system of Opel Vectra (1997) (Reimpell, Stoll, & Betzler, 2002)

All steering layouts are comprised of rods and arm joined together by ball joints. The ball joints enable track rods, drag-link rods and relay rods to swivel in both the horizontal and vertical planes relative to steering arms to which they are attached. Most ball joints are designed to tilt from the perpendicular through an inclined angle of up to 20° for the axle beam type front suspension, and as much as 30° in certain independent front suspension steering system (see Figure 2.21).

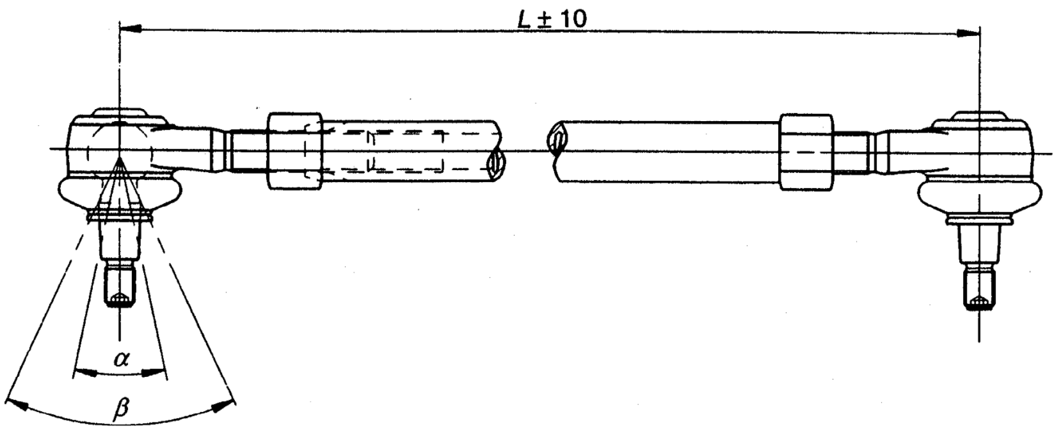


Figure 2.21. Tilt angles of ball joints (Lemförder, 2002)

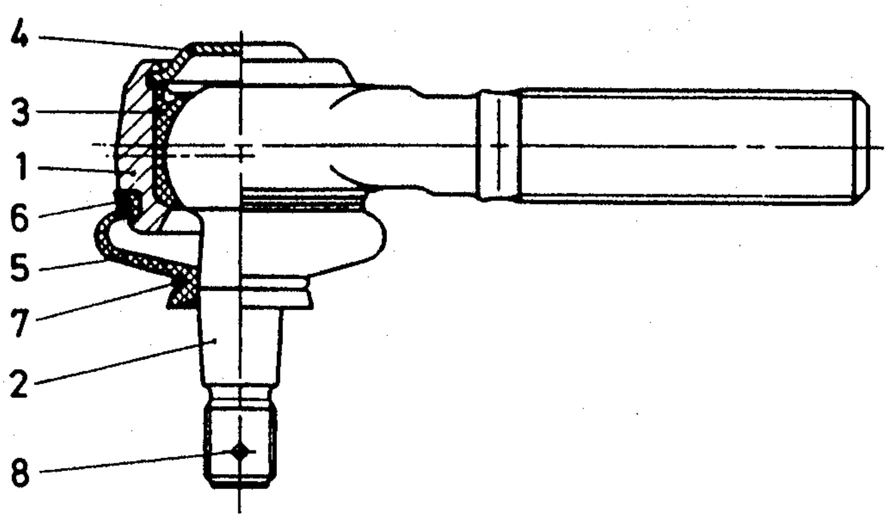
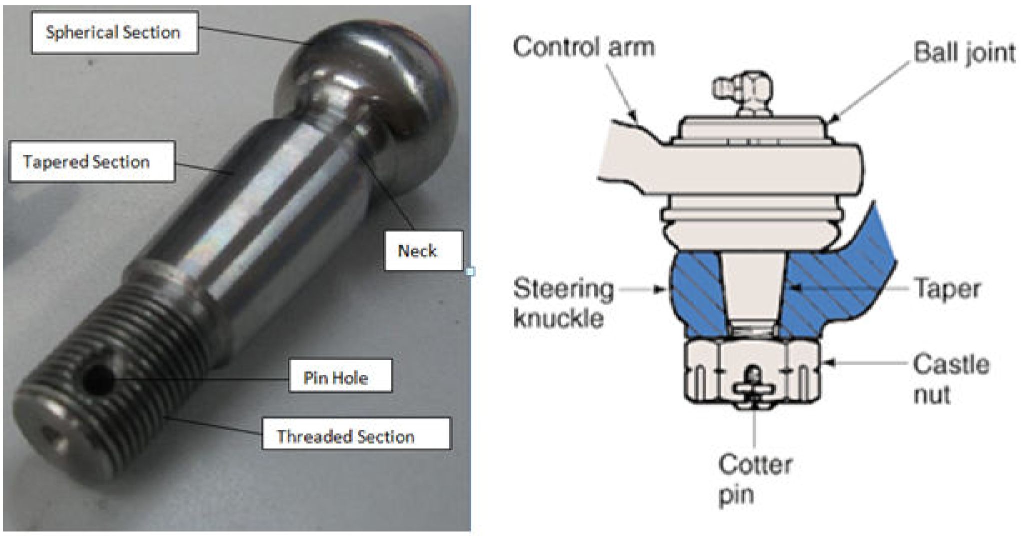


Figure 2.22. Tie rod joint: (1) Housing, (2) Ball Pin, (3) Bearing, (4) End Cap, (5) Dust Boot, (6/7) Clamping Ring, (8) Pin Hole (Lemförder, 2002).

In Figure 2.22, It can be seen a pre-lubricated tie rod joint used on passenger cars and light vans manufactured by Lemförder Fahrwektechnik. The joint housing (item 1) has a fine thread on the shaft and made of annealed steel C35V, surface-hardenable steel 41Cr4V is used for ball pin. The actual bearing element -the one-part-snap-on shell 3 made from polyacetal (e.g. DELRIN made by Dupond) surrounds the ball, the rolled end cap 4 ensures a dirt and waterproof seal. The polyurethane or rubber dust boot 5 is held against the housing by the clamping ring 6 and 7 (Reimpell, Stoll, & Betzler, 2002).

2.9.1. Ball Pin

The basic ball joint is comprised of a ball mounted in a socket housing (bearing). The ball pin profile can be divided into three sections; at one end the pin is parallel and threaded, the middle section is tapered and the opposite end is spherically shaped (Fig. 2.23-a). The tapered middle section of the pin fits into a similarly shaped hole made at one end of the steering arm so that when pin is drawn into the hole by the threaded nut the pin becomes wedged (Fig.2.23-b). A self locking nut can also be used instead of slotted castle nut and split pin (see Fig. 1.3) (Heisler, 2002).



(a) (b)

Figure 2.23.(a). Ball pin sections, (b) Ball pin connection (Heisler, 2002)

2.9.2. Bearing

The spherical end of the ball is sandwiched two half hemispherical socket sets which may be positioned at right angles to the pin axis. Modern ball and socket joints may use the ball housing itself as the half socket formed around the neck of ball pin (see Figure 2.24) (Heisler, 2002).

The other half socket which bears against the ball end of the ball pin is generally made from oil impregnated sintered iron, another type designed for automatic chassis lubrication, an induction hardened pressed steel half socket is employed. Both cases are spring loaded to ensure positive contact with the ball all times. A helical (slot) groove machined across the shoulder of the ball ensures that the housing half socket and ball top face is always adequately lubricated and at the same time provides a bypass passage to prevent pressurization within the joint (Heisler, 2002).

To reduce the risk of binding or seizure and to improve the smooth movement of the ball when it swivels, particularly if the dust cover is damaged and the joint becomes dry, non-metallic sockets are preferable (Figure 3.24-a). These may be made from moulded nylon and for same applications the nylon may be impregnated with molybdenum disulphide. Polyurethane and Teflon have also been utilized as a socket material to some extent. With the nylon sockets the ball pin throat half socket and the retainer cap is press fit in the bore of the housing and float. The coil spring accommodates initial settling of the nylon and subsequent wear and the retainer cap is held in position by spinning over a lip on the housing. To prevent the spring loaded half socket from rotating the ball, two shallow tongues on the insert half socket engage with slots in the floating half socket. These ball joints are suitable for light and medium duty and for normal load working conditions have an exceptionally longer service life (Heisler, 2002).

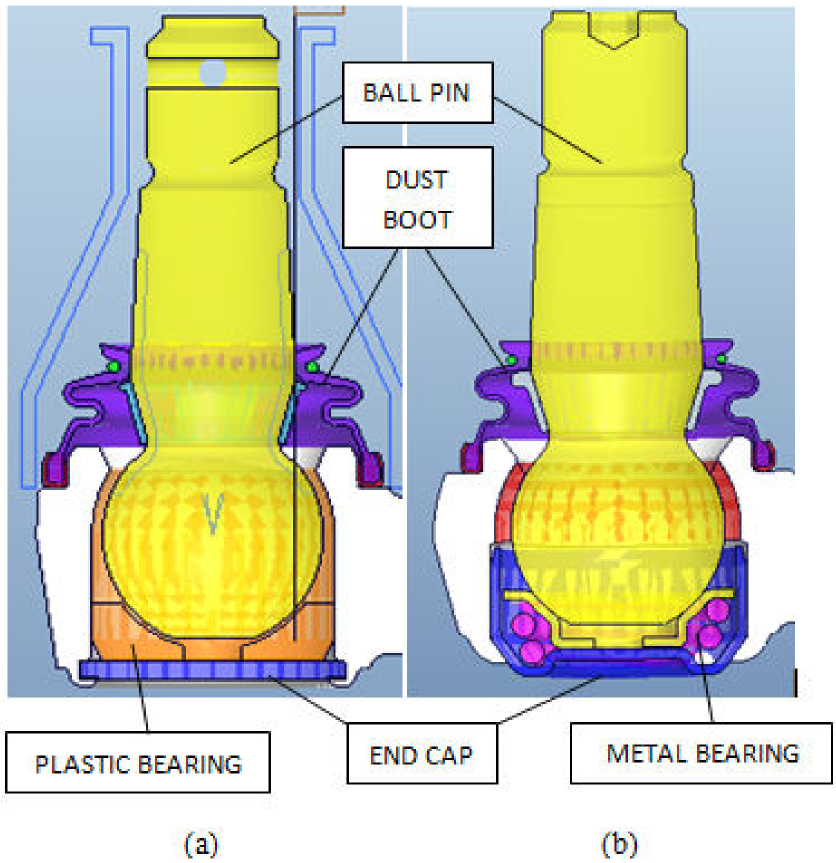


Figure 2.24.(a). Plastic bearing design, (b) Metal bearing design

2.9.3. Housing

Housing holds all bearing system and designed to provide sufficient tilt to ball joint angle for rotation of wheels. All parts are manufactured from low carbon steel and heat-treated for long life and wear resistance. In addition, they are zinc plated and yellow dichromate treated for corrosion resistance and consistent appearance. The precision ball stud is inserted into the shell and locked in place by a pre-located snap ring allowing for sufficient movement of the ball stud without excessive play. Right-left-handed and outer-inner threads (for metric sizes) are available in a variety of configurations (www.cablecraft.com). Housings are entitled by its articulation, it is called as elliptical housing if axial and transverse articulation angle is different if articulation angle is same it is called as not elliptical (Fig. 2.25). Tilt angles of ball joints are one of the most important design factors which effect steering and safety of vehicles (Heisler, 2002).

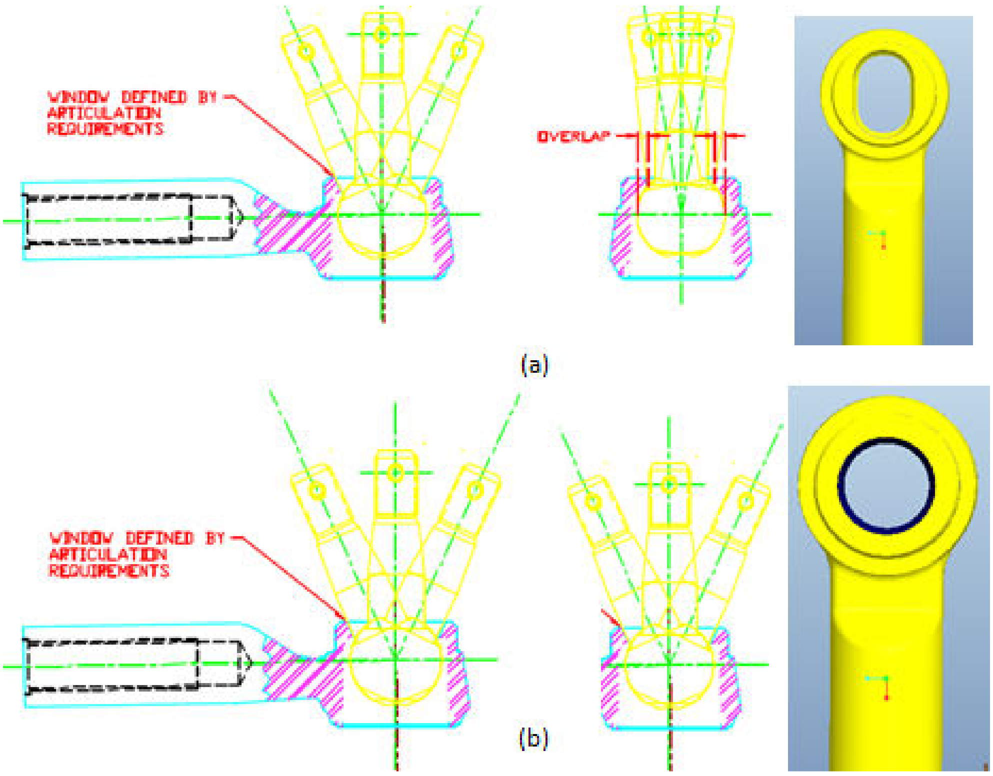


Figure 2.25. (a). Elliptical Housing design (b) Oval housing design

2.9.4. Dust Cover

An important feature for a ball type joints is its dust cover, often referred to as the boot or rubber gaiter but usually made from either polyurethane or nitrile rubber mouldings, since both of these materials have a high resistance to attack by ozone and do not tend to crack or to become hard and brittle at low temperature. The purpose of dust cover is to exclude road dirt, moisture and water, which if permitted to enter the joint would embed itself between the ball and socket rubbing surfaces. The consequence of moisture entering the working section of the joints is that when the air temperature drops the moisture condenses and floods the upper part of the joint. If salt products and grit are sprayed up from the road, corrosion and a mild grinding action might result which could quickly erode the glass finish of the ball and socket surfaces. This is then followed by the pitting of the spherical surfaces and a wear rate which will rapidly increase as the clearance between the faces becomes larger (Heisler, 2002).

Slackness within the ball joint will cause wheel oscillation (shimmy), lack of steering response, excessive tire wear and harsh or notchy steering feel.

Alternatively, the combination of grease, grit, water and salt may produce a solid compound which is liable to seize or at least stiffen the relative angular movement of the ball and socket joint, resulting in steering wander.

The dust boot must give complete production against exposure from the road but not so good that air and the old grease is pumped into the joint at high pressure, otherwise the boot will burst or it may be forced off its seat so that the ball and socket will become exposed to the surroundings.

The angular rotation of the ball joint, which might amount to 40° or even more, must be accommodated. Therefore, to permit relative rotation to take place between the ball pin and the dust cover, the boot makes a loose fit over the ball pin and is restrained from moving axially by the steering arm and ball pin shoulder while a steel ring is molded into the dust cover makes a tight fit over the large diameter socket housing by a steel band which tightly grips the boot (Heisler, 2002).

2.9.5. Ball Joint Lubrication

Before dust cover were fitted, ball joints needed to be greased at least every 1600 kilometers. The advent of dust covers to protect the joint against dirt and water enabled te grease recharging intervals to be extended to 160000 kilometers. With further improvements in socket materials, ball joint design and the choice of lubricant the intervals between greasing can be extended up to 500.000 kilometers under normal road working conditions. With the demand for more positive and reliable steering, joint lubrication and the inconvenience of periodic off the road time, automatic chasis lubrication system via plastic pipes have become very popular for heavy commercial vehicles so that a slow but steady displacementof grease through the ball joint system takes place. The introduction to split socket mouldings made from non metallic materials has enabled a range of light an medium duty ball and socket joints to be developed so that they are grease packed for life. They threfore require no further lubrication provided that the boot cover is a good fit over the socket housing and it does not become damaged in any way (Heisler, 2002).

2.9.6. End Cap

End cap is used to close bottom side of housing and top side is closed with dust boot so all ball bearing system is enclosed that provides a protective area against dirt, moisture and water. In addition, end cap must have enough strength to load coming axially from ball pin (pull-out load). They are zinc plated and yellow dichromate treated for corrosion resistance and consistent appearance according to housing which they are used together with (Heisler, 2002).

2.9.7. Spring

Spring is used in metal ball bearing designs and it effects the radial and axial elasticity of ball bearing. It also provides positive contact between ball pin and metal bearing (see Fig. 2.26)

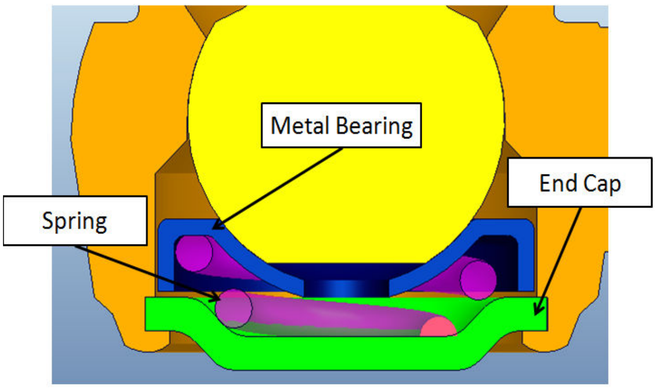


Figure 2.26. Spring with end cap

2.9.8. Clamping Ring

Clamping Ring is used to connect dust boot to housing and ball pin. Clamping ring can be produced from wire and sheet (Fig. 2.27). Sheet type clamping ring is only used to connect housing and dust boot. Wire type clamping ring can be used to use both dust boot-housing and dust boot-ball pin connection. Wire is cheaper than sheet but sheet is safer than wire. Therefore, wire is used ball joints for small sphere diameter (Ø27, Ø30), sheet is used for ball joints with bigger sphere diameters (Ø35, Ø40).

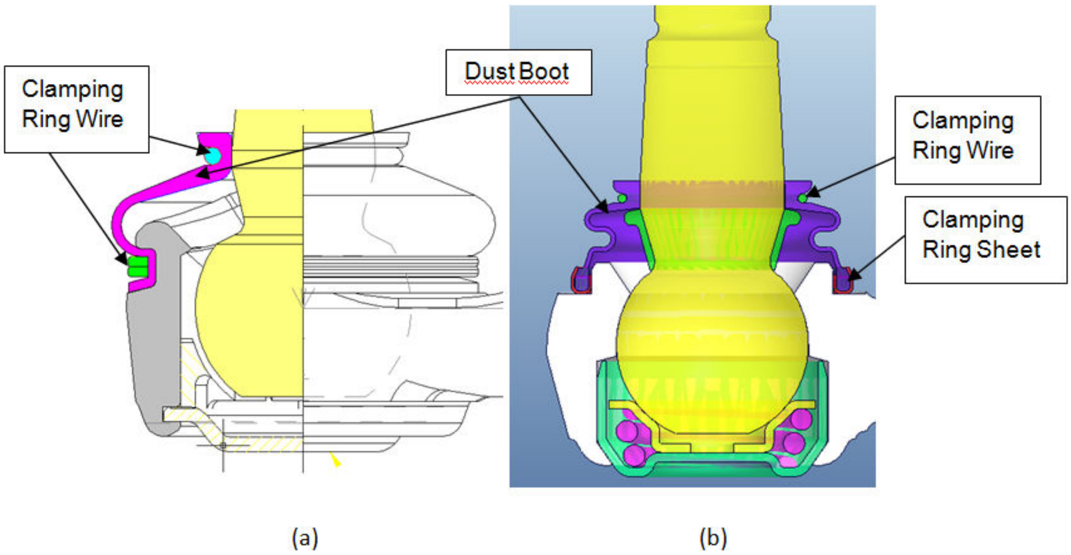


Figure 2.27. Wire and sheet clamping ring connection

3. MATERIAL AND METHOD

The computer is an important tool to design an engineering system. CAD (Computer Aided Design) system is inevitable in design practices. It is desired to have an application that supports the entire lifecycle of initial design, configuration design, detail design, manufacturing, and disassembly. Design changes frequently through the design stages while the optimal design is supported by the engineering analyses such as static analysis, dynamic analysis, and FEM (Finite Element Method). To meet the market requirements, the old mass production system is being changed into the mass customization system. The production of a small volume with diverse product types is the new trend. The parametric modeling technique is useful when the geometric model should be changed frequently during the design process. The geometric changes of a part can also influence the assembly of the model (Myung & Han, 2001).

There have been substantial developments in the methodologies and technologies used to represent assembly models. In the early days, assembly models were documented using text and graph based methods. These methods were then evolved into graph-based methods, providing two-dimensional and/or three dimensional visual representations of assembly models. The current state of the art and practice in industry for assembly model representation is the use of commercially available CAD systems. Such CAD systems can visually display the assembly as well as allow easy manipulation of the virtual assembly for design, planning and presentation purposes. It is believed that these digital assemblies, created using modern CAD systems, contain useful information for assembly sequence generation (Ou & Xu, 2013).

This chapter proposes a new approach to parametric design system which supports not only parts but also assembly using a commercial CAD system and a new developed software. The system uses feature representation, can handle geometry, and performs design reasoning process when design changes occur during the parametric modeling of products. Finally, the system has been applied to the design of ball joint assembly to verify the concept.

3.1. Development of A New Approach to Parametric Ball Joint Design

Parametric design should involve definition, reasoning and presentation phases. In definition phase, the problem is clearly defined and presented, in reasoning phase, symbolic processing is done while requirements and constraints are being evaluated until a solution is reached and in presentation phase, recommended solution is graphically presented (Badiru A. B., 1992). In this study, commercially available software are used harmonically for the parametric design of ball joints using a developed design platform that is called as *parametric design platform.* Problem is defined in a simple Microsoft Office Excel sheet which contains all parameters needed for reasoning phase. For reasoning PTC Pro/Engineer 5.0 Pro/program module is used. Pro/Program module contains a programmable menu to read and processing parameters in the Ms Excel sheet. The module is capable of making algebraic calculations with mathematical formulations, so any dimensions in 3D part or assembly model can be calculated according to parameters. In addition, the most important capability of Pro/program is that all modeling functions of Pro/engineer 5.0 assembly and part module can be activated or inactivated with pre-defined logical statements and rules. In order to apply rules to 3D models first, all combinations of assembly and parts are manually designed in PTC Pro/Engineer 5.0 then, necessary functions are activated or inactivated with logical statements and dimensions are calculated using mathematical formulation in Pro/program menu, consequently solutions are presented in PTC Pro/Engineer 5.0 as 3D model and 2D drawing. Phases of suggested parametric design which is called as parametric design platform is created and shown graphically in Fig. 3.1.

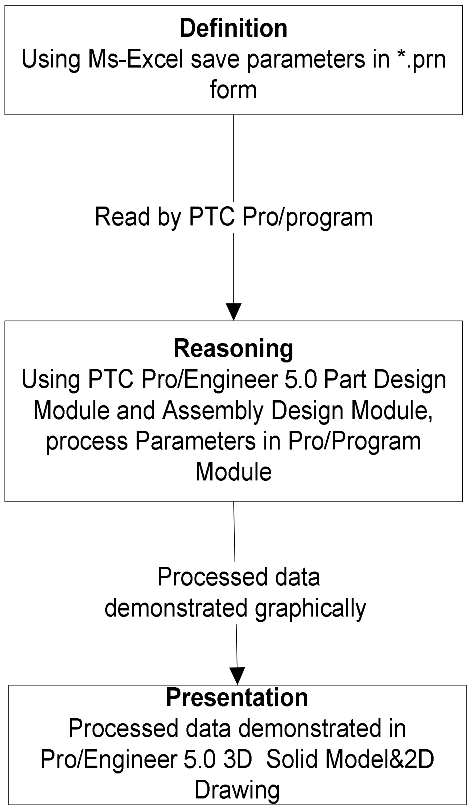


Figure 3.1. Parametric design platform

In order to generate a systematic parametric design system an algorithm has been developed and used to determine relations between definition, reasoning and presentation phases as shown Fig. 3.2.

The developed parametric design algorithm is as follows:

Step 1: Define parameters. Parameters must include targets of design and all of the data to reach targets.

Step 2: Design 3D solid part modeling with features manually in PTC Pro/Engineer. Solid parts consist of one or multiple features. As seen in the structure of feature in Fig. 3.3 every feature has a geometry consists of points, lines, curves, planes and surfaces and dimensions. Shape of geometry (points, lines, curves, planes and surfaces) are stable (cannot be programmable) but dimensions are programmable.

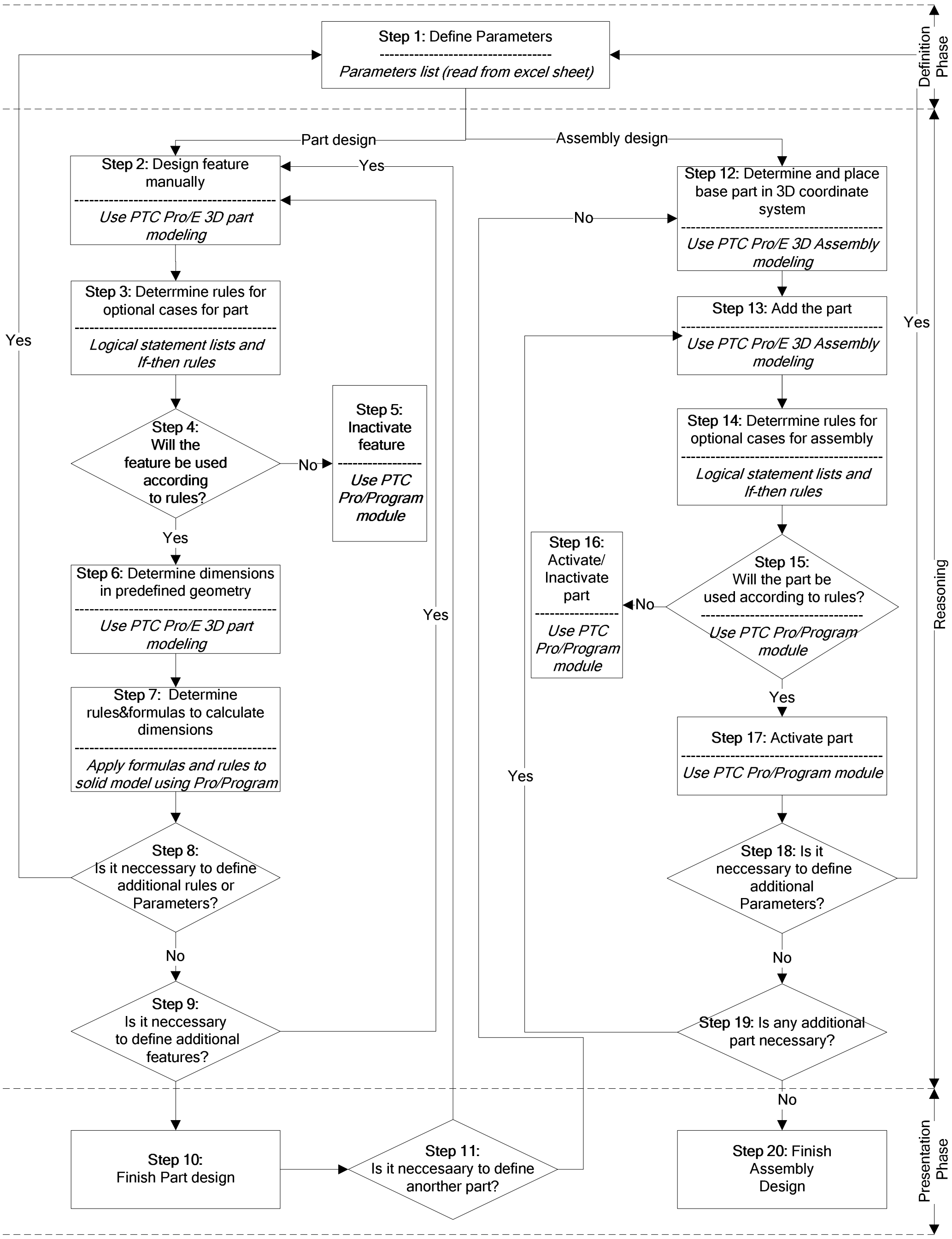


Figure 3.2. Proposed parametric design algorithm

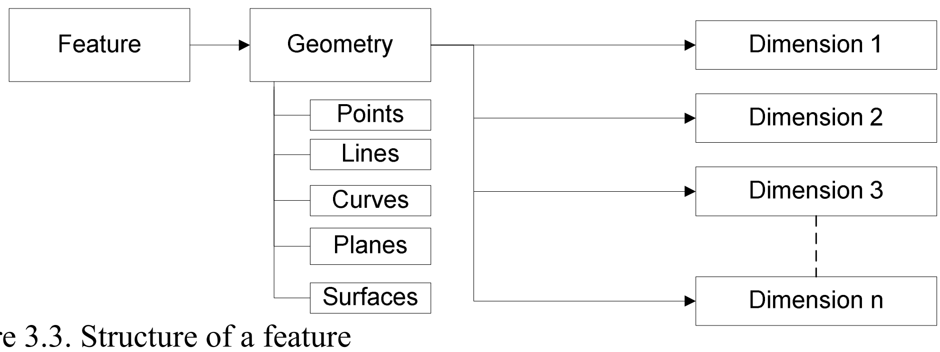


Figure 3.3. Structure of a feature

Step 3: Determine rules for optional cases for the part. Options are the alternatives to reach targets defined with parameters.

Step 4: Decide whether the feature is used or not according to rules in Step 3. If the feature isn't used, go to Step 5. If it is used, go to Step 6.

Step 5: In activate feature using PTC Pro/engineer Pro/program module.

Step 6: Determine each dimensions in 3D model and give a name for each dimensions.

Step 7: Determine rules and formulas to calculate dimensions and apply them to solid model of part using PTC Pro/program module. Formulas are used to process “defined parameters” based on the defined parameters at the Step 1, and rules set at Step 3.

Step 8: If defined parameters or rules are insufficient to calculate dimensions, algorithm directs the program to Step 1, and it repeats all these steps.

Step 9: This step is used to provide a decision whether the part is completed with designed features or not. If additional feature is needed, the algorithm directs the program to Step 2, and it repeats all the steps.

Step 10: Finish part design if completed features are suitable.

Step 11: This step is used to provide a decision whether the parametrically designed parts are suitable for perfect assembly or not. If not it repeats steps from 2 to 10 until all the remaining parts of the assembly are designed parametrically.

It should be noted that steps 1 to 11 must be applied to all of the potential parts in assembly. After finishing the design of all the parts of the assembly, the full assembly can be designed parametrically.

Step 12: It is named as "Determine and locate the base part in the coordinate system of 3D modeling program". In this step, the part must be stable (not optional) and convenient to build subsequent parts on it.
Step 13: This step is "Add part". The part is located in the reference positions of the base part. For the reference positions, constraints must be defined.
Step 14: This step is called as "Determine rules for optional cases for assembly". Options are the alternatives to reach targets defined with parameters. Even if targets are reachable with different ways, the best way is chosen with rules.
Step 15: This step is another decision step and named as "Will the part be used according to rules". In this step, the decision is given whether the part will be used according to rules as defined in Step 14 to activate or inactivate the part.
Step 16. This step is "Inactivate part". Inactivation of part means that part will not be seen on the screen of 3D modeling program. Inactivation is automatically done by Pro/program software.
Step 17: This step is "Activate part". Activation of part means that part will be seen on the screen of 3D modeling program located in the reference position of base part. Activation is automatically done by Pro/program software.
Step 18: This step is "Is it necessary to define additional parameters". If parameters are insufficient to give decisions according to rules to reach targets, it is necessary to add or include new additional parameters. If it is necessary, these are defined, and then all these steps are repeated.
Step 19: This is the last decision step and called as "Is any additional part needed". The proposed parametric design algorithm decides whether any additional part is needed to complete assembly design. If it is needed, it goes to Step 13 and repeats all these steps (from Step 13 to Step 19).
Step 20: This is the last step and called as "Finish Assembly Design". After all the rules are applied, and all the necessary dimensions are calculated, assembly design is finished.

3.1.1. Flow Diagram of New Approach

Parametric modeling allows re-use of existing products and rapid design modification based on the results of engineering analysis (Shin & Kwak, 1999). In a feature-based modeling system, the level of detail for assembly is important. It should be decided among which level of detail they are manipulated. The different levels of detail are parts (level 1), features (level 2), and dimensions (level 3) (Myung & Han, 2001).

Using the flow diagram of new approach as shown graphically in Fig. 3.4, a designer can easily design full assembly by providing a small number of values of specified parameters instead of the all enumerated description of each part. To do this, first, the specified parameters should be defined and listed in definition phase. Next, existing design should be modeled in CAD environment with top-down design approach as it has mentioned in Section 2.6 of this study. It should allow parametric modeling of an assembly using parameters in definition phase.

To implement the parametric modeling of an assembly, following steps have to be taken:

Step 1: Modeling the parts.
Step 2: Making an assembly with the parts.
Step 3: List up all critical dimensions for the parametric modeling and naming the dimensions.
Step 4: Making the design knowledge-base which contains the parametric relations of each dimension and formulae.

Assembly models developed using modern CAD systems are made up of a combination of different components imported into the assembly model. All the components imported have different constraints specified to define the location of the target positions relative to the assembly coordinate system (Ou & Xu, 2013).

Parametric modeling has two approaches to manipulate design. One is

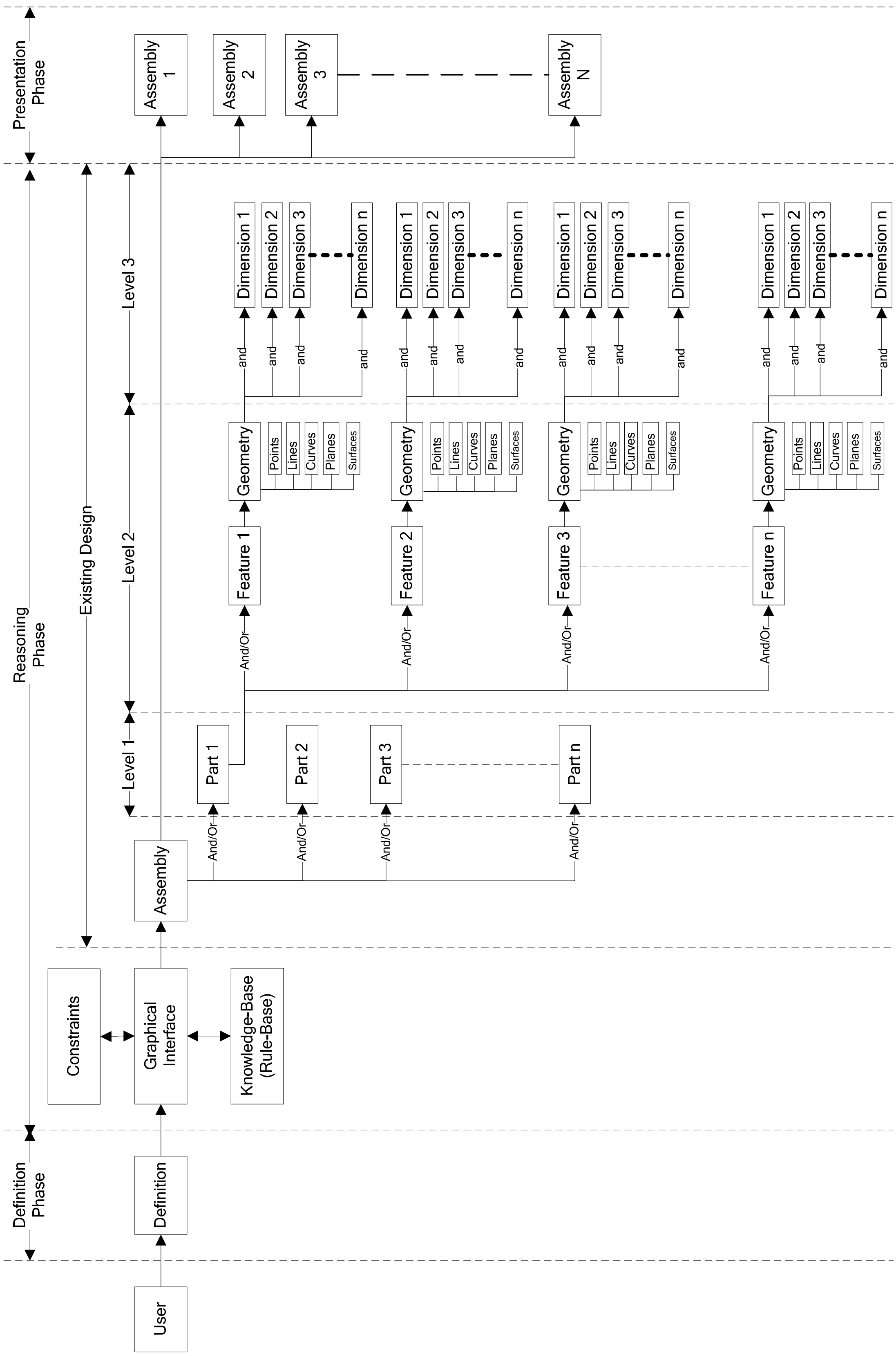


Figure 3.4. Flow diagram of new approach to parametric design

algebraic approach and the other is AI (Artificial Intelligence) approach (Verroust, Schonek, & Roller, 1992). In this work, AI approaches is used to manipulate level 1&2 (parts & features) of design, algebraic approach is used to manipulate level 3 (dimensions) of design. Algebraic approach calculates each value of dimensions with formulas using parameters in definition phase, AI approaches decides with logical statements which parts and features are or aren't used to do desired functions defined in definition phase.

Logical statements and formulas are stored in a knowledge-base and processed in Graphical Interface (GI). Knowledge based systems are computer applications that use stored knowledge for solving problems in a specific domain. The rules in the knowledge-base are used to process the data representation of the product. This representation is made of interrelated objects as shown in Fig. 3.4. They constitute the representational dimension. As a result of process in GI, the desired final assembly is made available by the *parametric design platform*.

In Knowledge-Based Engineering (KBE), all the possible expressions used to define attributes, specify the number and type of objects, communicate with external tools, and so on, are addressed with the generic term of rules (or engineering rules). For this reason, KBE is often addressed as a technology to perform rule based design. There is nothing wrong with this description of KBE, so long as the fundamental difference with conventional rule based systems is acknowledged, where all rules are of the type IF-THEN and there is a crisp separation between reasoning mechanism and knowledge base (La Rocca, 2012).

The steps of designing the parametric system with new approach as follows:

Step 1: Define the inputs to design all system parametrically.

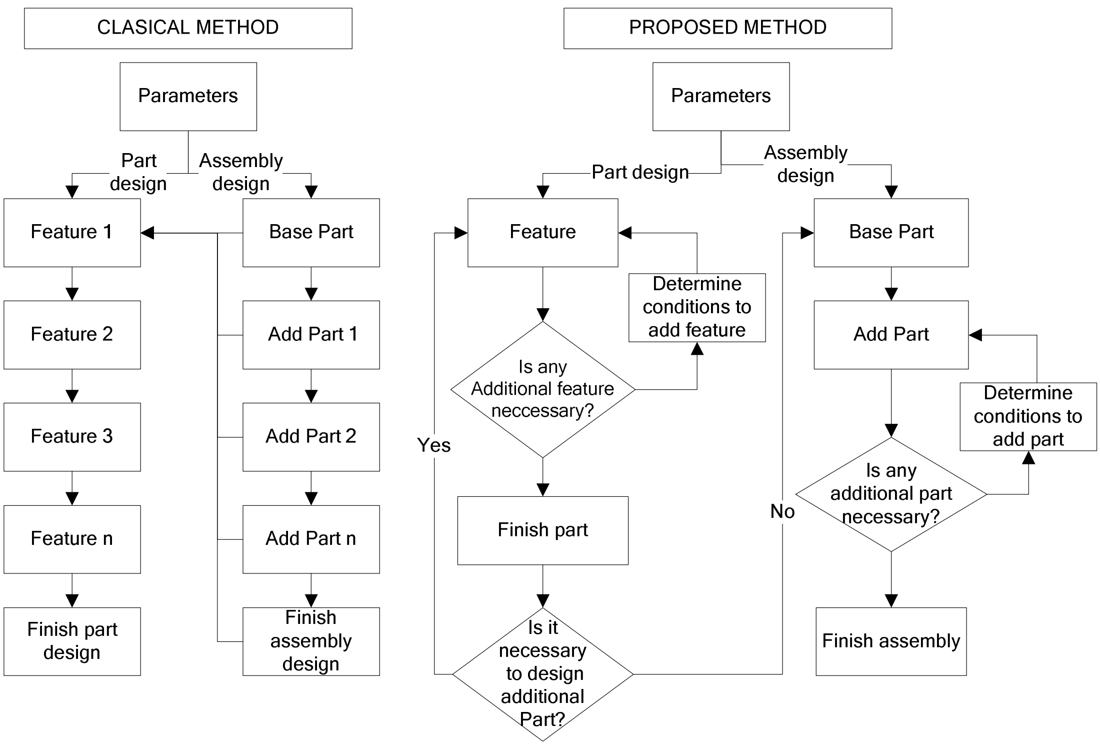
Step 2: Model all system in 3D with top-down approach using a CAD program.

Step 3: Prepare a knowledge-base and constraints to reach desired final product.

Step 4: Process inputs, knowledge-base in a graphical interface (GI) to present desired final product.

The major advantage of the proposed system is that the system can parametrically change assembly, part, part material, feature, geometry and dimensions in a programmable environment. This provide a wide range of alternative solutions to design every parts of ball joints systematically. Whereas, parametric systems which is not programmable provide change in dimensions only.

In classical parametric system assembly, parts of assembly and features of parts are stable as shown in Fig. 3.5 a. Dimensions can be changed parametrically and logical statements in programmable environment can only be applied on dimensions. The proposed parametric system can change assembly, parts of assembly, and features of part parametrically in a programmable environment as shown in Fig 3.5 b. These provide a wide range of alternative solution to design every parts of ball joint systematically. Additionally, rule-based algorithm of proposed system allows adding gained experiences to the proposed parametric design system.



(a) (b)

Figure 3.5. (a). Classical parametric design algorithm (b) Proposed parametric design algorithm

3.1.2. Design of Definition Phase

Definition phase, in other words user interface (UI) is a platform to convert parameters into outputs usable by the reasoning phase software of PTC Pro/E. Defined parameters must be adequate manipulating design logically and algebraically to reach desired final product. Figure 3.6 shows a representation about interface between parameters and outputs. User interface should be defined graphically, and be as simple as possible in order to save time and cost in the design process.

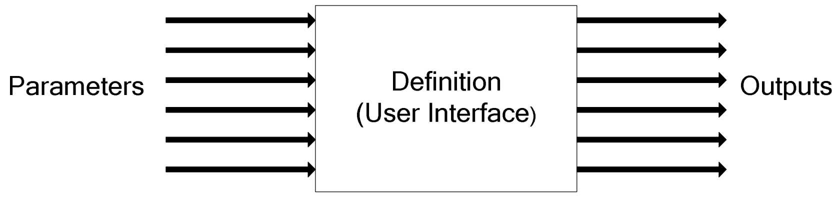


Figure 3.6. Parameters-outputs interface

Once the general problem area has been defined, the next function is to decide what to do about the problem. This is function of the reasoning phase. In this study, parameters are entered in a Microsoft Office Excel sheet (Fig. 3.7) for ball joint design. Excel sheet has been prepared graphically to enter input data easily. The output file must be in prn form (*.prn) which can be readable by Pro/Program module. Pro/Program is the Graphical Interface (GI) of parametric ball joint design (See Fig. 3.4).

Table 3.1 shows the parameters. Every parameter has a number, type and output and listed in the table. Parameters are divided into two types: string (text) and number (numeric). These are important for manipulating the design algebraically and artificially. Data entered in definition phase will be processed in Graphical Interface (GI) for artificially changing parts and features and calculating algebraically dimensions.

Table 3.1. Parameters for ball joint design

No.	**Parameters**	**Input Type**	**Outputs**
1	Bearing type	String	“Plastic” or “Metal”
2	Connection Type	String	“Castle nut” or “self-locking nut”
3	Housing Type	String	“Elliptic” or “Oval”
4	Housing shaft type	String	“Inner Thread” or “Outer Thread”
5	Dust boot working temperature	Number	(-60)-(+80)
6	Neck length	Number	neck length dimension
7	Taper length	Number	Any number
8	BP thread length	Number	Any number
9	BP thread	Number	Any number
10	Pin diameter	Number	Any number
11	Pin length	Number	Any number
12	Conic ratio	Number	5 or 6 or 8 or 10
13	Conic diameter	Number	Any number
14	Sphere diameter	Number	27 or 30 or 35 or 40
15	Housing shaft length	Number	Any number
16	Housing thread length	Number	Any number
17	Housing hole length	Number	Any number
18	Housing hole diameter	Number	Any number
19	Housing Shaft Diameter	Number	Any number
20	Allen Key Number	Number	6 or 8 or 10
21	Angle for Oval Housing	Number	Min 40
22	Angle for Elliptic Housing	Number	Min 20
23	Ball Pin Material	String	"41Cr4" or "42CrMo4"

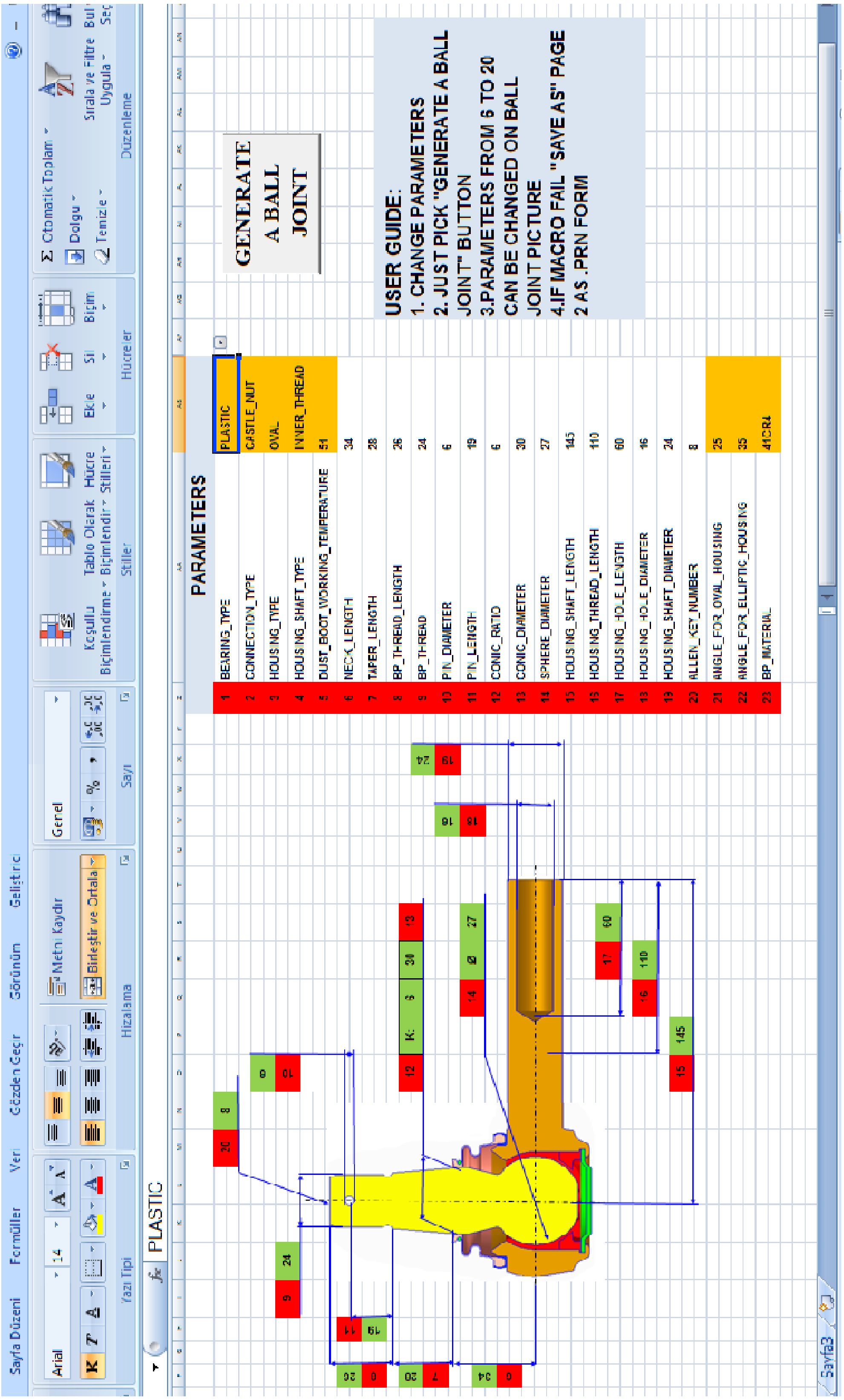


Figure 3.7. Definition sheet

3.1.3. Design of Reasoning Phase

Once the problem is defined, the next step is confronting the problem and finding a solution to it. This is reasoning phase and Pro/Engineer 5.0 and Pro/Program module is used in this study. In a parametric design, full system must be decomposed into final point that is controllable. In this study, ball joint system (assembly) is decomposed in Pro/Engineer 5.0.

Theoretical decomposition of an assembly in CAD system can be seen in Fig. 3.8. First, assembly is decomposed into potential parts (Level 1). Potential parts are consisted of features (Level 2). Features essentially define the basic structures that made up a particular model, thus a model is built up with one or more features as building blocks. In conventional modeling process, initially a base feature is created and this is further enhanced by adding other features or simply adding more details to it until the required model is obtained. Development of features follow the designer's intent and thus are subjected to changes as the design progress. Thus a model based on features can be changed by manipulating the features which in turns can be used to reduce the development time. If features are integrated with parameters and other features, changes made in one feature can be successfully propagated through the entire design (Gujarathi & Ma, 2011).

Features are revolve, extrude, hole, protrusion and similar functions of a CAD program (Pro/Engineer, CATIA, Unigraphics etc.). The selection of the feature is critical to the modeling process as it influences the parent/child references established within the model. The definition of modeling procedures for complex parts has to integrate information that will impact the choice of the based features and the selection of references/supports during a common modeling procedure (Bodein, Rose, & Caillaud, 2014).

It must be underlined that features has only one geometry consists of points, lines, curves, plane and surface. Shape cannot be changed in features but its dimensions can be altered by algebraic calculation using specified parameters (input) and knowledge base. All these potential parts, features, geometries, dimensions are listed and the possible relationship between them demonstrated graphically in Fig. 3.8.

In order to develop a parametric design for ball joint, the developed theoretical decomposition method has been applied to ball joint assembly as it can be seen in Fig. 3.8. First, ball joint is decomposed into potential parts. Potential parts of ball joint is ball pin, housing, plastic bearing, metal bearing, spring, dust boot, clamping ring wire, clamping ring sheet and end cap. As shown in Fig. 3.8 some parts and some features must be used essentially, some parts and some features are optional.

Optional parts and features are shown with "or" function, the essential parts and features are shown with "and" function.

Dimensions are also calculated by using mathematical functions given in Table 3.2 according to inputs and outputs in definition phase. Logical statements for optional parts and features of a ball pin are listed in Table 3.3. After determining parts and features for ball joints, it is necessary to find out dimensions.

In the design of ball joint, dimensions must be calculated using inputs given in Table 3.1. The uses of mathematical and logical functions are given in Table 3.2. As it can be shown in ball joint input sheet in Fig. 3.7 dimensions are related to "ball joint" or "housing". So, it can be understood that all other parts dimensions (bearings, dust boots, end caps, clamping rings, springs) can't be changed manually, it is noted that all parts will be designed according to inputs in the sheet. The most important parameter for these parts is the ball pin sphere diameter. According to sphere diameter, all the dimensions are calculated algebraically. The dimensions can be changed manually in definition sheet for "ball pin" and "housing", and these dimensions are calculated algebraically using input values.

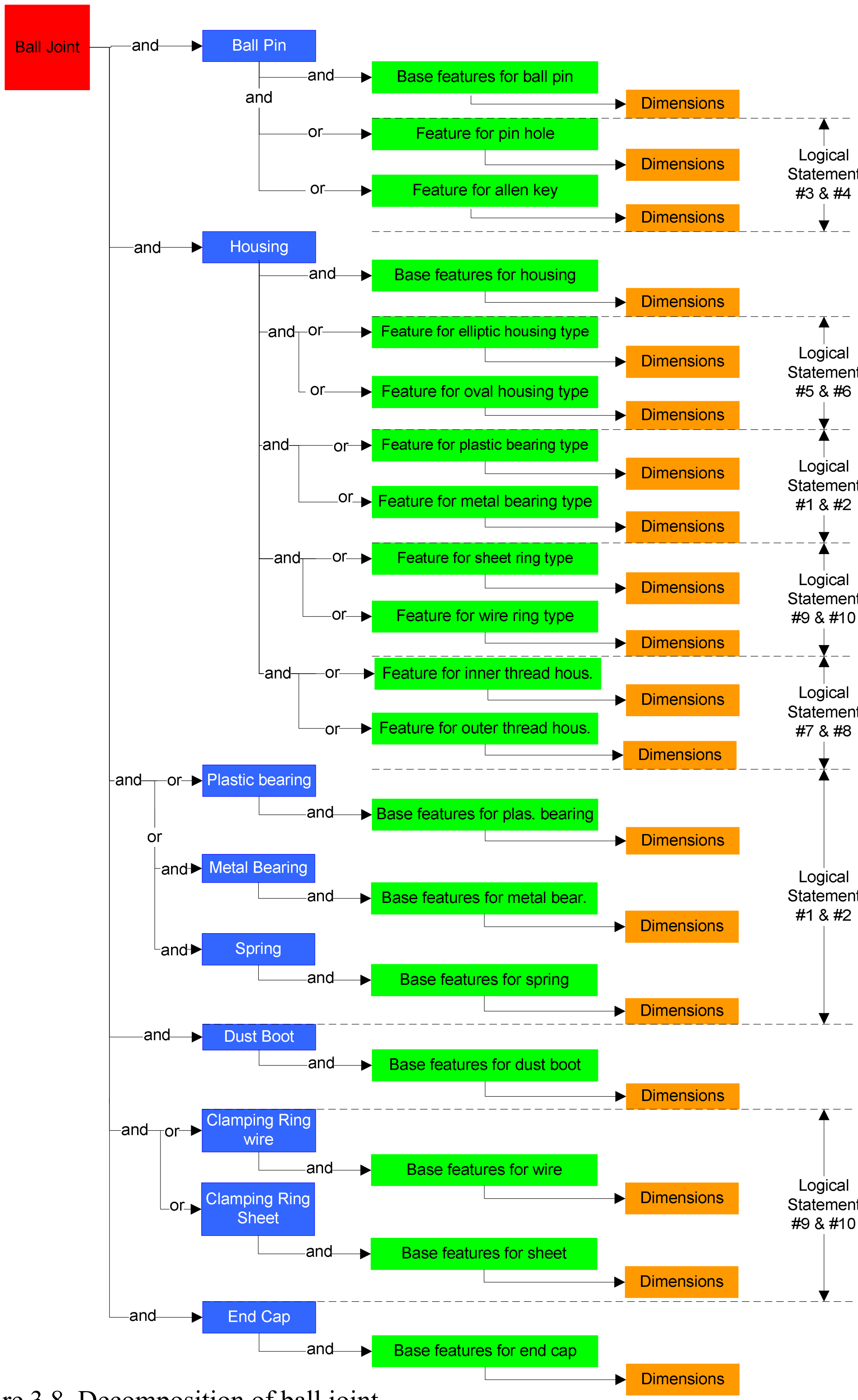


Figure 3.8. Decomposition of ball joint

Table 3.2. Mathematical and logical functions in PTC Pro/Program

+	Addition
-	Subtraction
/	Division
*	Multiplication
^	Exponentiation
=	Equal to
( )	Parentheses for Grouping
>	Greater than
>=	Greater than or equal to
!=	Not equal to
&	And
|	Or
!	Not
Sin ( ), Cos ( ), Tan( ), Cot ( )	Trigonometric Functions
Abs( ), Log ( ), ln( )	Absolute, Logarithmic Functions

Based on constraints, the CAD system should allow for quicker design solutions, giving time to the human team member to do what they do best, be creative and search for new scenarios. The system should give back the time to be an engineer (Chapman & Pinfold, 1999).

The type of joint used can be defined by the mechanism constraints that have been defined for connections between the components. The joints that are considered by the developed system include pin, slider, cylinder, planar, ball, bearing, Gimbal and slot. In a mechanism constraint, displacement limits may be defined. An assembly relationship between two components refers to a pair of geometric entities, one from each concerning component, that either completely or partially define an assembly constraint between the two components. To fully define a mechanism constraint, it is likely that more than one assembly relationship is required. Assembly relationships extracted are stored in the form of matrices (Ou & Xu, 2013). Part list, assembly relation matrix and critical dimension for ball joint assembly are shown in Fig. 3.9.

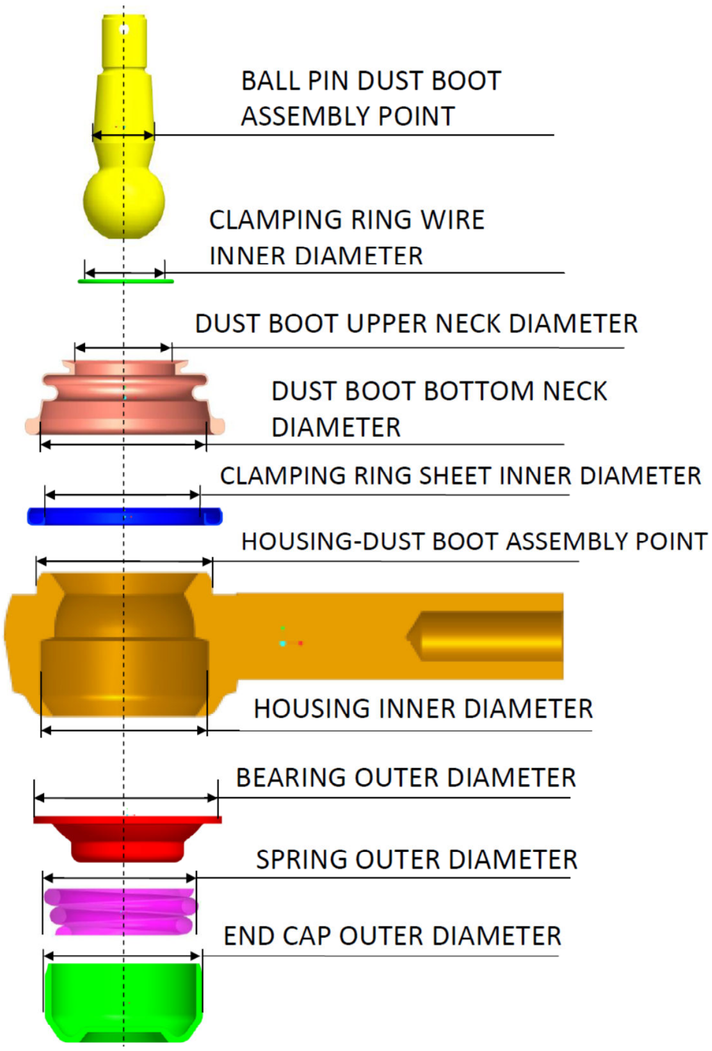
Table 3.3. Logical statements (Rules) for ball joint optional part and features

No	Input	Output	Logical statement
1	Bearing Type	“plastic”	If "bearing type" is "plastic" use "part (plastic bearing)" and in "part (housing)" use "feature cut for plastic bearing"
2	Bearing Type	“metal”	If "bearing type" is "metal" use "part (metal bearing)" and "part (spring)" and in "part (housing)" use "feature cut for metal bearing"
3	Connection Type	“castle nut”	If "connection type" is "castle nut" in "part (ball pin)" use "feature for pin hole"
4	Connection Type	“self-locking nut”	If "connection type" is "self-locking nut" in "part (ball pin)" use "feature for Allen key"
5	Housing Type	"elliptic"	If "housing type" is "elliptic" in "part (housing)" use "feature (cut) for ellipse"
6	Housing Type	"oval"	If "housing type" is "oval" in "part (housing)" use "feature (cut) for oval"
7	Housing Shaft Type	"Inner Thread"	If "housing shaft type" is "inner thread" in "part (housing)" use "feature for inner thread"
8	Housing Shaft Type	"Outer Thread"	If "housing shaft type" is "outer thread" in "part (housing)" use "feature for outer thread"
9	Sphere Diameter	27	If "sphere diameter " is "27" or "30" use "part (clamping ring wire) and in "part (housing)" use "feature for clamping ring wire"
10	Sphere Diameter	30 or 35 or 40	If "sphere diameter " is "35" or "40" use "part (clamping ring sheet) and in "part (housing)" use "feature for clamping ring sheet"

Assembly relations matrix is a square matrix of size equal to the total number of components. Relationships between components in the assembly are stored as an integer value. The values in the matrix represent three different scenarios; these are as follows:

- A value of ‘‘0’’ indicates that there are no relationships between the two components.
- A value of ‘‘1’’ indicates that there is only one relation stated in one constraint
- A value of ‘‘>1’’ indicates that there are multiple relationships between the components stated in multiple constraints

Constraints must also be stated in IF-ELSE form as in knowledge-base. Assemble constraints is prepared in Table 3.4 for ball joint according to critical dimension for assembly of ball joint parts (see Fig. 3.9).



COMPONENT LIST	
1	Ball Pin
2	Housing
3	Plastic bearing
4	Metal Bearing
5	Spring
6	Dust Boot
7	Clamping ring wire
8	Clamping ring sheet
9	End Cap

BALL PIN ASSEMBLY RELATION MATRIX

	1	2	3	4	5	6	7	8	9
1	0	0	1	1	0	1	1	0	0
2	0	0	1	1	1	1	1	0	1
3	1	1	0	0	0	0	0	0	0
4	1	1	0	0	1	0	0	0	0
5	0	1	0	1	0	0	0	0	1
6	1	1	0	0	0	0	1	1	0
7	1	1	0	0	0	1	0	0	0
8	0	0	0	0	0	1	0	0	0
9	0	1	0	0	0	0	0	0	0

Figure 3.9. Part list, critical dimension for assembly and assembly relation matrix for ball joint

Table 3.4. Ball joint assembly constraints

1	Ball Pin sphere diameter=Plastic Bearing diameter
2	Ball Pin sphere diameter=Metal Bearing diameter
3	Dust boot upper neck diameter=ball pin-dust boot assembly point diameter
4	Clamping Ring wire diameter=ball pin-dust boot assembly point diameter
5	Plastic bearing outer diameter=Housing inner diameter
6	Metal bearing outer diameter=Housing inner diameter
7	Spring outer diameter<Housing inner diameter
8	End cap outer diameter=Housing inner diameter
9	Spring outer diameter<metal bearing inner diameter
10	Dust boot bottom neck diameter=housing-dust boot assembly point diameter
11	Dust boot bottom neck diameter=Climbing Ring sheet diameter

3.1.4. Design of Presentation phase

The design of ball joint is composed of solid models of different parts constructed by Pro/E. Once a case is retrieved by reasoning process, the solution contains dimensions and technical specifications in 3D and 2D form (Fig. 3.10 and 3.11). The corresponding ball joint model can be reviewed for the following adaptation and revision to fulfill the redesign work. All the solid models are parameterized and the design provides rapid and convenient adaptation. With its parametric design capabilities, design knowledge can be easily added into the design during parameterization, which makes the part dimensions and assembly relations easily driven or regenerated by the parametric definitions. The Pro/ASSEMBLY module provides the functions of the part assembly, skeleton modeling and top–down design which are essential to our design system development. All in all, the CAD software operates as a platform and plays a core role in this system.

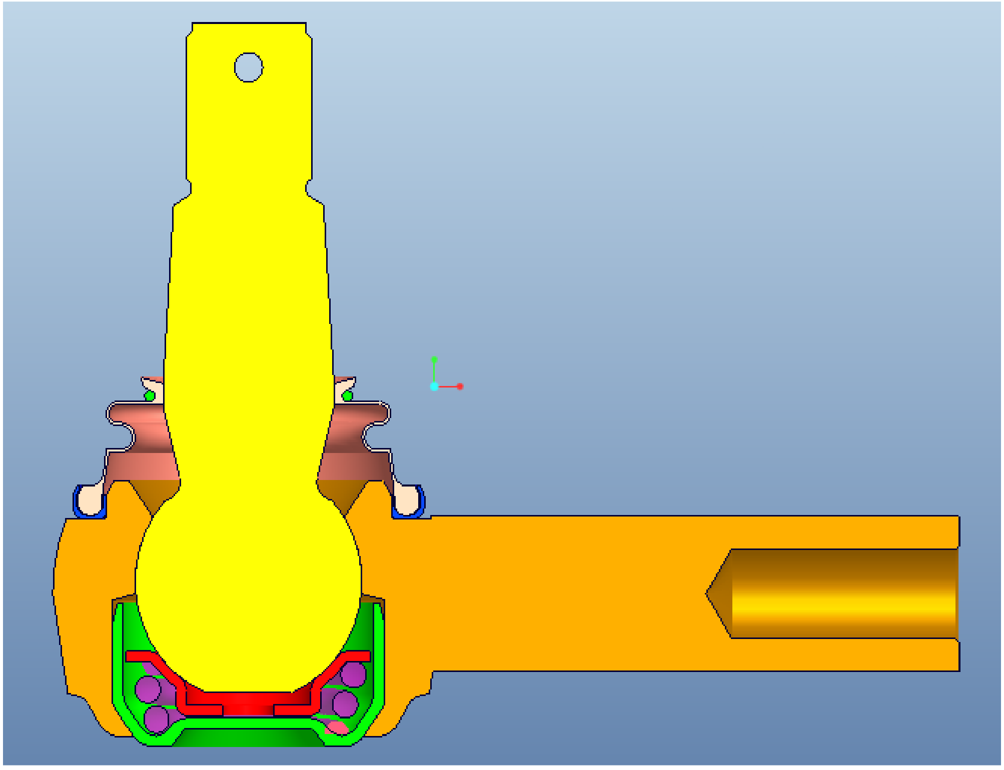


Figure 3.10. 3D form of ball joint

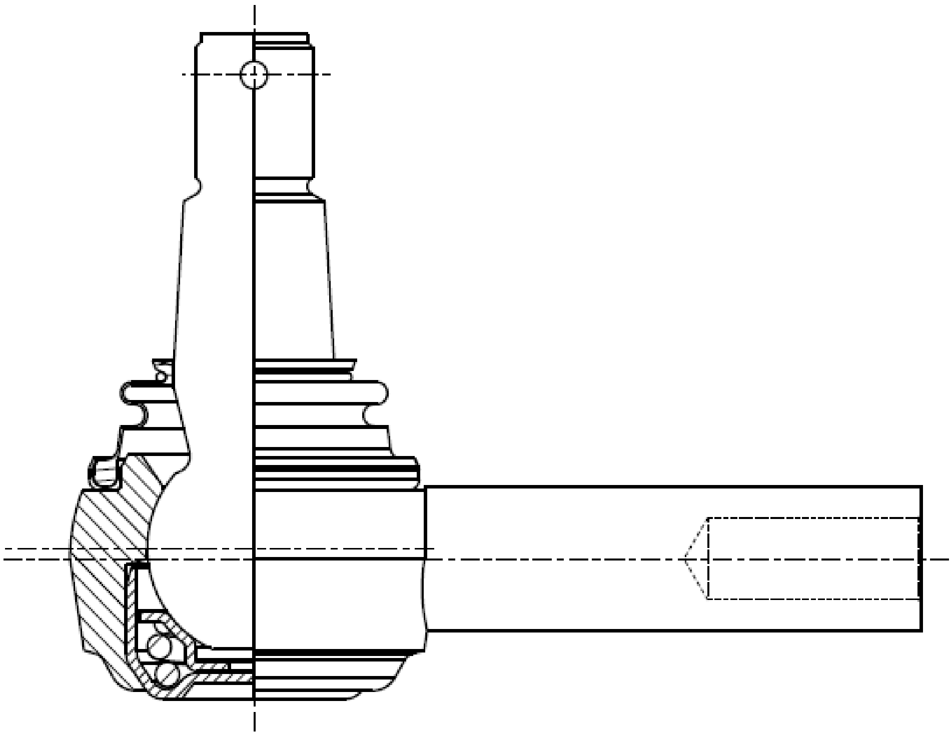


Figure 3.11. 2D form of ball joint

3.2. Technical Specifications of Ball Joints

First, forces applied on a ball joints in a steering system during movements of vehicles should be determined for the design of a ball joint. As seen in Fig. 3.12, ball joint is exposed an axial force (F1) on housing, an axial force on ball pin (F2), and a torsion on ball pin (M). Failure (fracture) of ball joints causes loss of control of vehicle and finally to fatal accidents, so in any condition even if it reduce the comfort conditions of the vehicle, there must be no fracture. Any failing in comfort variables of ball joints for example hard or soft steering wheel or short maintenance period effects customer satisfaction causing sharp falling in sales of vehicles.

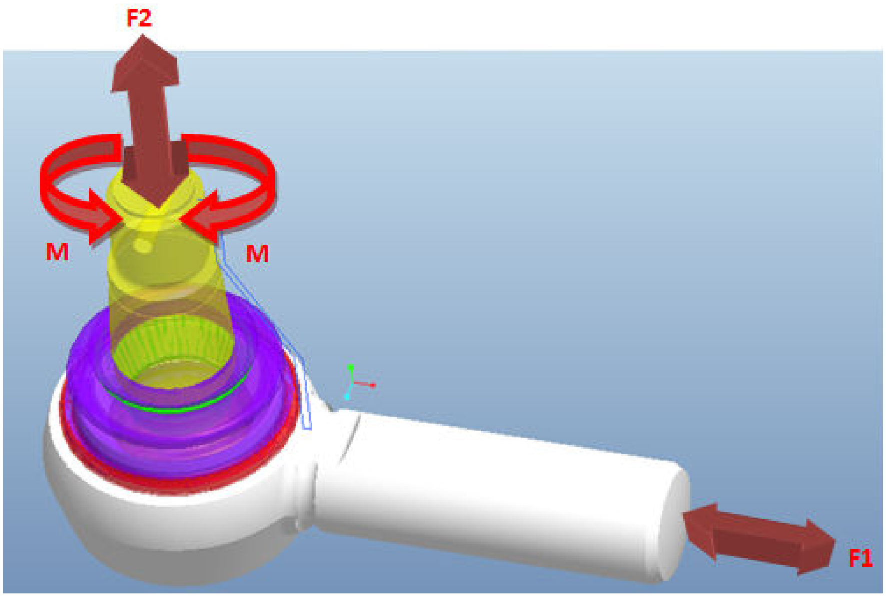
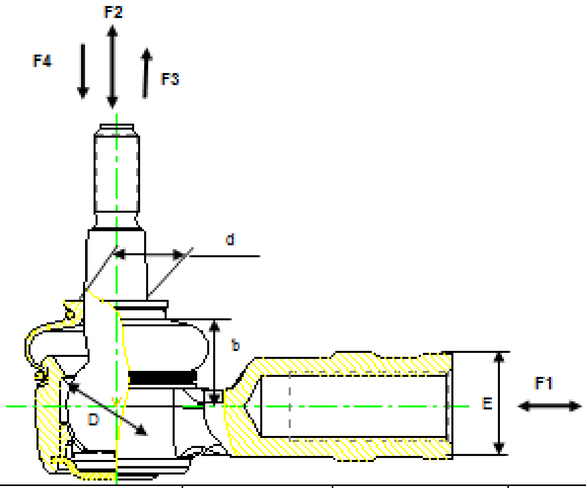


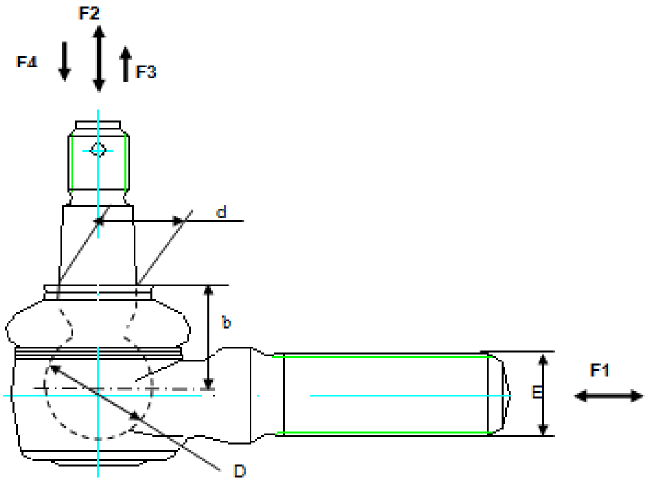
Figure 3.12. The forces applied on ball joints

Table 3.5. Properties of plastic bearing ball joint



Sphere diameter- Ø D	**Ø27**		**Ø 30**		**Ø 35**		**Ø 40**	
Conic diameter d	16	18	16	22	18	24	20	27
Conic height b	26	25	30	26	33	26	40	37
1. Ball pin load (F1 direction) Max static bending load (KN) Material: 41Cr4 (5140) Material: 42CrMo4 (4140)	10 12	15 17	9 10	27 30	12 13	35 40	13 15	35 40
2. Fatigue Strength (Ball Pin) (cycle) Min 200.000 cycle (KN)	±6	±8	±6	±15	±6	±20	±8	±20
3. Radial Elasticity (mm) F1 direction (KN)	≤6 ±3		≤5 ±3		≤4 ±3		≤4 ±3	
4. Axial Elasticity (mm) F2 direction (N)	≤0,25 ±1000		≤0,25 ±1000		≤0,2 ±1000		≤0,2 ±1000	
5.Max Break Moment (Static Moment) (Nm)	10		10		15		18	
6. Dynamic Moment (Nm)	0,5-5		0,5-5		0,5-8		0,5-8	
7. Pull-out Load F3 direction (KN)	30		35		40		45	
8. Push-out Load F4 direction (KN)	20		25		30		35	
9. Wear Test Test load (F1 dir.) f=±2 Hz (KN) Oscillation angle f=±2 Hz (KN) Rotation angle=35° f=±0,5Hz 500.000 cycle	±6		±8		±10		±12	
10. Dust Cover	PUR				Working Temp (°C) -30/+50		Max. Temp. (°C) -40/+80	
10. Dust Cover	POLYCHLROPEN				Working Temp (°C) -40/+90		Max. Temp. (°C) -45/+95	

Table 3.6. Properties of metal bearing ball joint



<table>
<tr><td>Sphere diameter- Ø D</td><td colspan="2">Ø27</td><td colspan="2">Ø 30</td><td colspan="2">Ø 35</td><td colspan="2">Ø 40</td></tr>
<tr><td>Conic diameter d</td><td>18</td><td>20</td><td>20</td><td>21</td><td>22</td><td>28</td><td>26</td><td>32</td></tr>
<tr><td>Conic height b</td><td>20</td><td>24</td><td>27</td><td>22</td><td>28</td><td>32</td><td>35</td><td>35</td></tr>
<tr><td><u>1. Ball pin load</u> (F1 direction)
Max static bending load (KN)
Material: 41Cr4 (5140)
Material: 42CrMo4 (4140)</td><td>22
25</td><td>28
32</td><td>25
30</td><td>35
40</td><td>32
36</td><td>58
65</td><td>42
50</td><td>80
90</td></tr>
<tr><td><u>2. Fatigue Strength</u> (Ball Pin) (cycle)
Min 200.000 cycle (KN)</td><td>±10</td><td colspan="2">±15</td><td colspan="2">±20</td><td colspan="2">±30</td><td>±40</td></tr>
<tr><td><u>3. Radial Elasticity</u> (mm)
F1 direction (KN)</td><td colspan="2">≤0,4
±4</td><td colspan="2">≤0,4
±5</td><td colspan="2">≤0,6
±6</td><td colspan="2">≤0,6
±9</td></tr>
<tr><td><u>4. Axial Elasticity</u> (mm)
F2 direction (N)</td><td colspan="2">≤0,4
±650</td><td colspan="2">≤0,4
±750</td><td colspan="2">≤0,4
±850</td><td colspan="2">≤0,4
±1050</td></tr>
<tr><td>5.Max Break Moment
(Static Moment) (Nm)</td><td colspan="2">10</td><td colspan="2">12</td><td colspan="2">15</td><td colspan="2">18</td></tr>
<tr><td>6. Dynamic Moment (Nm)</td><td colspan="2">1-7,5</td><td colspan="2">1-9</td><td colspan="2">1,5-11</td><td colspan="2">2-14</td></tr>
<tr><td>7. Pull-out Load
F3 direction (KN)</td><td colspan="2">60</td><td colspan="2">80</td><td colspan="2">90</td><td colspan="2">120</td></tr>
<tr><td>8. Push-out Load
F4 direction (KN)</td><td colspan="2">35</td><td colspan="2">50</td><td colspan="2">70</td><td colspan="2">70</td></tr>
<tr><td>9. Wear Test
Test load (F1 dir.) f=±2 Hz (KN)
Oscillation angle f=±2 Hz (KN)
Rotation angle=35° f=±0,5Hz
500.000 cycle</td><td colspan="2">±8</td><td colspan="2">±10</td><td colspan="2">±15</td><td colspan="2">±20</td></tr>
<tr><td rowspan="2">10. Dust Cover</td><td rowspan="2" colspan="4">PUR</td><td colspan="2">Working Temp (°C)</td><td colspan="2">Max. Temp. (°C)</td></tr>
<tr><td colspan="2">-30/+50</td><td colspan="2">-40/+80</td></tr>
<tr><td rowspan="2">10. Dust Cover</td><td rowspan="2" colspan="4">POLYCHLROPEN</td><td colspan="2">Working Temp (°C)</td><td colspan="2">Max. Temp. (°C)</td></tr>
<tr><td colspan="2">-40/+90</td><td colspan="2">-45/+95</td></tr>
</table>

What are the design variables of a ball joint for failure, endurance and comfort? The forces applied on ball joints, the type of tests, basic dimensions and material used in ball joints are given in Table 3.5 and Table 3.6 for both plastic and metal bearing. The tables are summarized from Turkish standards with the standard number of 5476 and 9444.

3.3. Design of Ball Joints for Fatigue Strength

While a vehicle is driven on the road, the steering wheel is turned on left or right, to follow the road for the desired route. The wheels of vehicle under motion moves up and down perpendicularly to road depending on the surface conditions of the road. As a result of these, the direction and amounts of loads on ball joint always fluctuate along the trip which may cause fatigue failure. The amounts of maximum allowable fatigue loads are given in Table 3.5 and Table 3.6 according to type of ball joint.

As seen in Fig. 3.13b, the tapered section of ball joints is fixed to steering arm, subjected to F1, F2 load and moment M. The direction of forces and moments change opposite sides equally so mean stress is zero;

$$\sigma_m = 0 \tag{3.1}$$

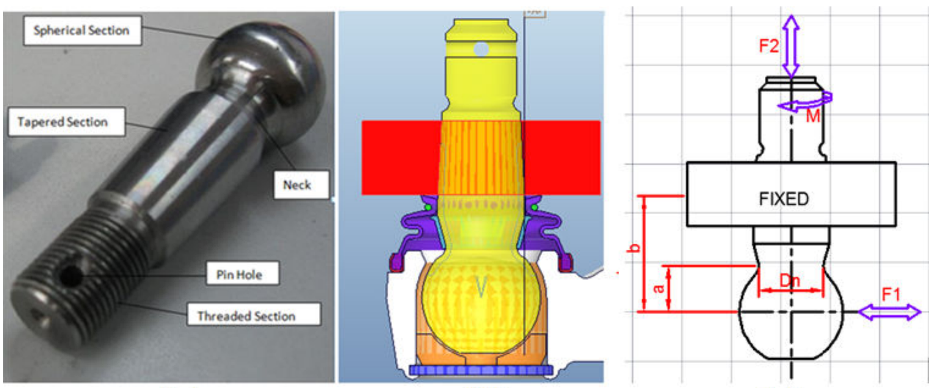


Figure 3.13. (a). Sections of ball joints (b) Ball Joint cross-section (c) Forces on ball pin and critical dimensions

To calculate fatigue strength stress amplitude at critical point must also be found. Experiments and tests shown that ball joints are always broken in neck of ball pin (Fig. 3.13a). Fig. 3.13b shows assembled cross-section of ball joint. The neck point with its dimensions and forces on ball joint can be seen in Fig. 3.13c.

The free body diagram of ball pin are drawn and given in Fig. 3.14 together with forces and moments.

The normal stresses amplitude on neck point is;

$$\sigma_a = \frac{F_2}{A} + \frac{M.(\frac{D_n}{2})}{I} \tag{3.2}$$

where F_2 is normal force at neck point, A is area of neck point, M is bending moment at neck point, D_n is the neck point diameter, I is inertia moment of neck point.

Bending moment at neck point is;

$$M = F_1.a \tag{3.3}$$

where F_1 is the perpendicular force to sphere of ball pin and "a" is distance between sphere center of ball pin and neck point.

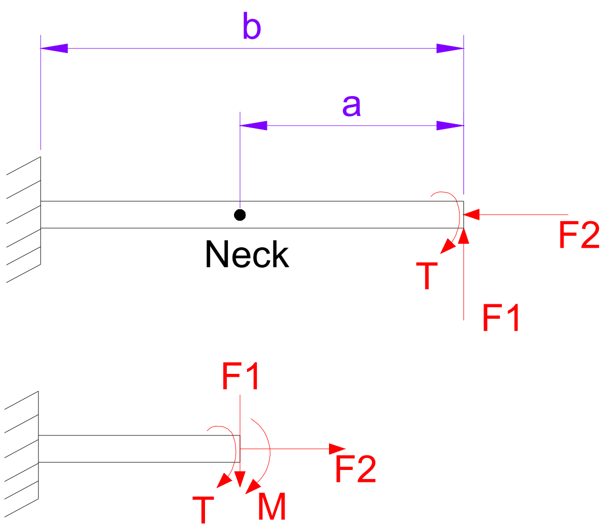


Figure 3.14. Free body diagram of ball pin

Inertia moment at neck point is;

$$I = \frac{\pi . D_n^4}{64} \tag{3. 4}$$

Area at neck point is;

$$A = \frac{\pi . D_n^2}{4} \tag{3.5}$$

Shear stress amplitude on neck point is;

$$\tau_a = \frac{F_1}{A} + \frac{T . (\frac{D_n}{2})}{J} \tag{3.6}$$

where F_1 is the perpendicular force to sphere, A is area of neck point, T is torsion at neck point, D_n is the neck point diameter, J is polar moment of inertia at neck point.

Polar moment of inertia at neck point is;

$$J = \frac{\pi D_n^4}{32} \tag{3.7}$$

Since neck point is subjected to both shear and bending moment, the von Misses Stresses are found to be;

$$\sigma_a' = \sqrt{\sigma_a^2 + 3\tau_a^2} \tag{3.8}$$

Since mean stress at neck point is zero, factor of safety,

$$n = \frac{S_e}{K_f \sigma_a'} \tag{3.9}$$

Where S_e is endurance limit at critical location and given as a function of some modifiers,

$$S_e = k_a . k_b . k_c . k_d . k_e . k_f . S'_e \tag{3.10}$$

Where,

k_a : surface condition modification factor.
k_b : size modification factor.
k_c : load modification factor.
k_d : temperature modification factor
k_e : Reliability factor
k_f: Miscellaneous-effects modification factor
S'_e : Rotary beam test specimen endurance limit

Ball pin material is generally DIN 41Cr4 (AISI 5140) or DIN 42CrMo4 (AISI 5140). After cold forging and quenching and tempering processes ball pins are finished by turning operation.

For 41Cr4 (5140) steel; $S_{ut} = 900\ Mpa$
For 42CrMo4 (4140) steel; $S_{ut} = 1100\ Mpa$

$$k_a = aS_{ut}^b \tag{3.11}$$

where, *a=4.51* and *b= (-0.265)* for machined part

$$k_b = 1.24d^{-0.107} \text{ for } 2.79 \leq d \leq 51\ mm \tag{3.12}$$

$k_c = 1$ for parts subjected to combined load
$k_d = 1$ because of $S_T/S_{RT} = 1$
$k_e = 0.897$ for reliability %90

$$K_f = 1 + \frac{K_t - 1}{1 + \sqrt{a/r}} \tag{3.13}$$

where

$$\sqrt{a} = 0.245799 - 0.307794\,(10^{-2})S_{ut} + 0.150874(10^{-4})S_{ut}^2 - 0.266978(10^{-7})S_{ut}^3 \tag{3.14}$$

All factors are taken from Shigley (9th edition, 2006).

4. RESULTS AND DISCUSSION

As mentioned in the previous (material and method) Chapter, a new approach has been proposed for parametric design of assembled mechanical parts. In this section, the new approach that was developed on a parametric design platform using PTC Pro/Engineer solid part design module, assembly design module, Pro/Program module, Microsoft Excel 2007, has been applied to automotive ball joint design. The results of the implementation of parametric modeling of ball joint assembly and its parts are explained steps by steps in the following sections.

4.1. Parametric Design of Ball Joint Parts

A generic algorithm that was developed for parametric design of mechanical assemblies given in Section 3.1 and Fig. 3.2 has been applied to the parts of the ball joint assembly. As it was mentioned in the related section and in Figure 3.2 the algorithm consists of 20 steps. For Step 1, parameters defined in Table 3.1 have been used to design ball joint parts parametrically. Parts of the ball joints have been designed parametrically following the steps of the algorithm, and the details of the results are explained below.

4.1.1. Housing Design

Ball joints are classified into four types and as it can be seen in Table 3.1, the first four parameters define the types of ball joints. Bearing type, housing type, housing shaft type are related to the housing design. The algorithm given in Fig. 3.2 has been applied to housing design as seen in Fig. 4.1.

Totally 7 features has been designed manually in PTC Pro/Engineer part module for housing. Rules for optional cases are listed below for features of housing and they are shown graphically in Fig.4.1.

Rules for optional cases of housing features:

1. If housing type is oval, use feature #3
2. If housing type is elliptical, use feature #4
3. If bearing type is metal, use feature #5
4. If bearing type is plastic, use feature #6
5. If sphere diameter is 30 or 35 or 40, use feature #7
6. If sphere diameter is 27, use feature #8
7. If housing shaft type is inner thread, use feature #9

Pro/program software were edited to manipulate housing design according to the rules. Finally, dimensions have been defined for each feature.

Feature #1: The feature is used to design housing head. Dimensions of housing head are obviously determined according to sphere diameter.

Feature #2: The feature is used to design housing shaft. Dimensions of housing shaft are determined by the designer in Definition Phase.

Feature #3 and Feature #4: The features are used to design oval or elliptical hole respectively. Dimensions of these features are determined according to tilt angles.

Feature #5 and Feature #6: The features are used to design bearing hole for metal and plastic bearing respectively. Dimensions of bearing holes are obviously determined according to sphere diameter.

Feature #7 and Feature #8: The features are used to design dust boot-housing connection. For feature #7, dimensions are determined according to sphere diameter. Feature #8 is used only when sphere diameter is 27 mm.

Feature #9: The feature is used to design hole for inner thread shaft type. Dimensions of housing shaft hole are determined by the designer in Definition Phase.

After designing features of housing in Pro/Engineer part module manually, a program has been written in Pro/program as it can be seen APPENDIX A to calculate dimensions and to activate or inactivate features according to rules.

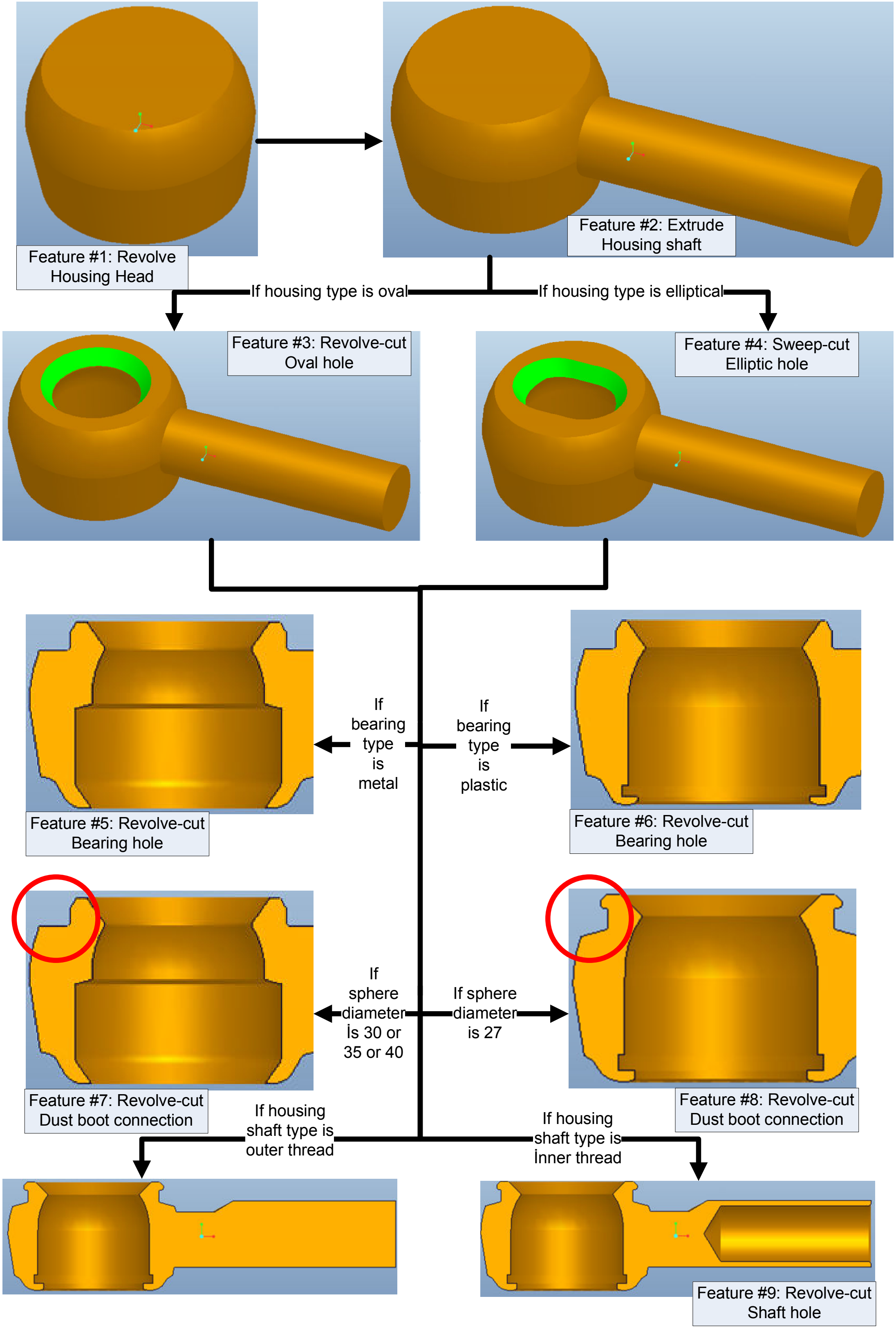


Figure 4.1. Housing parametric design

The knowledge of dimensions has generally been gained from experiences. For example diameter and height of housing head are almost same dimensions in all ball joint manufacturers. The dimensions and its tolerances of bearing holes and dust boot-housing connections are determined according to the ball joint tests as explained in Section 1.2.2.

4.1.2. Ball Pin Design

Only connection type of ball joints is related to ball pin design. A pin hole is designed on ball pin for castle nut type ball joints and an Allen key is designed on ball pin for self-locking type of ball joints as seen in Fig. 4.2.

In order to design of ball pin parametrically, ball pin body has been modeled using revolve feature, thereafter revolve feature, Allen key and extrude feature for pin hole were modeled in Pro/Engineer part module manually. Then, following rules have applied in Pro/program module.

1. If connection type is self-locking, use feature#2
2. If connection type is castle nut, use feature#3

All dimensions of ball pin have been defined in definition phase without neck diameter. In Pro/program software, relations have been established between parameters in definition sheet and ball pin dimensions using the provided an equations in section 3.3.

A fatigue strength design method has been developed for ball pin neck diameter in Section 3.3. Neck diameter can be calculated according to equation developed with the method. But, some information must be taken from specifications defined in Section 3.2. For this purpose, an algorithm has been developed and applied to Pro/program software (see APPENDIX B).

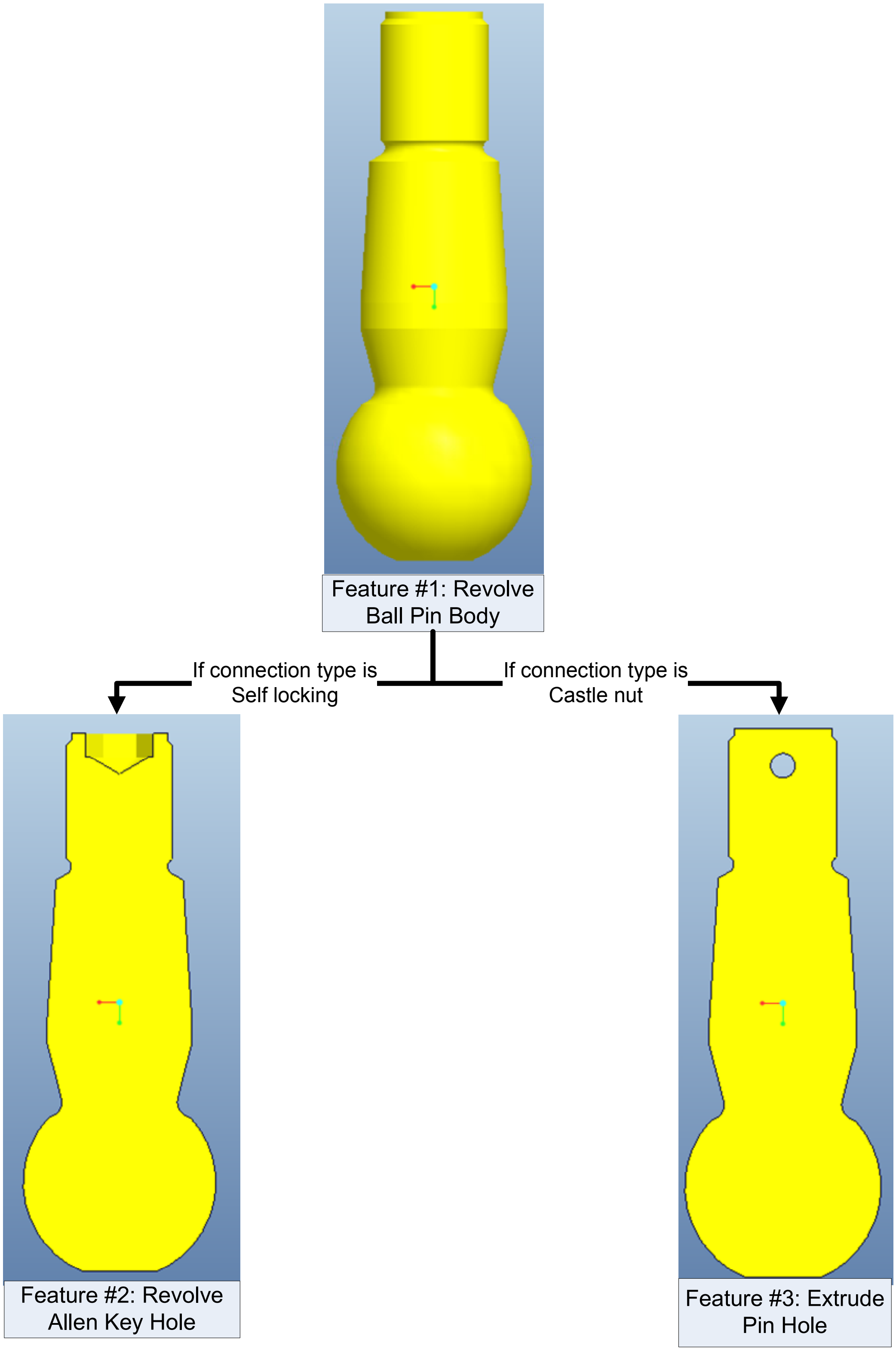


Figure 4.2. Ball pin parametric design

As seen in Fig. 4.3 the dimensions of ball pin has been determined and given a notation to the dimension from bp_1 to bp_18. Then the rules have been defined in Table 4.1 for every dimension.

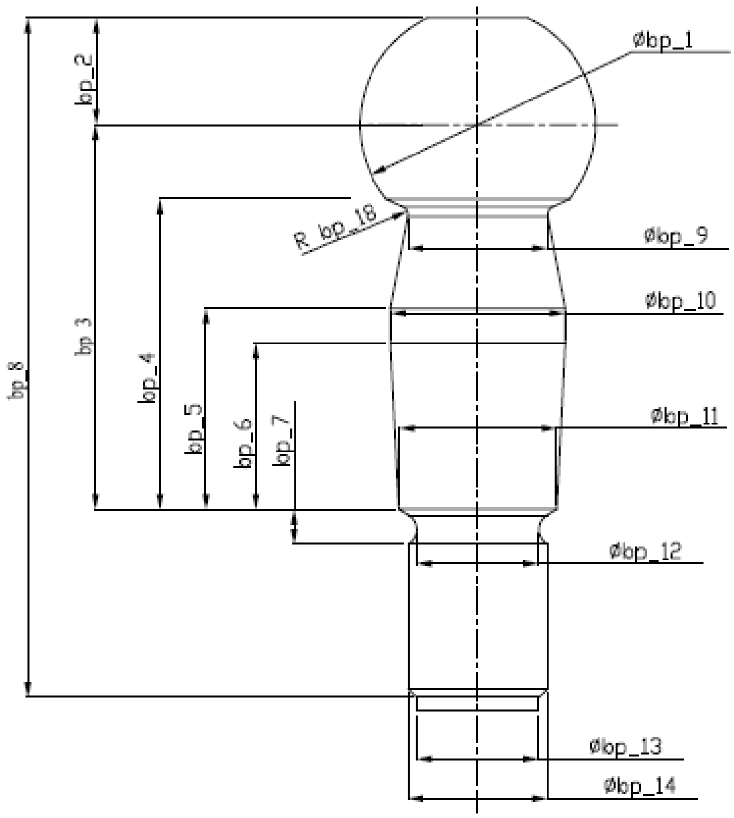


Figure 4.3. Ball joint base geometry and its dimensions

Table 4.1. Rules of dimensions for ball pin base feature

Dimensions	Rules	Explanation
bp_1	bp_1 = Sphere diameter	It must be 27 or 30 or 35 or 40. It is TAKEN FROM EXCELL Definition SHEET .
bp_2	If sphere diameter = 40 then bp_2=18.5 If sphere diameter = 35 then bp_2=16 If sphere diameter = 30 then bp_2=13 If sphere diameter =27 then bp_2=12.2	Automatically calculated.
bp_3	bp_3 = neck length + taper length	User defined. It is read from Definition.

bp_4	If sphere diameter = 40 then
	bp_4 = taper length + 20.5
	If sphere diameter = 35 then
	bp_4 = taper length + 18.5
	If sphere diameter = 30 then
	bp_4 = taper length + 17.5
	If sphere diameter =27 then
	bp_4 = taper length + 16.5
		User defined. It is read from Definition.

bp_5	bp_5=taper lengt+5	User defined. It is read from Definition.
bp_6	bp_6=taper length	User defined. It is read from Definition.
bp_7	If sphere diameter=40 or 35 or 30 then bp_7=4.3 If sphere diameter=27 then bp_7=4	

Table 4.2. (Continue)

bp_8	bp_8= bp_2+neck length + taper length + ball thread length	User defined. It is read from Definition.
bp_9		Calculated according to fatigue design explained in section 3.3
bp_10	bp_10=conic diameter	User defined. It is read from Definition.
bp_11	bp_11=conic diameter - (taper length/conic ratio)	User defined. It is read from Definition.
bp_12	If sphere diameter = 40 then bp_12=21.8 If sphere diameter = 35 then bp_12=17.8 If sphere diameter = 30 then bp_12=15.8 If sphere diameter =27 then bp_12=14	Automatically calculated.
bp_13	bp_13=ball pin thread-2	User defined
bp_14	bp_14=ball pin thread	User defined
bp_18	If sphere diameter = 40 then bp_18=2 If sphere diameter = 35 then bp_18=2 If sphere diameter = 30 then bp_18=2 If sphere diameter = 27 then bp_18=2	

4.1.3. Metal Bearing Design

Revolve feature is used in the parametric design of metal bearing. Two different types of metal bearing geometry exist in the design of ball joints. Geometry changes according to sphere diameter. Geometry #1 shown in Fig 4.4 is used if sphere diameter is 40 and geometry #2 is used if sphere diameter is 27 or 30 or 35.

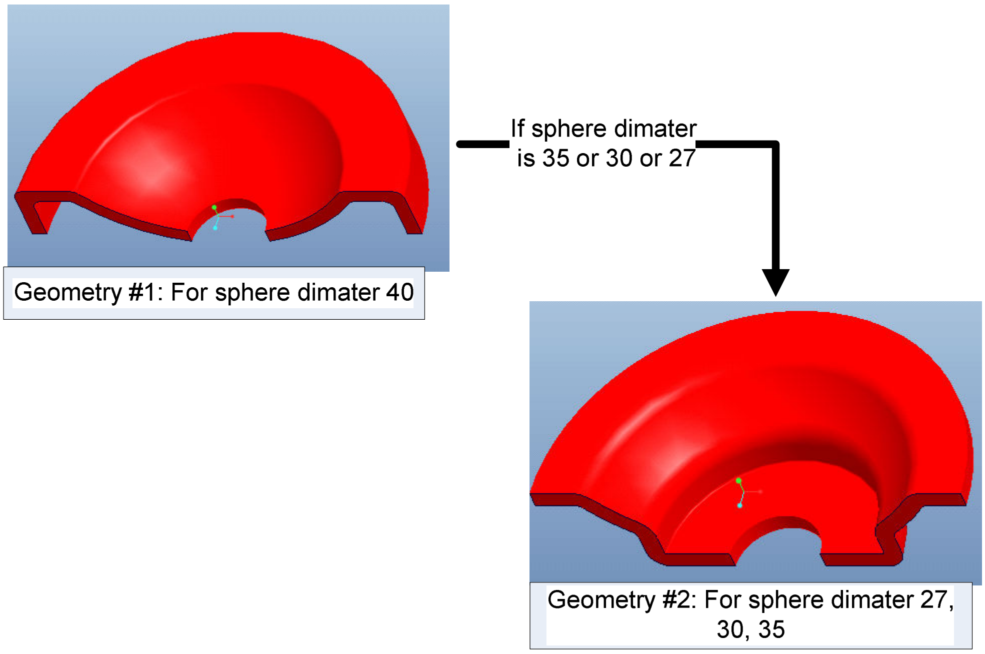


Figure 4.4. Geometries of metal bearing

After metal bearings were designed manually in Pro/engineer part design module with two different geometries, a program has been written in Pro/program module to calculate the dimensions of bearing (see APPENDIX C.)

4.1.4. Plastic Bearing Design

Modeling of plastic bearing in 3D is complex and twelve features are needed to model plastic bearing manually (see Fig. 4.5). In the parametric design of ball joint, only four different plastic bearing are necessary for every sphere diameter type. Therefore four different plastic bearing has been designed manually for each sphere diameter (27, 30, 35, and 40) to reduce design time. In the design of parametric assembly in Pro/program module, a program has been written for choosing convenient plastic bearing as seen in APENDIX J.

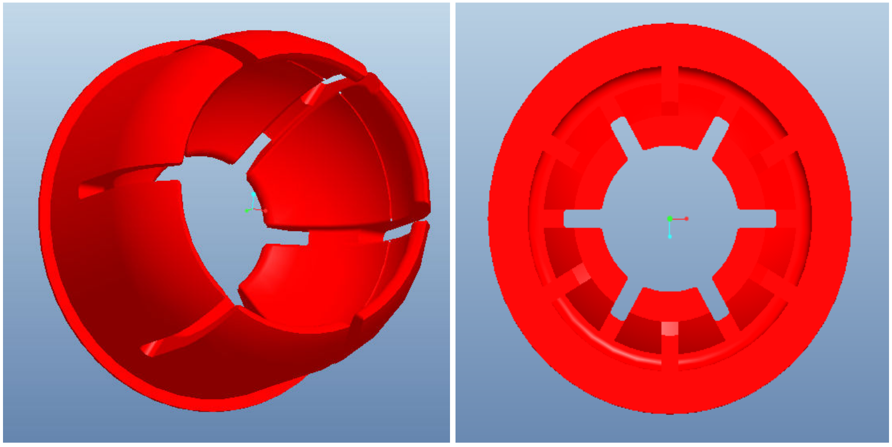


Figure 4.5. Shape of plastic bearing

4.1.5. Spring Design

In the parametric design of spring, two features have been used. First feature is protrusion and it is used to determine spring wire section, dimensions and number of coils (see fig 4.6). Second feature cut is used to make ground ends in both ends of the spring.

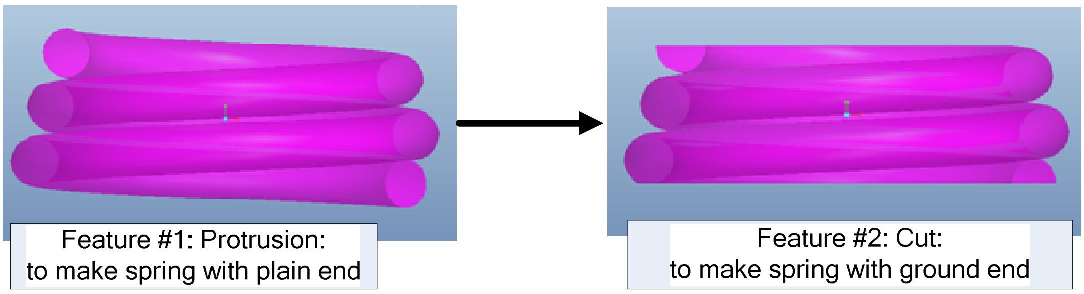
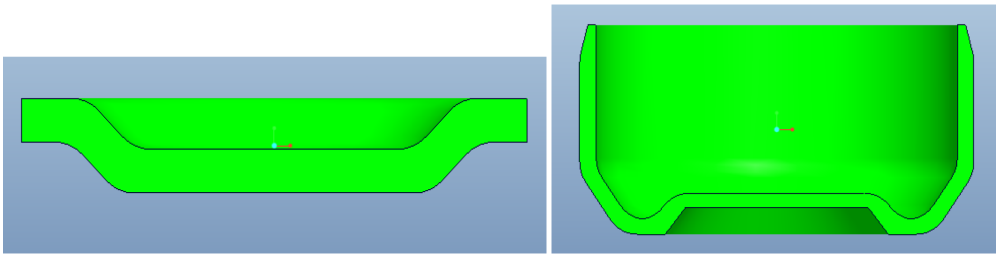


Figure 4.6. The features of spring

Both of the features is used for every spring design, there is no rules for the features of springs. For the calculation of dimensions, a program has been written in Pro/program module and can be seen in APPENDIX D.

4.1.6. End Cap Design

Revolve feature is used in the parametric design of the end cap. Two different types of end cap geometry are used in the design of ball joints. Geometry changes according to sphere diameter and bearing type. Geometry shown in Fig 4.7(a) is used if sphere diameter is 27, 30 or 35 for both metal and plastic bearing and plastic bearing with 40 sphere diameter. Geometry shown in Fig. 4.7(b) is used if sphere bearing is metal with 40 sphere diameter.



(a) (b)

Figure 4.7. Geometries of end cap

End cap in Fig.4.7(a) has been modeled with one revolve feature. A program has been written in Pro/program module to calculate dimensions as seen in APPENDIX E.

End cap in Fig.4.7(b) has been modeled with one revolve feature. Since metal bearing is used only with 40 sphere diameter there is no need to programming.

4.1.7. Clamping Ring Sheet Design

In the parametric design of clamping ring sheet only one feature has been used (see Fig. 4.8). However, a program has been written in Pro/program module to calculate dimensions as seen in APPENDIX F.

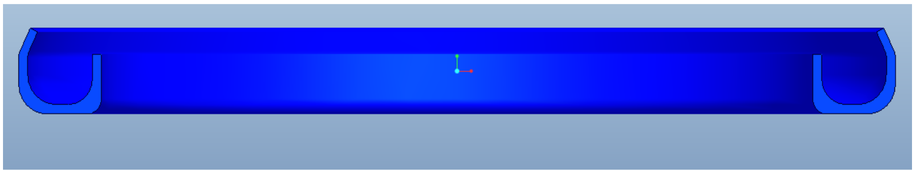


Figure 4.8. Feature of clamping ring sheet

4.1.8. Clamping Ring Wire Design

Two different types of clamping ring wire are used in ball joints. They are called as clamping ring wire up and down as seen in Fig. 4.9.

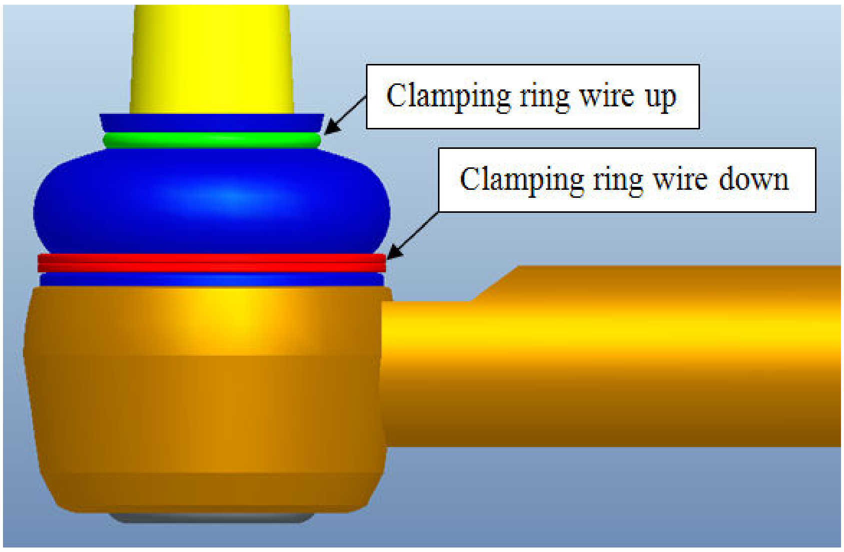


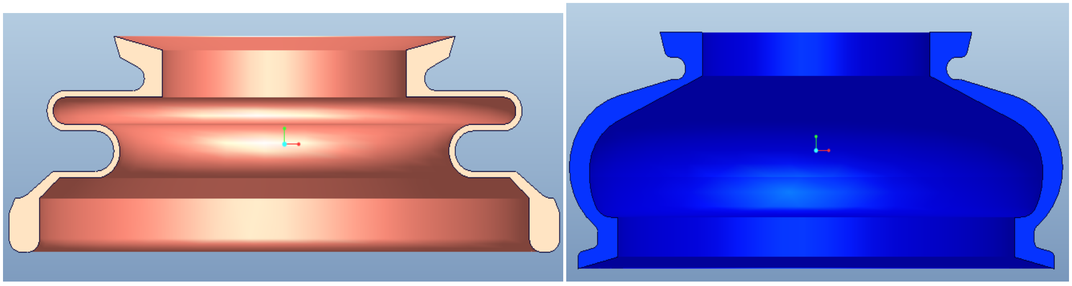
Figure 4.9. Clamping ring up and down

Clamping ring down is used only for 27 sphere diameter and modeled with only one feature. Therefore, there is no need to programming.

Clamping ring up is used for all type of ball joints and modeled with one feature-revolve. A program has been written in Pro/program module to calculate dimensions as seen in APPENDIX G.

4.1.9. Dust Boot Design

Revolve feature is used in the parametric design of dust boot. Two different types of dust boot geometry is used in the design of ball joints. Geometry changes according to sphere diameter. Geometry shown in Fig 4.10(a) is used if sphere diameter is 30, 35 or 40 mm and geometry shown in Fig. 4.10(b) is used if sphere diameter is 27.



(a) (b)

Figure 4.10. Geometries of dust boot

Dust boot for 30, 35 and 40 mm sphere diameter has been modeled with one revolve feature. A program has been written in Pro/program module to calculate dimensions as seen in APPENDIX H.

Dust boot for 27 sphere diameter has been modeled with two revolve feature. A program has been written in Pro/program module to calculate dimensions as seen in APPENDIX I.

4.2. Parametric Design of Ball Joint Assembly

The algorithm developed for parametric design in section 3.1 has been applied to ball joints assembly from Step 12 to Step 20. For assembly, the parts designed parametrically in section 4.1 have been used. As it can be seen in Fig. 4.11, ball pin assembly design begins with Step 12, locating (putting) base part (housing) and continue Step 13 to add the remaining parts. All parts are added manually in Pro/engineer assembly design module and after that, rules are determined (Step 14) for assembly whether the part is used or not. If the part is used, it is activated using pro/program module and logical statements in section 3.1.3.

In the parametric assembly design, parts are activated according to "bearing type and "sphere diameter". The logical statements can be shown in Fig. 4.11.

1. If bearing type is metal, metal bearing, spring, end cap for metal bearing is activated.
2. If bearing type is plastic, plastic bearing and end cap for plastic bearing is activated.
3. If sphere diameter is 27, first dust boot is added and then clamping ring spring added.
4. If sphere diameter is 30, 35 or 40 mm, first clamping ring wire is added and then the dust boot is added.

After parts of ball joint assembled in Pro/engineer assembly module manually a program has been written in Pro/program module to apply rules (logical statements) for ball joint assembly (see APPENDIX J).

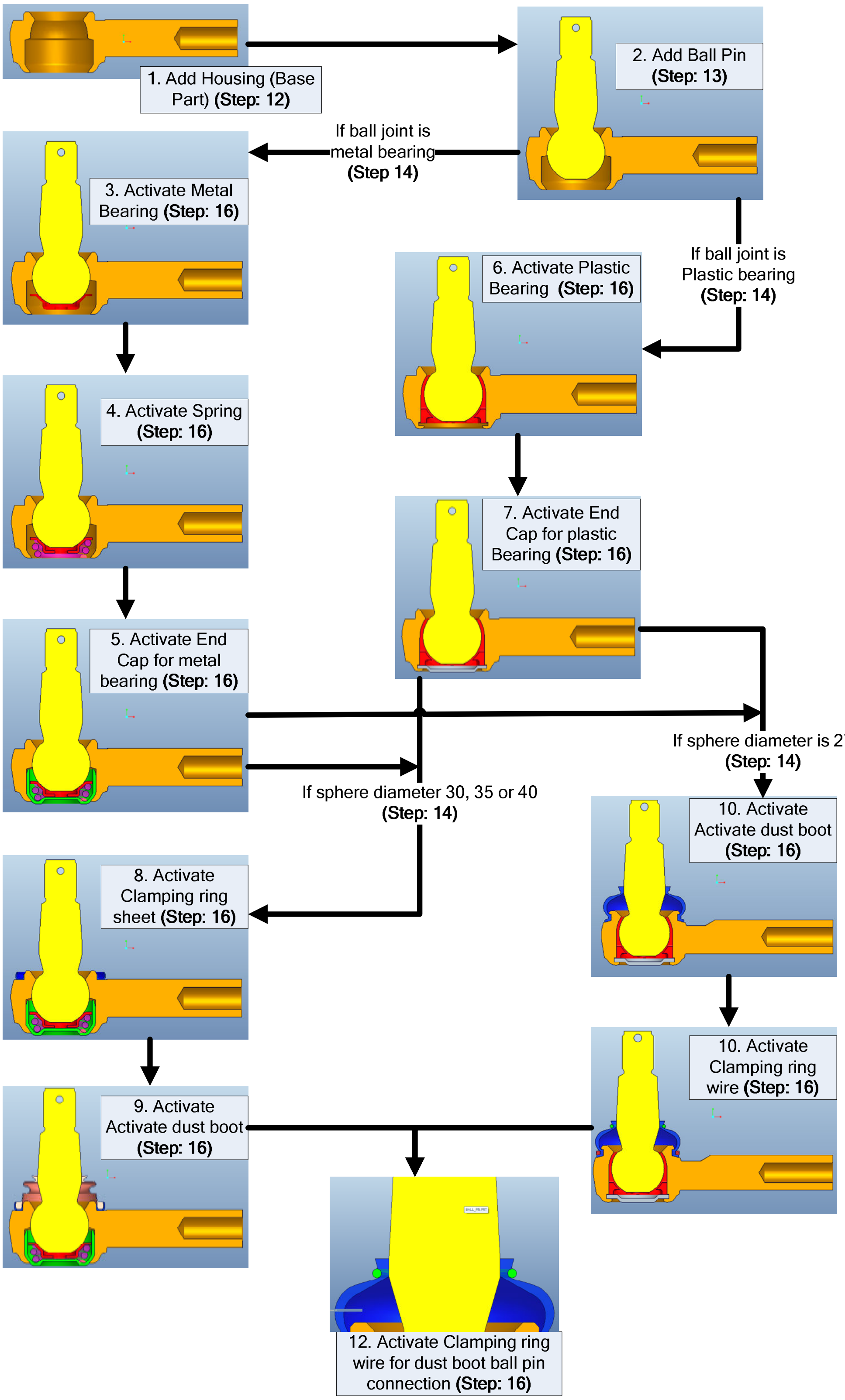


Figure 4.11. Parametric ball joint assembly design

4.3. Validation of Parametric Design of Ball Joint

After the parametric design of ball joint has been completed, the parametric design platform has been tested with following examples to validate the design. Tests have been applied for;

- Types of ball joints
- Dimensions
- Design for fatigue strength

In the tests, firstly, last ball pin design has been prepared in the screen, and then parameters were was changed randomly. Parameters can be changed in definition sheet or Pro/engineer program itself. Both of them are used in tests. First, types of ball joint were tested.

Bearing type test: As known, two types of bearing can be used in ball joints. Ball joint had a metal bearing in Figure 4.12. When bearing type was changed as "plastic" the bearing type was automatically changed to plastic bearing by the platform (see fig. 4.13).

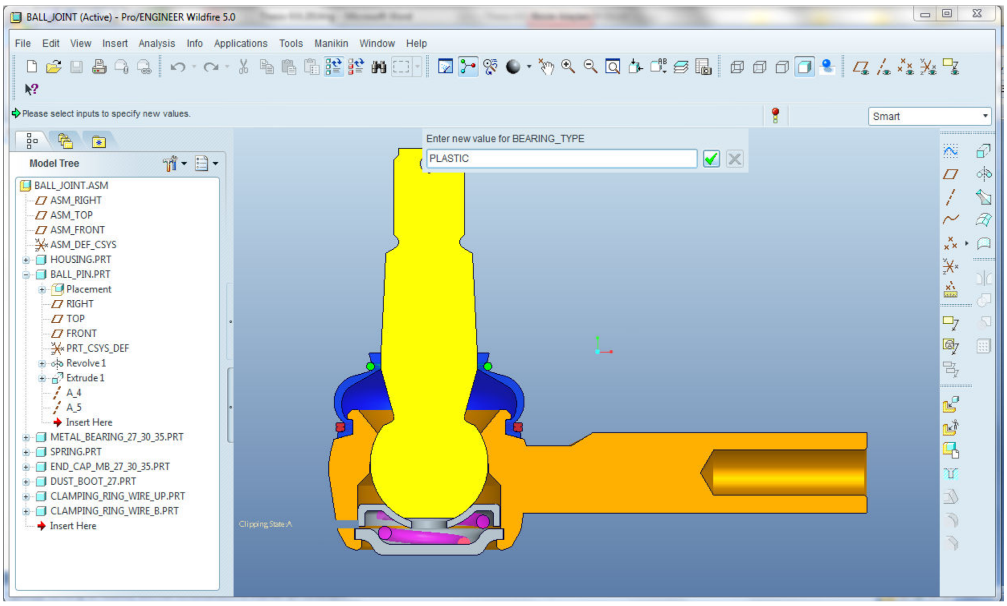


Figure 4.12. Metal Bearing ball joint

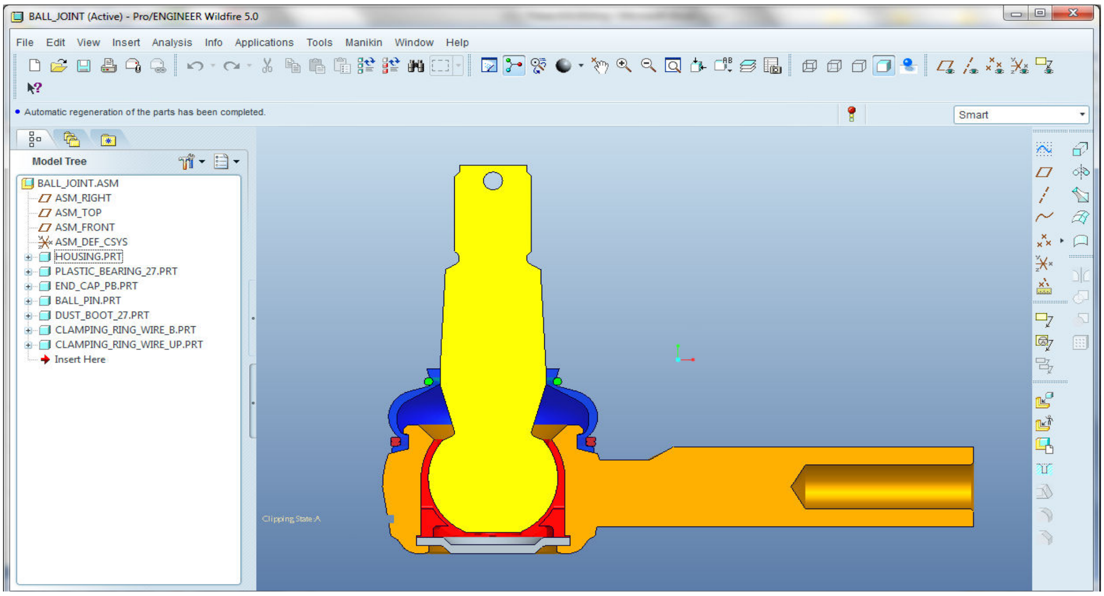


Figure 4.13. Transformed ball joint into plastic bearing

Connection type test: Ball joint can be assembled with castle nut or self-locking nut. In the definition sheet of parametric design platform connection type has been changed from castle nut to self-locking (see Fig. 4.14), the parameter was processed in Pro/program and 3D model of ball joint was transformed from castle nut (see Fig. 4.15) to self-locking (see Fig. 4.16). As it can be seen in the figures 3D model has information text showing ball joint types, dust boot material and sphere diameter.

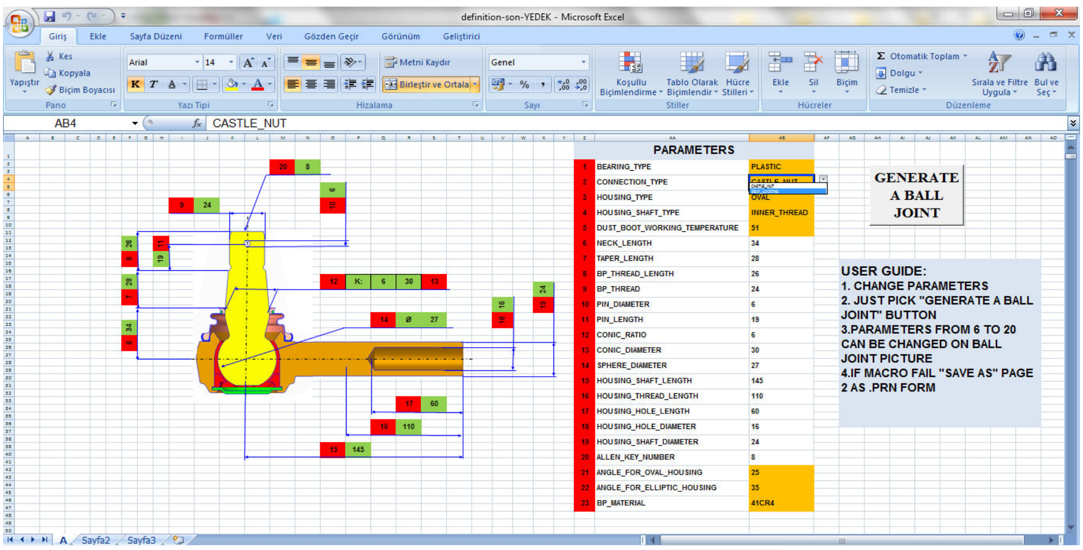


Figure 4.14. Definition sheet of parametric design platform

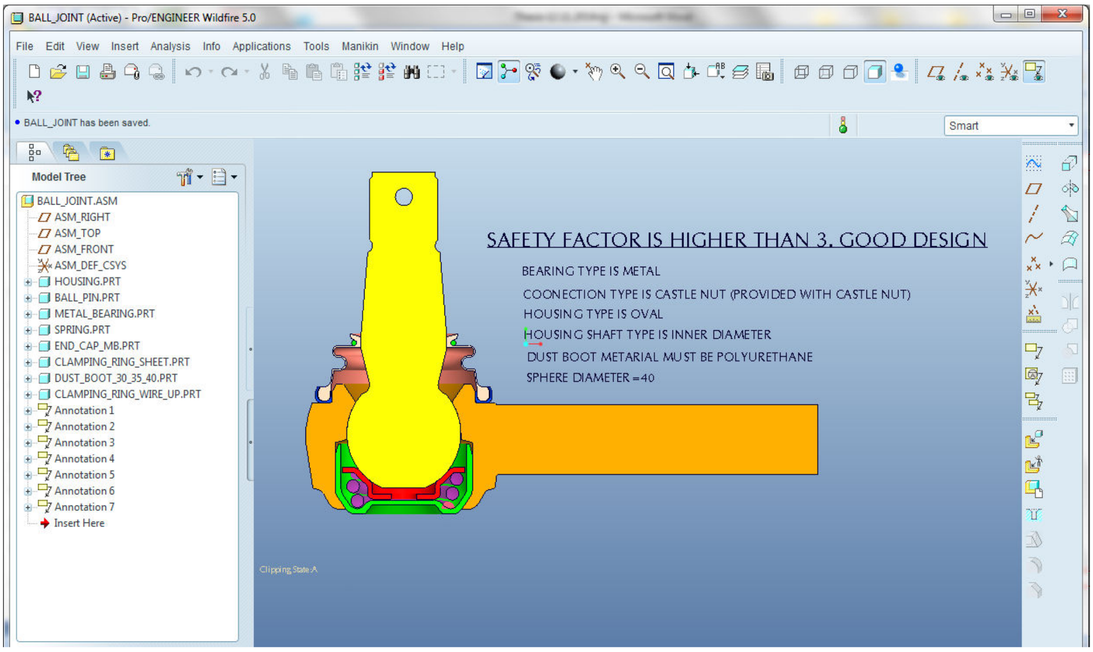


Figure 4.15. Ball joint connection type is castle nut

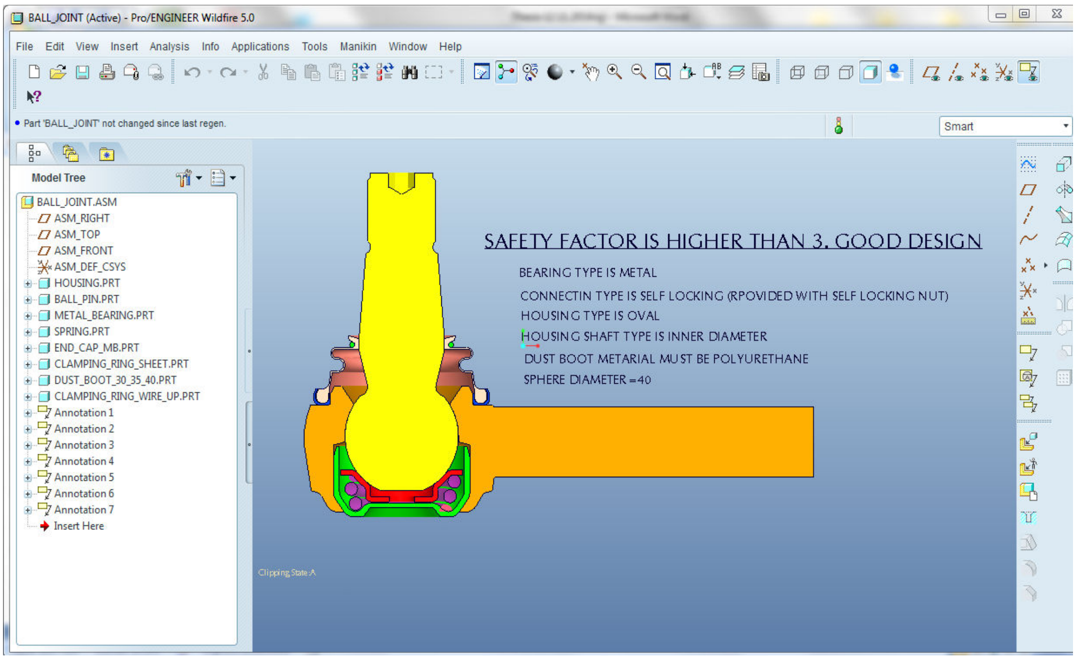


Figure 4.16. Ball joint connection type is self locking (transformed with the platform)

Housing type test: The housing type has also been changed in definition sheet and program has changed housing type in 3D model from oval to elliptical successfully. Transformation cannot be seen in ball joint assembly, it can be seen in housing (see Fig. 4.17 and 4.18)

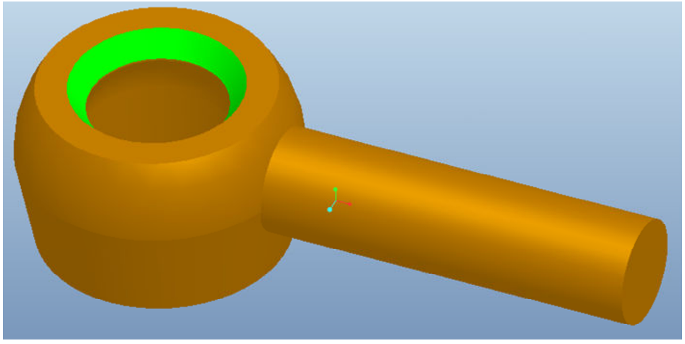


Figure 4.17. Oval housing (before transformation)

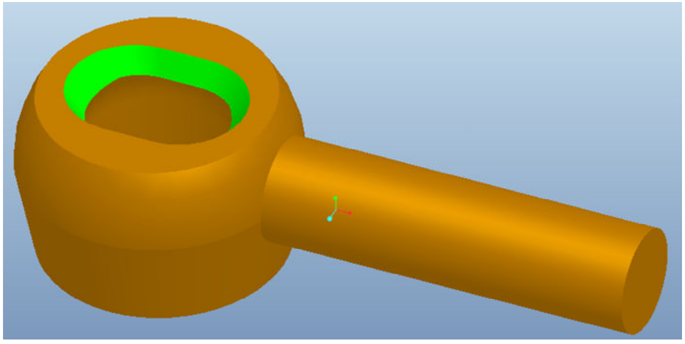


Figure 4.18. Elliptical housing (after transformation)

Housing shaft type test: Housing shaft can be inner threat or outer threat to assembly vehicles steering system. In the parametric design platform, definition sheet housing shaft type has been changed from inner threat to outer threat and in 3D model the type of ball joint changed successfully (see Fig. 19 and Fig. 20).

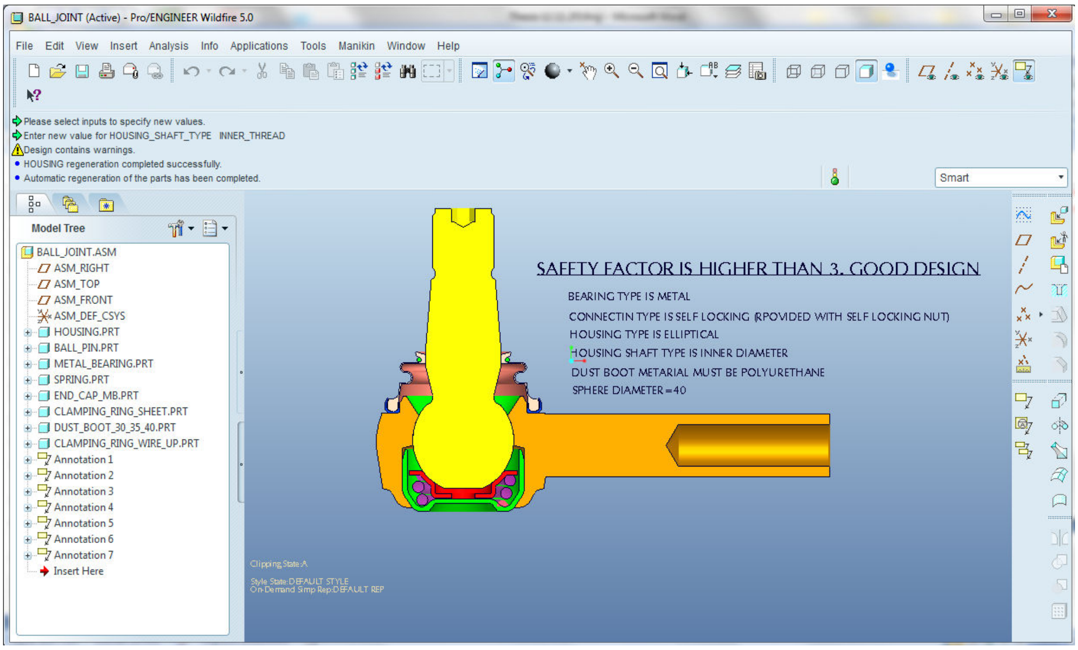


Figure 4.19. Ball joint type is inner thread (before transformation)

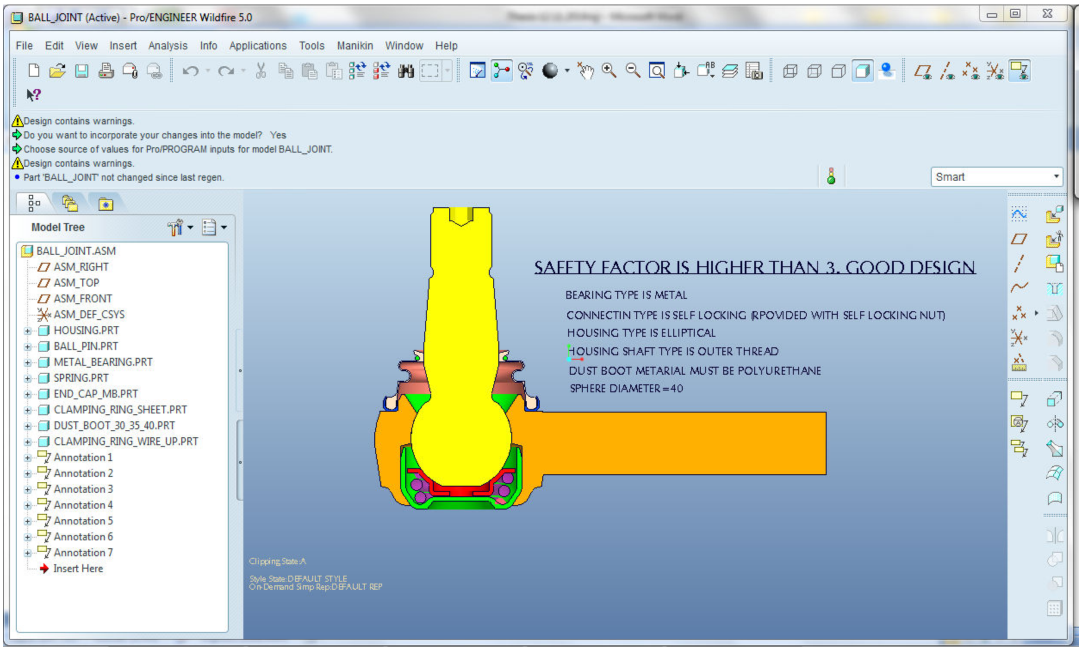


Figure 4.20. Ball joint type is outer thread (after transformation)

Dimensional Tests: 18 different dimensional parameters are available in the design of ball joint as it can be seen parameter list in Table 3.1 (Parameter number 5-22). Each dimension has been tested.

1. Parameter 5: Dust boot working temperature

As it can be seen in Table 3.5 and 3.6, PUR (polyurethane) and POLYCHLROPEN are dust boot materials which are used in ball joints and choice of dust boot material depends on working temperature of ball joint. PUR is used between -30 °C and +50 °C, POLYCHLROPEN is used -40 °C and +90 °C. Therefore, the following rules have been applied to Pro/program module;

- If dust boot working temperature is between -30 °C and +50 °C then, dust boot material is polyurethane.
- If dust boot working temperature is smaller than -30 °C or higher than +50 °C then, dust boot material is POLYCHLROPEN.

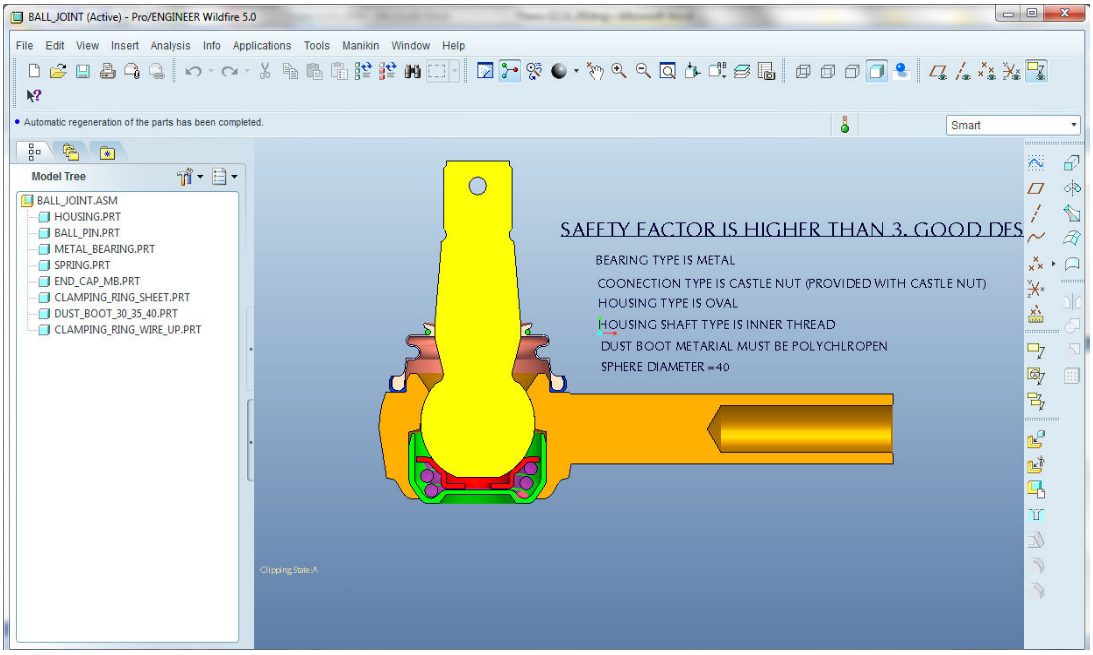


Figure 4.21. Ball joint model with dust boot material polyurethane

A text can be seen in 3D model of ball joint to demonstrate dust boot material (Fig. 4.21). In the figure dust boot material was polyurethane and working temperature was +49 °C. When working temperature is changed to +60 °C or -40 °C dust boot material was changed as POLYCHLROPEN.

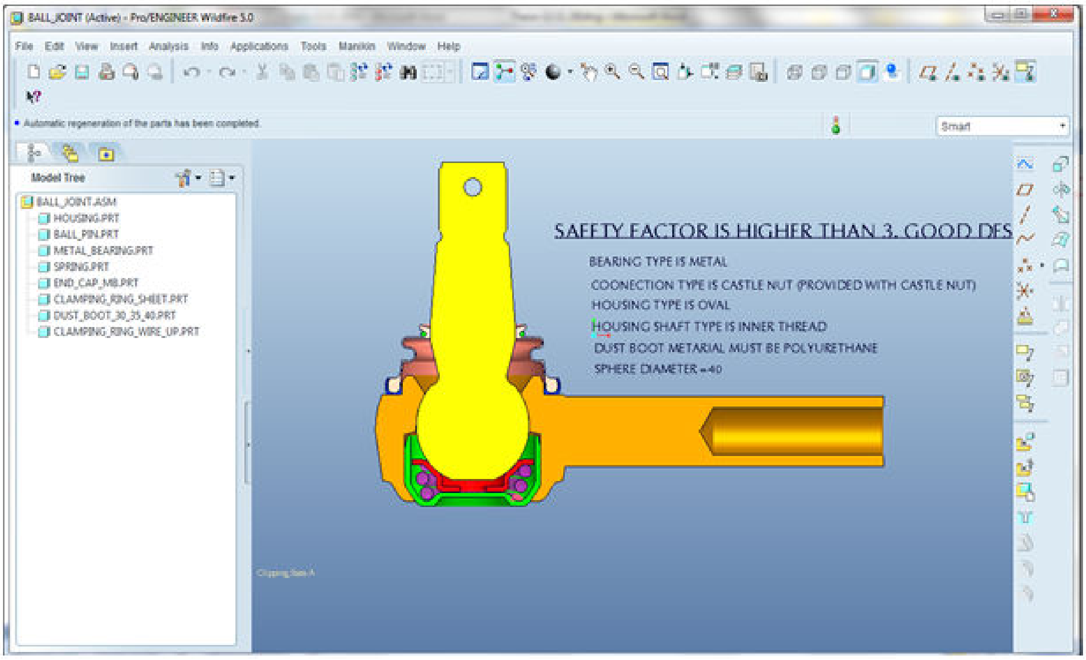


Figure 4.22. Ball joint model with dust boot material POLYCHLROPEN

2. Parameter 6-22: Dimensions of ball joint

First Ø27 mm, Ø30 mm, Ø35 mm and Ø40 mm sphere diameter sheets have been prepared (Fig. 4.23, 4.26, 4.29 and 4.32). Then the file of definition sheets have been saved in *.PRN form. After that the *.PRN file has been read by Pro/Engineer Pro/program module and then 3D model and 2D drawing have been generated (Fig. 4.24 and 4.25 for Ø27 mm sphere diameter, Fig. 4.27 and 4.28 for Ø 30 mm sphere diameter, Fig. 4.30 and 4.31 for Ø 35 mm sphere diameter, Fig. 4.33 and 4.34 for Ø 40 mm sphere diameter). Dimensions in 2D drawing and definition sheet have been compared. After comparison, it is validated that dimensions in the definitions sheets were same as in 2D drawing.

Consequently it is proven that the ball joint can be successfully generated by *"parametric design platform"*.

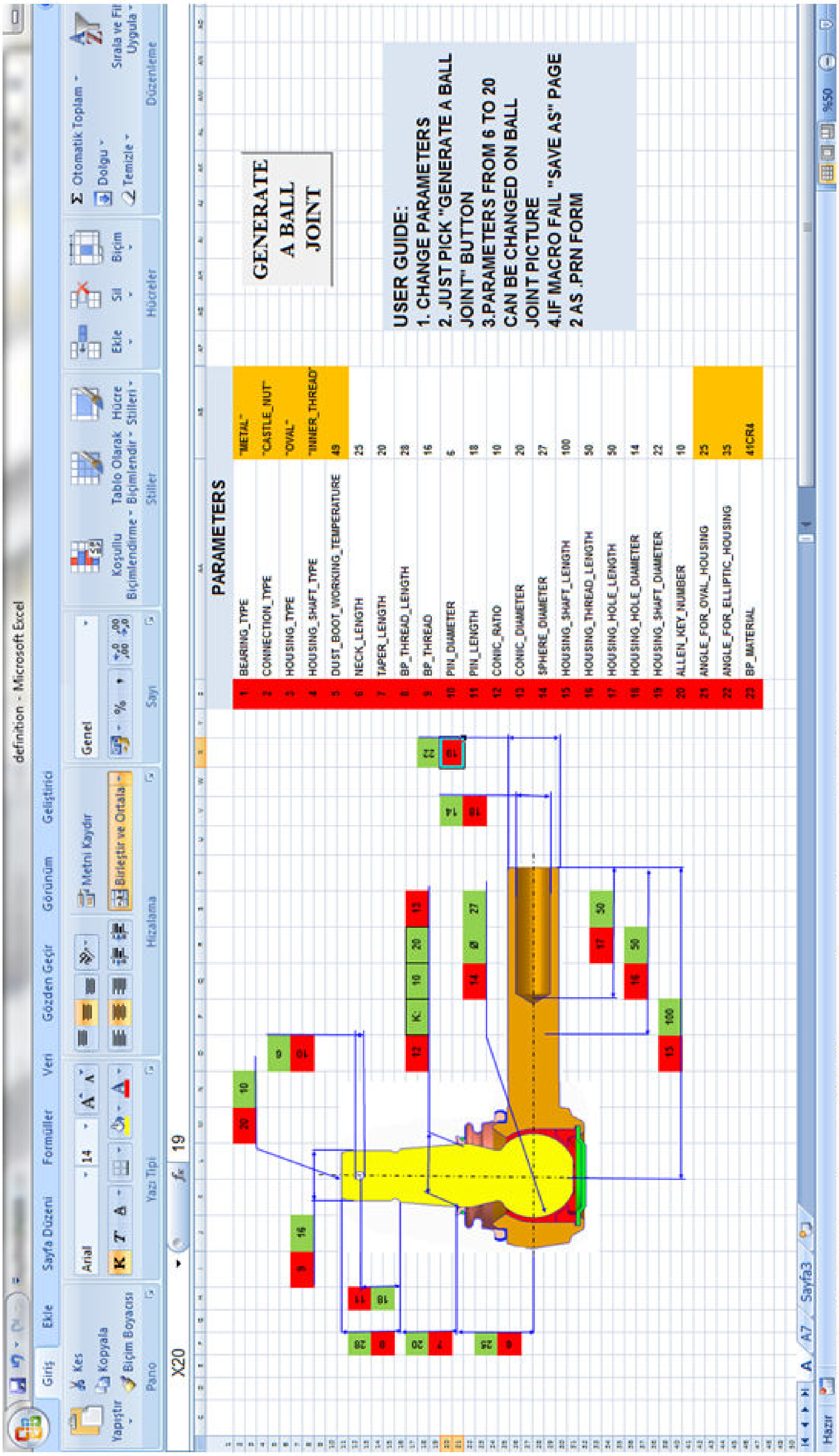


Figure 4.23. Definition sheet of Ø27 mm sphere diameter ball joint

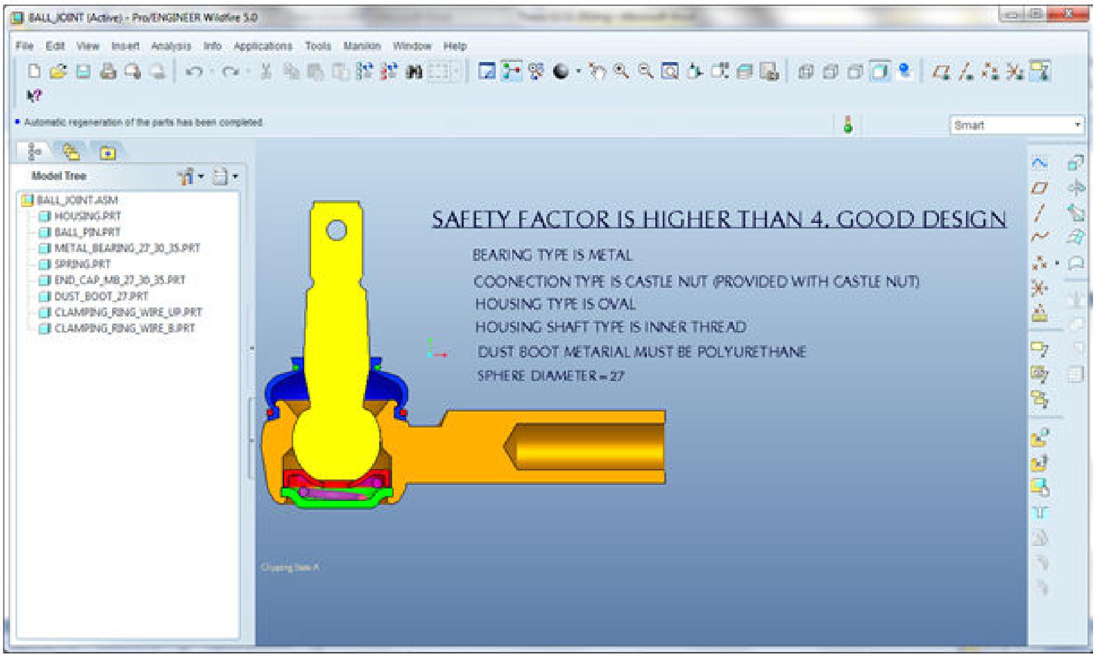


Figure 4.24. Generated 3D model of ball joint from definition sheet of Ø27 mm sphere diameter ball joint

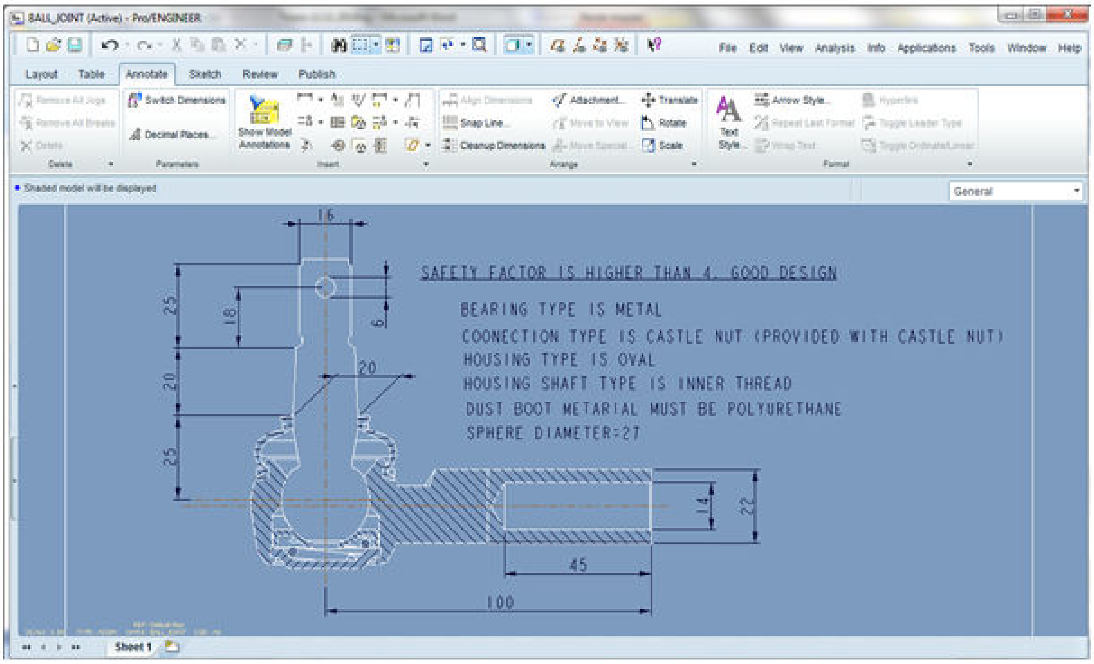


Figure 4.25. Generated 2D drawing of ball joint from definition sheet of Ø27 mm sphere diameter ball joint

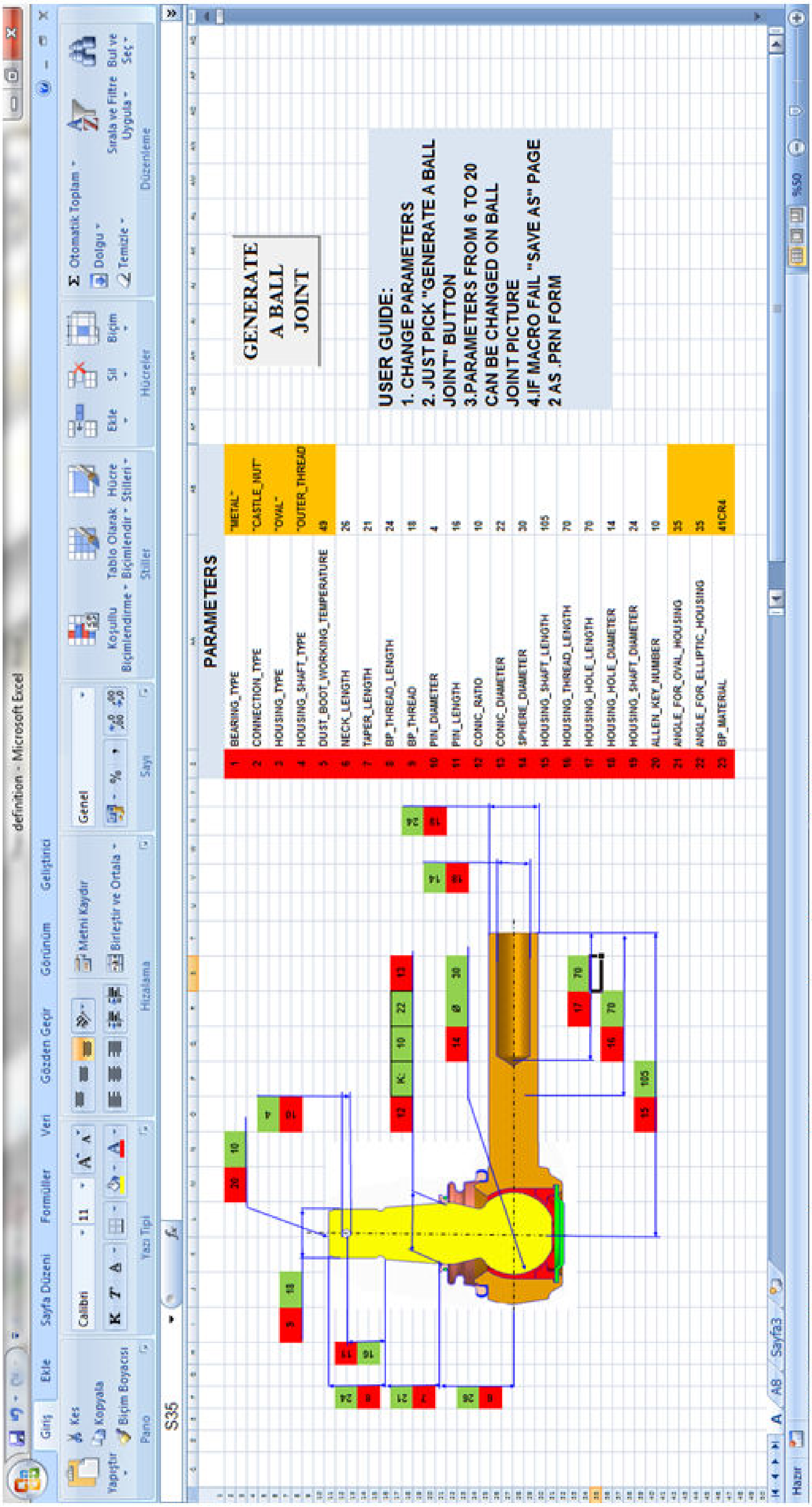


Figure 4.26. Definition sheet of Ø30 mm sphere diameter ball joint

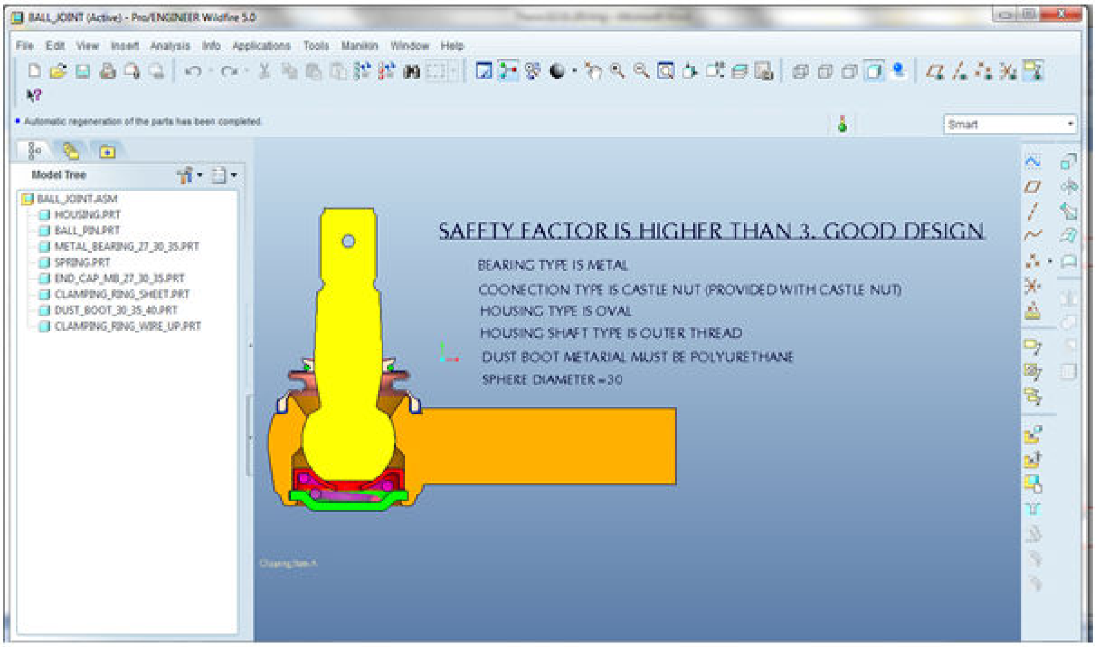


Figure 4.27. Generated 3D model of ball joint from definition sheet of Ø30 mm sphere diameter ball joint

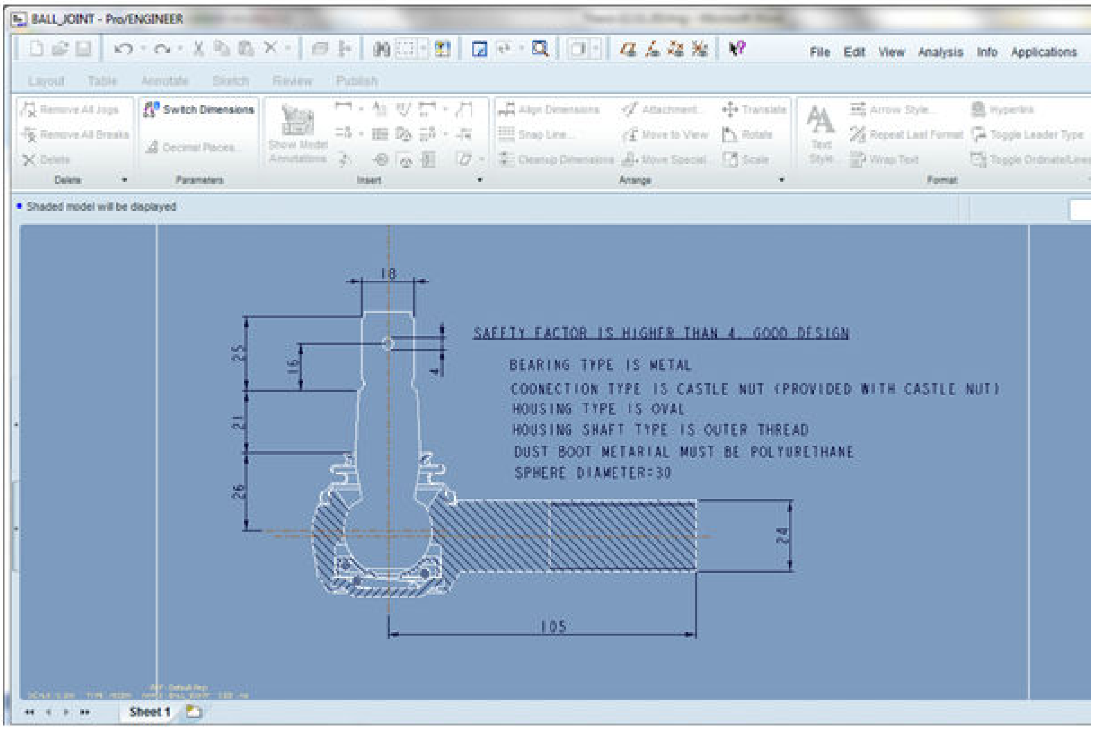


Figure 4.28. Generated 2D drawing of ball joint from definition sheet of Ø30 mm sphere diameter ball joint

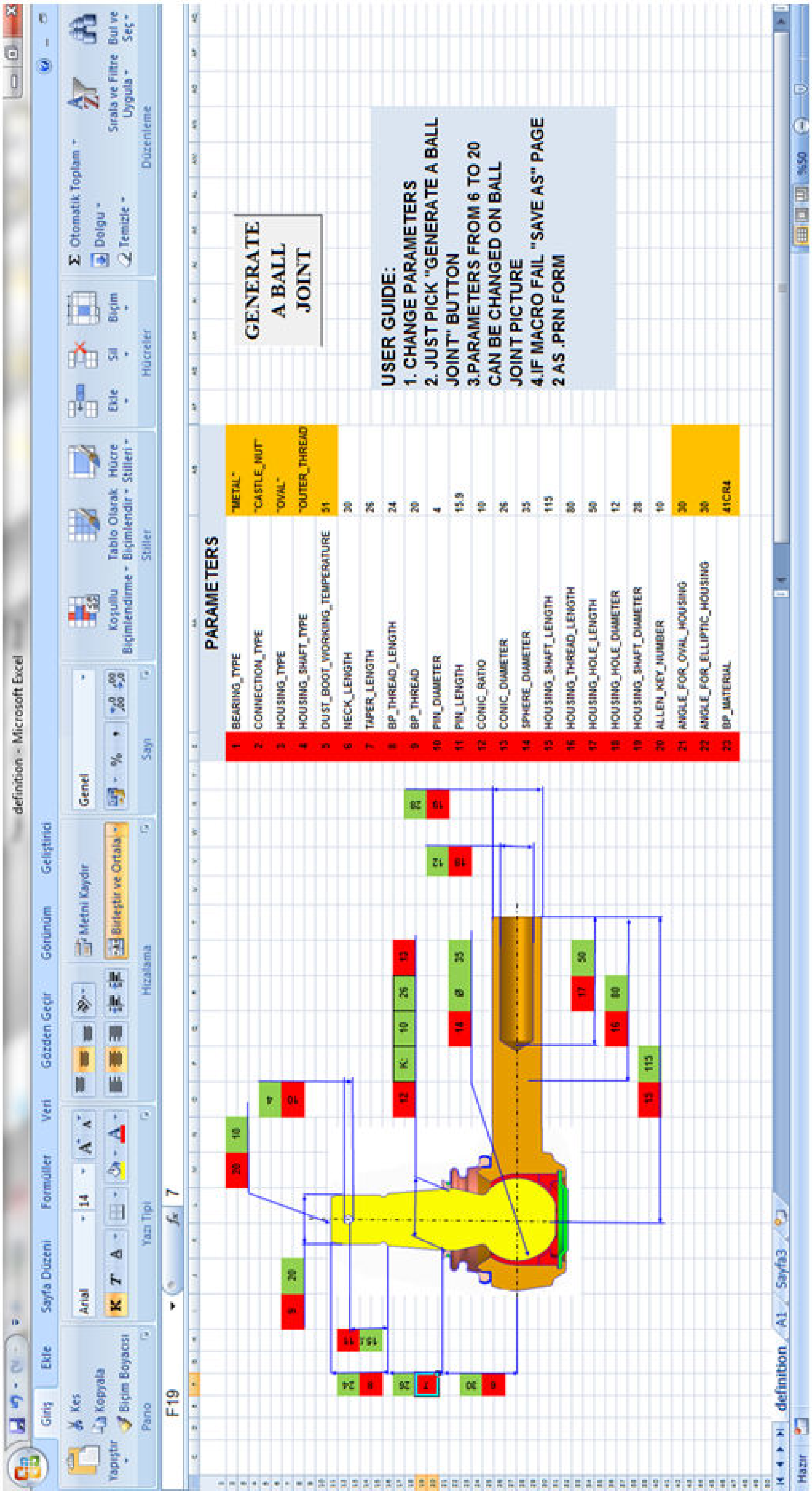


Figure 4.29. Definition sheet of Ø35 mm sphere diameter ball joint

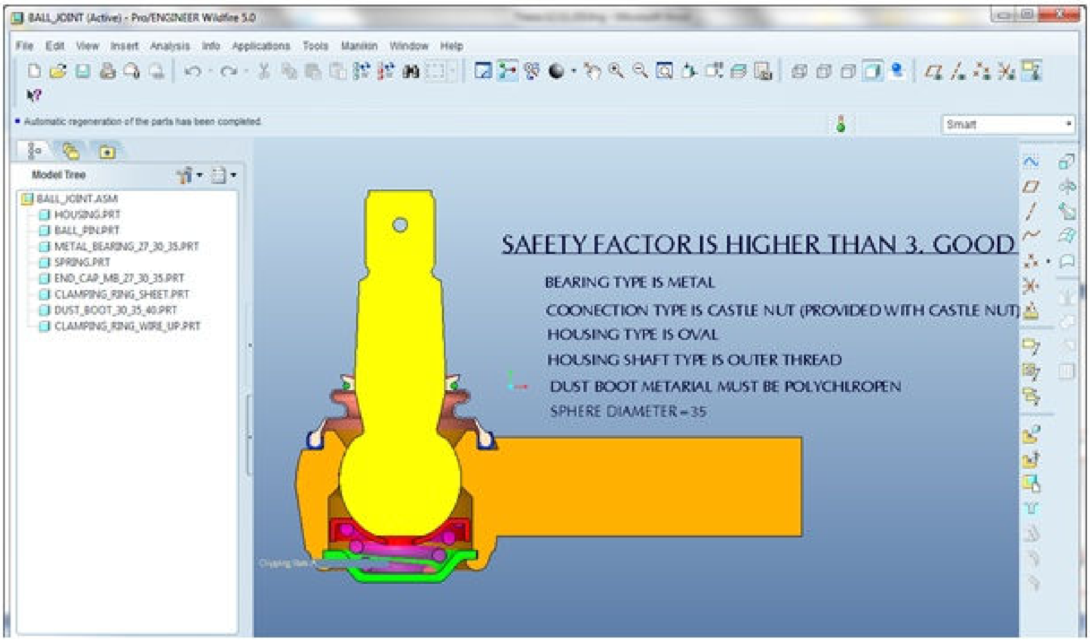


Figure 4.30. Generated 3D model of ball joint from definition sheet of Ø35 mm sphere diameter ball joint

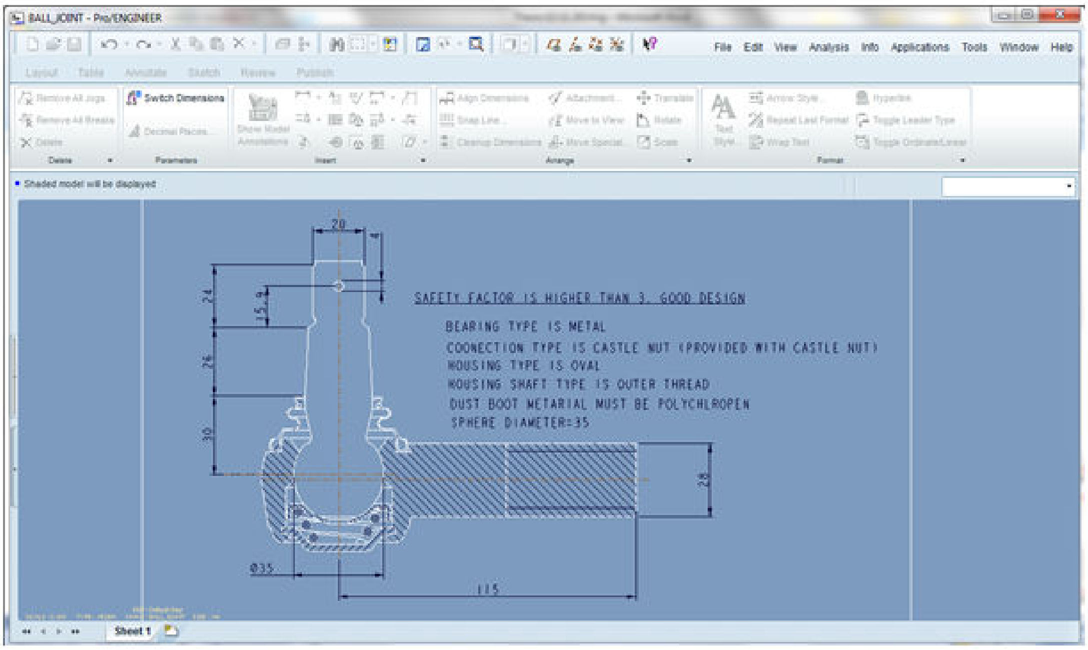


Figure 4.31. Generated 2D drawing of ball joint from definition sheet of Ø35 mm sphere diameter ball joint

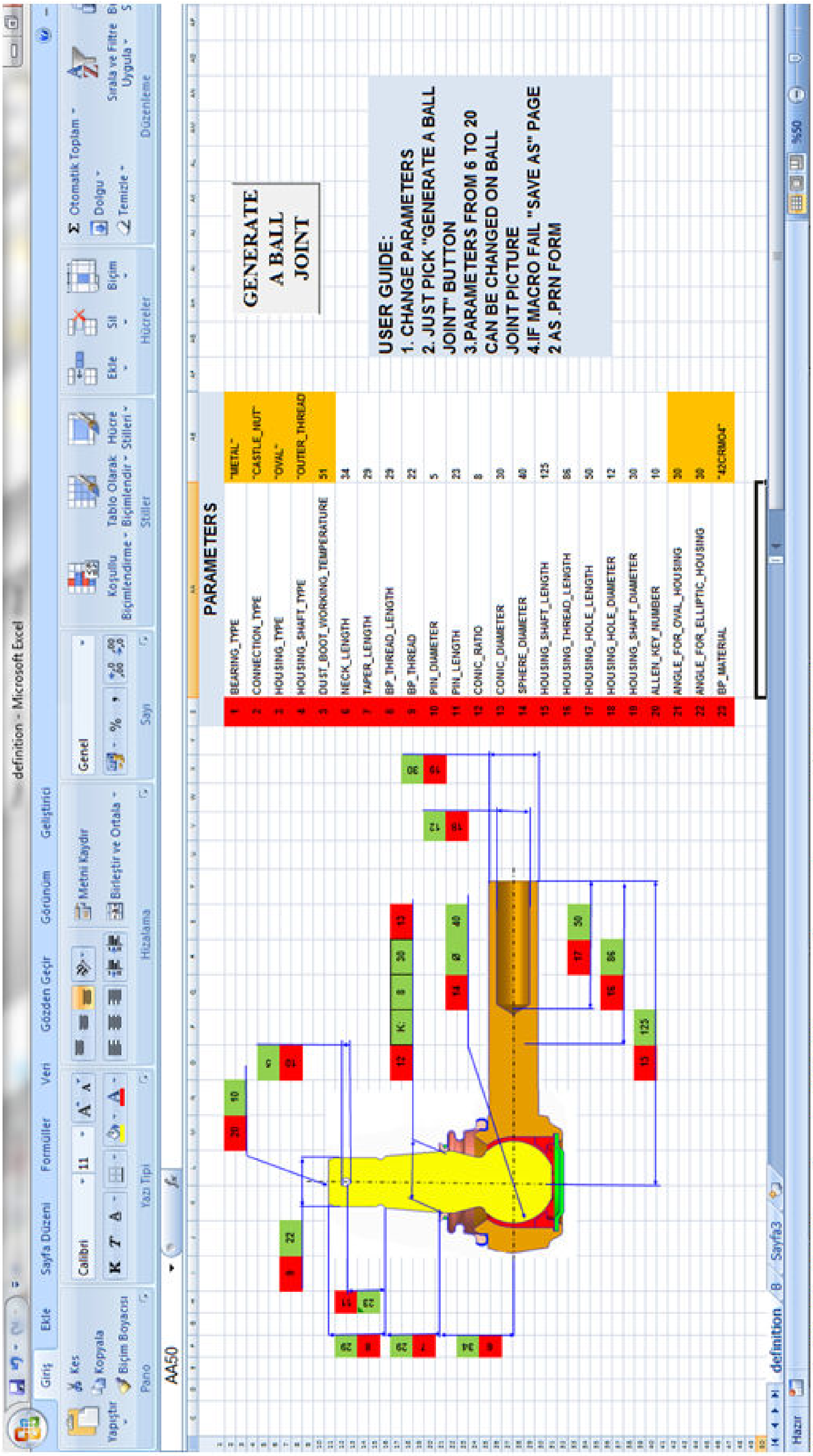


Figure 4.32. Definition sheet of Ø40 mm sphere diameter ball joint

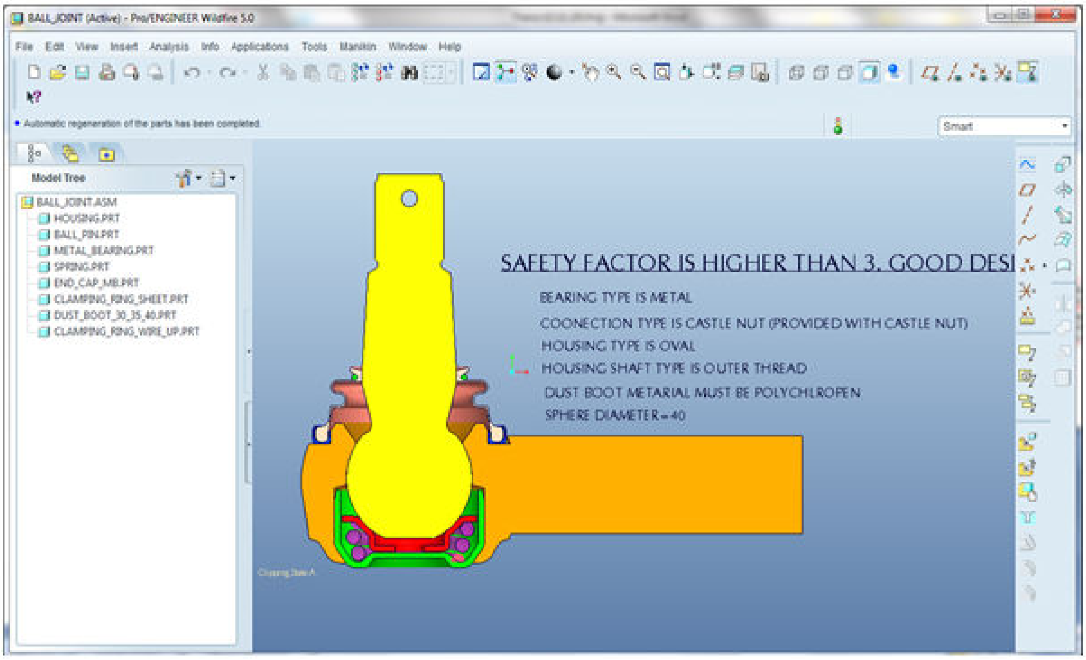


Figure 4.33. Generated 3D model of ball joint from definition sheet of Ø40 mm sphere diameter ball joint

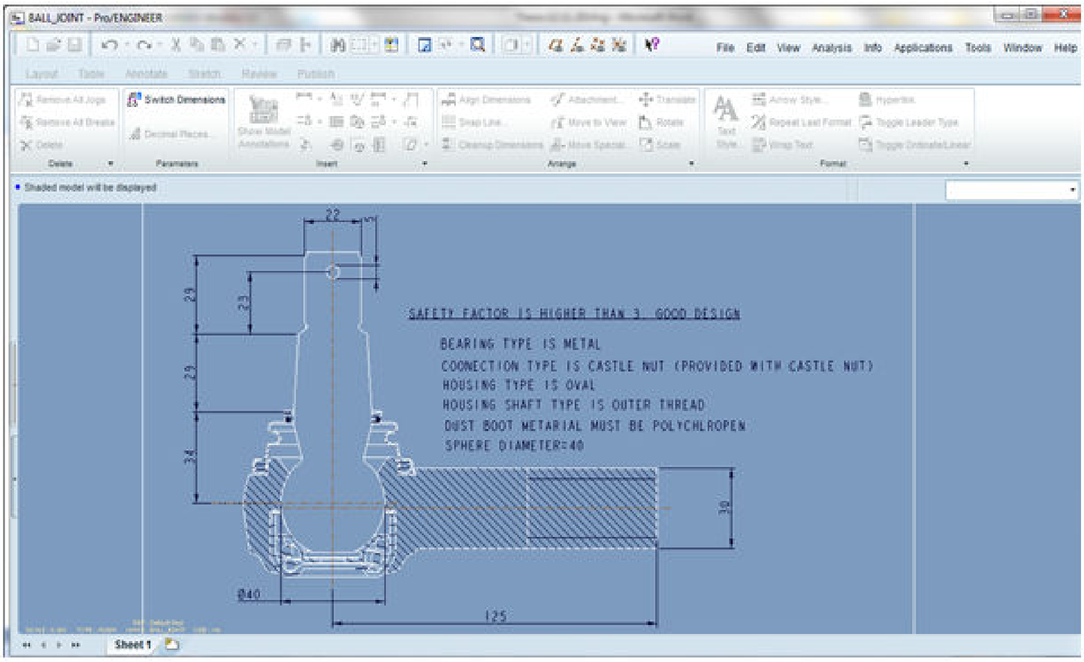


Figure 4.34. Generated 2D drawing of ball joint from definition sheet of Ø40 mm sphere diameter ball joint

Design for fatigue strength test: Formulas given in Section 3.1 were used for fatigue strength calculations. For calculations, mathematical functions in PTC Pro/program in Table 3.2 have been used. Firstly, normal and shear stresses were calculated with formula 3.2 and 3.8 respectively and then von Misses stress was calculated with formula 3.10. After that K_f was calculated using the formulas 3.15 and 3.16 and endurance limit was calculated using the formulas 3.12, 3.13 and 3.14. Lastly safety factor was calculated using formula 3.11.

Two types of ball pin material 41Cr4 and 42CrMo4 were used with ultimate tensile strength 900 Mpa and 1100 Mpa respectively. Forces on ball pin were taken from Table 3.5 and 3.6. As a result of calculations a safety factor was determined. Safety factor were shown in 3D model as seen in Fig 4.35.

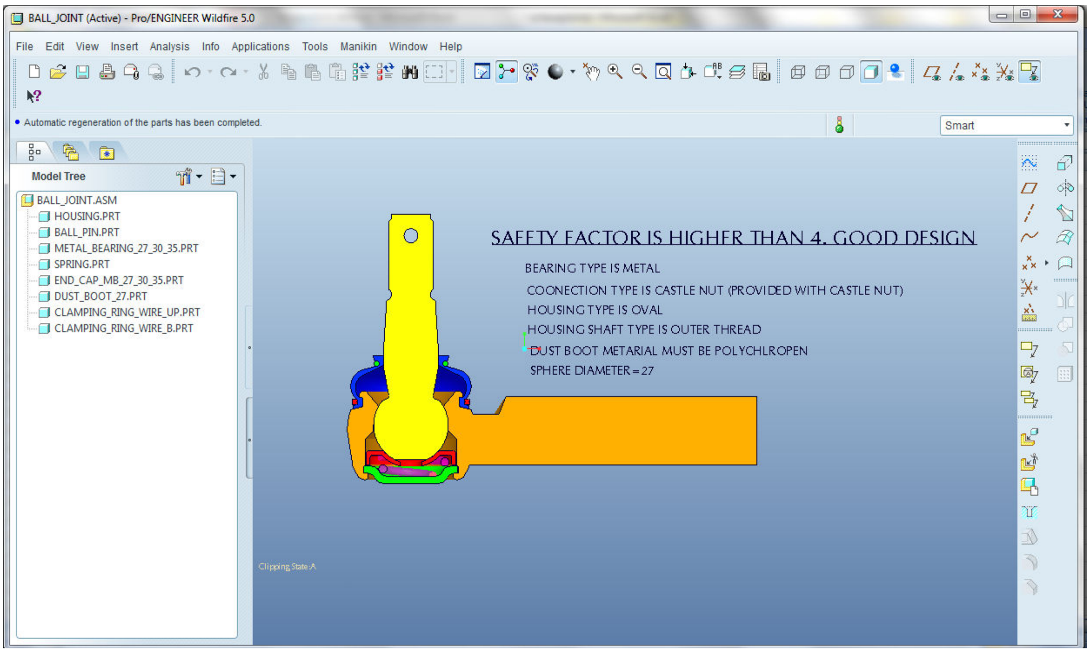


Figure 4.35. Demonstration of safety factor in 3D model

5. CONCLUSION AND FUTURE WORK

Products contain many systems sub-systems, assemblies and parts which are designed using commercial CAD software. Systems and sub-systems of any product can be considered as an assembly in CAD concept. A CAD assembly model can be decomposed as parts, features of parts, geometry of features and dimensions of features. Modification in CAD assembly model can be provided with changing any item of decomposition manually or parametrically. In this study, a platform called as "*parametric design platform*" has been developed for the parametric design of automotive ball joint.

In the design of the *"parametric design platform"* a parametric design algorithm which contain definition, reasoning and presentation phases has been proposed for part and assembly design. Most of the 3D parametric design algorithms can only be used for part design. The proposed system can be used for part and assembly design with top-down design approach, additionally, the major advantage of the proposed system is that the system can parametrically change assembly, part, part material, feature, geometry and dimensions in a programmable environment. This provides a wide range of alternative solutions to design every parts of ball joints systematically. Whereas parametric systems, which is not programmable, provide change in dimensions only.

In classical parametric system assembly, parts of assembly and features of parts are stable and only dimensions can be changed parametrically and logical statements in programmable environment can only be applied on dimensions. The proposed parametric system can change assembly, parts of assembly, and features of part parametrically in a programmable environment. Additionally, rule-based algorithm of proposed system allows adding gained experiences to the proposed parametric design system.

The *"parametric design algorithm"* has been applied on automotive ball joints with top-down approach. Assembly of ball joint contains all possible alternatives and, all parts of ball joints, every feature of each part, geometries of features and all dimensions have been designed in Pro/Engineer program with the *"flow diagram of* *new approach"*. The original *"flow diagram"* and *"decompositions technique"* are very useful method for the top-down design approach since it shows the relationships between assembly, parts, features, geometry, dimensions and logical statements and algebraic calculations. The relationships also allow adding gained knowledge to the design.

The logical and algebraic statements are applied on Pro/program module with software. The commercial software used in the development of the platform has been prepared as visual and user-friendly as possible. However, the links between definition, reasoning and presentation require manual interventions by the user, and the developed platform needs further enhancement for professional use.

After completing the development of *"parametric design platform"* tests have been applied to validate design. The tests applied on platform shows that the platform is able to design all parts of ball joints and assembly parts according to input parameters. In addition, the type and dimensions of ball joint can be changed rapidly and easily. Considering that hundreds of different types and configurations of ball joints arise with the many model changes, updates, redesign on vehicles by the Original Equipment Manufacturers (OEMs), the need of ball joints design with minimum time and effort has been met by the developed platform. The proposed approach can also be applied in similar designs.

Consequently, the proposed algorithm for *"the parametric design platform"* can successfully be used for the automotive ball joints and similar parametric designs. The method reduces design time and costs, demonstrate results in 3D and 2D graphical views and can apply gained experiences to existing design.

In future studies, additional modules can be adapted to the platform to improve parametric design. Probable modules that can be adapted to platform are;

1. Manufacturing module to make NC codes. The module can select suitable machine tool according to parameters and prepare NC codes automatically.
2. Mold modules to design hot forging, cold forging, casting or plastic molding dies. The type of molds can be selected and full 3D model of dies can be prepared according to parameters.

3. Analysis programs (Ansys, Msc Nastran etc.) can be adapted to make FEA analyses of parts or assemblies.

REFERENCES

AIT-AOUDIA, S. and FOUFOU, S., 2010. A 2D geometric constraint solver using a graph reduction method. Advances in Engineering Software, 41:1187-1194.

AMADORI, K., TARKIAN, M., ÖLVANDER, J., and KRUS, P., 2012. Flexible and robust CAD models for design automation. Advanced Engineering Informatics.

BADIRU, A. B., 1992, Expert System Applications in Engineering and Manufacturing, Prentic Hall .

BADIRU, A. B., CHEUNG, J. Y., 2002, Fuzzy Engineering Expert Systems with Neural Network Applications. John Wiley&Sons.

BODEIN, Y., ROSE, B., & CAILLAUD, E., 2014. Explicit reference modeling methodology in parametric CAD system. Computers in Industry 65:136-147.

BYRNE, J., CARDIFF, P., BRABAZON, A., and O'NEILL, M., 2014. Evolving parametric aircraft models for design exploration and optimisation. Neurocomputing 142:39–47.

CHAPMAN, C. B., & PINFOLD, M., 1999. Design engineering—a need to rethink the solution using knowledge based engineering. Knowledge-Based Systems 12:257-267.

CHAPMAN, C. B., & PINFOLD, M., 2001. The application of knowledge based engineering approach to the rapid design and analysis of an automotive structure. Journal of Advances in Engineering Software 32:903-912.

CHU, C. H., SONG, M. C., & LUO, V. C., 2006. Computer aided parametric design for 3D tire mold production. Computers in Industry V.57:11-15.

CHUNG, J. C., HWANG, T. S., WU, C. T., & JIANG, Y. W., 2000. Framework for integrated mechanical design automation. Computer-Aided Design V. 32:355-365.

DAVIS, D., 2013. Modelled on Software Engineering: Flexible Parametric Models in the Practice of Architecture. Phd Thesis, RMIT University.

GE, J.-X., CHOU, S.-C., and GAO, X., 1999. Geometric constraint satisfaction using optimization methods. Computer-Aided Design 31:867-879.

GUJARATHI, G. P., and MA, Y.-S., 2011. Parametric CAD/CAE integration using a common data model. Journal of Manufacturing Systems 30:118-132.

HEISLER, H., 2002. Advanced Vehicle Technology. Butterworth-Heinemann, London 654p.

HERNANDEZ, C. R., 2006. Thinking Parametric Design: Introducing Parametric Gaudy. Design Studies 27:309-324.

http://www2.mae.ufl.edu/designlab/Class%20Projects/Background%20Information/Steering.htm.

LA ROCCA, G., 2012. Knowledge based engineering: Between AI and CAD. Review of a languagebased technology to support engineering design. Advanced Engineering Informatics 26:159-179.

LI, Y., HU, S.-M., & SUN, J.-G., 2002. A constructive approach to solving 3-D geometric constraint systems using dependence analysis. Computer-Aided Design 34:97-108.

LIU, G., 2011. Parameterized Mockup Design of PDC Bits Based on Pro/E. Procedia Earth and Planetary Science 3:435-439.

MERMOZ, E., LINERAS, J. M., & BERNARD, A., 2011. Benefits and limitations of parametric design implementation in helicopter gearbox design phase. CIRP Annals - Manufacturing Technology V.60:199-202.

MOK, H.-S., KIM, C.-H., & KIM, C.-B., 2011. Automation of Mold Designs with the Reuse of Standard Parts. Expert Systems with Applications 38:12537–12547.

MONEDERO, J., 2000. Parametric design: a review and some experiences. Automation, 369–377.

MYUNG, S., and HAN, S., 2001. Knowledge-based parametric design of mechanical products based on configuration design method. Expert Systems with Applications 21: 99-107.

OU, L.-H., & XU, X., 2013. Relationship matrix based automatic assembly sequence generationfrom a CAD model. Computer-Aided Design 45:1053-1067.

QIAOSHENG, L., & XI, J., 2011. Case-based parametric design system for test turntable. Expert Systems with Applications V.38:6508-6516.

RAHMAN, A., ZLATANOVA, S., & PLIOUK, M., 2000)-. The 3D GIS software development: global efforts from researchers and vendors. Geoinformation Science Journal, vol. 1, no. 2.

REIMPELL, J., STOLL, H., & BETZLER, J. W., 2002. The Automotive Chassis: Engineering Principles. Society of Automotive Engineers Inc, SAE.

SHIN, J. H., & KWAK, B. M., 1999. Integration of commercial codes from CAD to optimization for structural design. First Chine-Japan-Korea Joint Symposium on Optimization of Structural and Mechanical System , 649-655.

TS 5476.

TS 9444.

TUAN, A., 2013. Overview of Three-Dimensional GIS Data Models. International Journal of Future Computer and Communication, Vol. 2, No. 3:270-274.

VERROUST, A., SCHONEK, F., & ROLLER, D., 1992. Rule-oriented method for parameterized computer-aided design. Computer Aided Design, 24:531-540.

WEISBERG, D., 2008. The Engineering Design Revolution: The People,Companies and Computer Systems that Changed Forever the Practice of Engineering. http://www.cadhistory.net/.

WOODBURY, R., 2010. Elements of Parametric Design. Bentley Institute Press.

WU, S. D., FUH, J. Y., & LEE, K. S., 2007. Semi-automated parametric design of gating systems for die-casting die. Semi-automated paraComputers & Industrial Engineering V. 53:222-232.

www.cablecraft.com.

www.powersandsonsllc.com.

www.ptc.com.

www.steelnumber.com.

www.zf.com.

CURRICULUM VITAE

Osman Oktay AKÇALI was born in Kozan/Adana in 1978. He graduated from Çukurova University as mechanical engineer in 2000.

After he has completed his military service as a reserve officer in 2003, he started to work in Ditaş company as product development engineer. He was responsible to design ball joints for OEMs (Original Equipment Manufacturers) and spare markets. In 2007, He started to work as mechanical engineer in DSI (State Hydraulic Works) responsible for maintenance of construction machines. In 2010, he started to work in Botaş (Petrolium Pipeline Corporation) as maintenance engineer.

He is married and has a son.

APPENDIX A

Housing program

```
VERSION 5.0
REVNUM 5025
LISTING FOR PART HOUSING

INPUT
 BJ_1H STRING
 BJ_2H STRING
 BJ_3H STRING
 BJ_4H STRING
 BJ_14H NUMBER
 BJ_15H NUMBER
 BJ_17H NUMBER
 BJ_18H NUMBER
 BJ_19H NUMBER
 BJ_16H NUMBER
END INPUT

RELATIONS

IF BJ_14H==40
H_1=43
H_2=19
H_3=69
H_4=9
ENDIF

IF BJ_1H=="METAL"
IF BJ_14H==40
H_5=48
H_6=40
H_7=2
H_8=34
H_9=55
H_10=0.5
H_11=15
ENDIF
```

```
ENDIF

IF BJ_14H==35
H_1=40.5
H_2=16
H_3=60
H_4=9
ENDIF

IF BJ_1H=="METAL"
IF BJ_14H==35
H_5=41.1
H_6=35
H_7=2
H_8=30.5
H_9=55
H_10=0.1
H_11=45
ENDIF
ENDIF

IF BJ_14H==30
H_1=36.5
H_2=16
H_3=52
H_4=9
ENDIF

IF BJ_1H=="METAL"
IF BJ_14H==30
H_5=36
H_6=30
H_7=2
H_8=26.5
H_9=55
H_10=0.1
H_11=45
```

```
ENDIF
ENDIF

IF BJ_14H==27
H_1=32
H_2=14
H_3=46
H_4=9
ENDIF

IF BJ_1H=="METAL"
IF BJ_14H==27
H_5=32.8
H_6=27
H_7=2
H_8=23.5
H_9=50
H_10=0.1
H_11=45
ENDIF
ENDIF

H_37=BJ_18H
H_36=BJ_17H
H_37_1=BJ_18H+1

H_34=BJ_19H
H_35=BJ_15H
END RELATIONS

 ADD FEATURE (initial number 1)
 INTERNAL FEATURE ID  1


 DATUM PLANE

  NO.     ELEMENT NAME    INFO
```

```
---     -------------   -------------
 1      Feature Name    Defined
 2      Constraints     Defined
 2.1    Constraint #1   Defined
 2.1.1  Constr Type     X Axis
 3      Flip Datum Dir  Defined
 4      Fit             Defined
 4.1    Fit Type        Default

NAME = RIGHT

  FEATURE IS IN LAYER(S) :
    01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
    01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 2)
INTERNAL FEATURE ID  3


DATUM PLANE

 NO.    ELEMENT NAME    INFO
---     -------------   -------------
 1      Feature Name    Defined
 2      Constraints     Defined
 2.1    Constraint #1   Defined
 2.1.1  Constr Type     Y Axis
 3      Flip Datum Dir  Defined
 4      Fit             Defined
 4.1    Fit Type        Default

NAME = TOP

  FEATURE IS IN LAYER(S) :
    01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
```

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)
INTERNAL FEATURE ID 1210
PARENTS = 3(#2)

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined

2 Constraints Defined
2.1 Constraint #1 Defined
2.1.1 Constr Type Offset
2.1.2 Constr References Surface TOP of feat #2 (DATUM PLANE)
2.1.3 Constr Ref Offset Value = 2.0000
3 Flip Datum Dir Defined
4 Fit Defined
4.1 Fit Type Default

NAME = DTM1

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d98 = (Displayed:) 2
(Stored:) 2.0 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 5)
INTERNAL FEATURE ID 1212
PARENTS = 1(#1) 1210(#4)

DATUM AXIS

NO. ELEMENT NAME INFO
--- ------------- -------------
1 Feature Name Defined
2 Constraints Defined
2.1 Constraint #1 Defined
2.1.1 Type Through
2.1.2 Reference RIGHT:F1(DATUM PLANE)
2.2 Constraint #2 Defined
2.2.1 Type Through
2.2.2 Reference DTM1:F4(DATUM PLANE)
3 Fit Defined
3.1 Fit Type Default

NAME = A_15

FEATURE IS IN LAYER(S) :

02___PRT_ALL_AXES - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 6)

INTERNAL FEATURE ID 1247

PARENTS = 5(#3) 1210(#4)

DATUM AXIS

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Type	Through
2.1.2	Reference	FRONT:F3(DATUM PLANE)
2.2	Constraint #2	Defined
2.2.1	Type	Through
2.2.2	Reference	DTM1:F4(DATUM PLANE)
3	Fit	Defined
3.1	Fit Type	Default

NAME = A_16

FEATURE IS IN LAYER(S) :

02___PRT_ALL_AXES - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 7)

INTERNAL FEATURE ID 7

TYPE = COORDINATE SYSTEM

NAME = PRT_CSYS_DEF

```
  FEATURE IS IN LAYER(S) :
   05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN
   05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 8)
INTERNAL FEATURE ID  39
PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

 NO.    ELEMENT NAME        INFO
 ---    -------------       -------------
  1     Feature Name        Defined
  2     Extrude Feat type   Solid
  3     Material            Add
  4     Section             Defined
  4.1    Setup Plane        Defined
  4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2  View Direction     Side 1
  4.1.3  Orientation        Right
  4.1.4  Reference          RIGHT:F1(DATUM PLANE)
  4.2    Sketch             Defined
  5     Feature Form        Solid
  6     Revolve Axis        Defined
  7     Revolve Axis Option  Internal Centerline
  8     Direction           Side 2
  9     Angle               Defined
  9.1    Side One           Defined
  9.1.1  Side One Angle     None
  9.2    Side Two           Defined
  9.2.1  Side Two Angle     Variable
  9.2.2  Value              360.00

SECTION NAME = S2D0015
```

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d0 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
h_3 = (Displayed:) 69 Dia
(Stored:) 69.0 (0.01, -0.01)
h_4 = (Displayed:) 9
(Stored:) 9.0 (0.5, -0.5)
h_2 = (Displayed:) 19
(Stored:) 19.0 (0.01, -0.01)
h_1 = (Displayed:) 43
(Stored:) 43.0 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 9)
INTERNAL FEATURE ID 90
PARENTS = 3(#2) 5(#3) 1(#1)

PROTRUSION: Extrude

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	RIGHT:F1(DATUM PLANE)
4.1.2	View Direction	Defined
4.1.3	Orientation	Top
4.1.4	Reference	TOP:F2(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid

6 Direction Side 1
7 Depth Defined
7.1 Side One Defined
7.1.1 Side One Depth None
7.2 Side Two Defined
7.2.1 Side Two Depth Variable
7.2.2 Value 145.00

SECTION NAME = S2D0016

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
h_35 = (Displayed:) 145
(Stored:) 145.0 (0.01, -0.01)
h_34 = (Displayed:) 24 Dia
(Stored:) 24.0 (0.01, -0.01)
END ADD

IF BJ_14H==27

ADD FEATURE
INTERNAL FEATURE ID 3276
PARENTS = 39(#8) 1(#1) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1

4.1.3 Orientation Right
4.1.4 Reference RIGHT:F1(DATUM PLANE)
4.2 Sketch Defined
5 Feature Form Solid
6 Material Side Side Two
7 Revolve Axis Defined
8 Revolve Axis Option Internal Centerline
9 Direction Side 2
10 Angle Defined
10.1 Side One Defined
10.1.1 Side One Angle None
10.2 Side Two Defined
10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

SECTION NAME = S2D0002

FEATURE'S DIMENSIONS:
d213 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d226 = (Displayed:) 70 Dia
(Stored:) 70.0 (0.01, -0.01)
d227 = (Displayed:) 8
(Stored:) 8.0 (0.01, -0.01)
d228 = (Displayed:) 0 (weak)
(Stored:) 0.0 (0.01, -0.01)
d229 = (Displayed:) 63.66 Dia (weak)
(Stored:) 63.66 (0.01, -0.01)
END ADD
END IF

IF BJ_1H=="METAL"

ADD FEATURE (initial number 10)
INTERNAL FEATURE ID 122
PARENTS = 1(#1) 3(#2) 39(#8) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0017

FEATURE'S DIMENSIONS:
d7 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
h_6 = (Displayed:) 40 Dia
(Stored:) 40.0 (0.01, -0.01)
h_7 = (Displayed:) 2
(Stored:) 2.0 (0.01, -0.01)
h_11 = (Displayed:) 15
(Stored:) 15.0 (0.5, -0.5)

```
  h_10 = (Displayed:) 0.5
       (   Stored:) 0.5 ( 0.01, -0.01 )
  h_9 = (Displayed:) 55
      (   Stored:) 55.0 ( 0.5, -0.5 )
  h_8 = (Displayed:) 34 Dia
      (   Stored:) 34.0 ( 0.01, -0.01 )
  h_5 = (Displayed:) 48 Dia
      (   Stored:) 48.0 ( 0.01, -0.01 )
  END ADD
END IF

IF BJ_1H=="METAL"
  IF BJ_3H=="ELLIPTICAL"

    ADD FEATURE
    INTERNAL FEATURE ID  215
    PARENTS = 39(#8) 122(#10) 1(#1) 5(#3)

PROTRUSION: Revolve

 NO.    ELEMENT NAME        INFO
 ---    -------------       -------------
  1     Feature Name        Defined
  2     Extrude Feat type   Solid
  3     Material            Add
  4     Section             Defined
  4.1    Setup Plane        Defined
  4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2  View Direction     Side 1
  4.1.3  Orientation        Right
  4.1.4  Reference          RIGHT:F1(DATUM PLANE)
  4.2    Sketch             Defined
  5     Feature Form        Solid
  6     Revolve Axis        Defined
  7     Revolve Axis Option  Internal Centerline
  8     Direction           Side 2
  9     Angle               Defined
```

9.1 Side One Defined

9.1.1 Side One Angle None

9.2 Side Two Defined

9.2.1 Side Two Angle Variable

9.2.2 Value 360.00

SECTION NAME = S2D0018

FEATURE'S DIMENSIONS:

d15 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

d16 = (Displayed:) 180 (weak)

(Stored:) 180.0 (0.5, -0.5)

END ADD

ADD FEATURE

INTERNAL FEATURE ID 314

PARENTS = 1(#1) 5(#3) 215(*) 39(#8)

CUT: Blend, Parallel, Regular Sections

NO.	ELEMENT NAME	INFO	STATUS
1	Attributes	Smooth	Defined
2	Section	Sk. plane - Surface of feat #8 (PROTRUSION)	Defined
3	MaterialSide	Inside section	Defined
4	Direction		Defined
5	Depth	References are missing.	Defined

SECTION NAME = S2D0001

FEATURE'S DIMENSIONS:

d17 = (Displayed:) 7.5

(Stored:) 7.5 (0.01, -0.01)

d20 = (Displayed:) 45

(Stored:) 45.0 (0.01, -0.01)

d21 = (Displayed:) 42

```
        (   Stored:) 42.0 ( 0.01, -0.01 )
      d24 = (Displayed:) 35
        (   Stored:) 35.0 ( 0.01, -0.01 )
      d25 = (Displayed:) 34
        (   Stored:) 34.0 ( 0.01, -0.01 )
      END ADD
   END IF
END IF

IF BJ_14H==40

   ADD FEATURE (initial number 11)
   INTERNAL FEATURE ID  423
   PARENTS = 1(#1) 39(#8) 5(#3)

CUT: Revolve

 NO.     ELEMENT NAME        INFO
 ---     -------------       -------------
  1      Feature Name        Defined
  2      Extrude Feat type   Solid
  3      Material            Remove
  4      Section             Defined
  4.1     Setup Plane        Defined
  4.1.1   Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2   View Direction     Side 1
  4.1.3   Orientation        Right
  4.1.4   Reference          RIGHT:F1(DATUM PLANE)
  4.2     Sketch             Defined
  5      Feature Form        Solid
  6      Material Side       Side Two
  7      Revolve Axis        Defined
  8      Revolve Axis Option  Internal Centerline
  9      Direction           Side 2
  10      Angle              Defined
  10.1    Side One           Defined
  10.1.1  Side One Angle     None
```

10.2   Side Two   Defined
10.2.1 Side Two Angle   Variable
10.2.2 Value   360.00

SECTION NAME = S2D0020

FEATURE'S DIMENSIONS:
d26 = (Displayed:) 360
    (Stored:) 360.0 (0.5, -0.5)
d27 = (Displayed:) 50.2 Dia
    (Stored:) 50.2 (0.01, -0.01)
d28 = (Displayed:) 6.5
    (Stored:) 6.5 (0.01, -0.01)
d31 = (Displayed:) 0.5R
    (Stored:) 0.5 (0.01, -0.01)
d32 = (Displayed:) 47.2 Dia (weak)
    (Stored:) 47.2 (0.01, -0.01)
d33 = (Displayed:) 60 (weak)
    (Stored:) 60.0 (0.5, -0.5)
END ADD

ADD FEATURE (initial number 12)
INTERNAL FEATURE ID 494
PARENTS = 1(#1) 39(#8) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right

```
 4.1.4   Reference            RIGHT:F1(DATUM PLANE)
 4.2     Sketch               Defined
 5       Feature Form         Solid
 6       Material Side        Side Two
 7       Revolve Axis         Defined
 8       Revolve Axis Option  Internal Centerline
 9       Direction            Side 2
 10      Angle                Defined
 10.1    Side One             Defined
 10.1.1  Side One Angle       None
 10.2    Side Two             Defined
 10.2.1  Side Two Angle       Variable
 10.2.2  Value                360.00

   SECTION NAME = S2D0021

   FEATURE'S DIMENSIONS:
   d34 = (Displayed:) 360
        (   Stored:) 360.0 ( 0.5, -0.5 )
   d35 = (Displayed:) 60
        (   Stored:) 60.0 ( 0.5, -0.5 )
   d36 = (Displayed:) 135
        (   Stored:) 135.0 ( 0.5, -0.5 )
   d37 = (Displayed:) 4.4
        (   Stored:) 4.4 ( 0.01, -0.01 )
   d38 = (Displayed:) 50 Dia
        (   Stored:) 50.0 ( 0.01, -0.01 )
   d39 = (Displayed:) 60.75 Dia
        (   Stored:) 60.75 ( 0.01, -0.01 )
   END ADD
END IF

IF BJ_14H==35

   ADD FEATURE
   INTERNAL FEATURE ID  663
   PARENTS = 1(#1) 39(#8) 5(#3)
```

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0015

FEATURE'S DIMENSIONS:

d50 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)

d51 = (Displayed:) 44.2 Dia
(Stored:) 44.2 (0.01, -0.01)

d52 = (Displayed:) 5.5
(Stored:) 5.5 (0.01, -0.01)

d53 = (Displayed:) 1.5
(Stored:) 1.5 (0.01, -0.01)
d54 = (Displayed:) 30
(Stored:) 30.0 (0.5, -0.5)
d55 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 734
PARENTS = 1(#1) 39(#8) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0016

FEATURE'S DIMENSIONS:

d56 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

d57 = (Displayed:) 48 Dia

(Stored:) 48.0 (0.01, -0.01)

d58 = (Displayed:) 20

(Stored:) 20.0 (0.5, -0.5)

d59 = (Displayed:) 6

(Stored:) 6.0 (0.01, -0.01)

d60 = (Displayed:) 6.5

(Stored:) 6.5 (0.01, -0.01)

END ADD

END IF

IF BJ_14H==30

IF BJ_1H=="METAL"

ADD FEATURE

INTERNAL FEATURE ID 789

PARENTS = 1(#1) 39(#8) 122(#10) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)

4.2 Sketch Defined

5 Feature Form Solid

6 Material Side Side Two

7 Revolve Axis Defined

8 Revolve Axis Option Internal Centerline

9 Direction Side 2

10 Angle Defined

10.1 Side One Defined

10.1.1 Side One Angle None

10.2 Side Two Defined

10.2.1 Side Two Angle Variable

10.2.2 Value 360.00

SECTION NAME = S2D0017

FEATURE'S DIMENSIONS:

d61 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

d62 = (Displayed:) 38 Dia

(Stored:) 38.0 (0.01, -0.01)

d63 = (Displayed:) 6.5

(Stored:) 6.5 (0.01, -0.01)

END ADD

ADD FEATURE

INTERNAL FEATURE ID 843

PARENTS = 1(#1) 39(#8) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined

4.1.1 Sketching Plane FRONT:F3(DATUM PLANE)
4.1.2 View Direction Side 1
4.1.3 Orientation Right
4.1.4 Reference RIGHT:F1(DATUM PLANE)
4.2 Sketch Defined
5 Feature Form Solid
6 Material Side Side Two
7 Revolve Axis Defined
8 Revolve Axis Option Internal Centerline
9 Direction Side 2
10 Angle Defined
10.1 Side One Defined
10.1.1 Side One Angle None
10.2 Side Two Defined
10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

SECTION NAME = S2D0018

FEATURE'S DIMENSIONS:
d64 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d65 = (Displayed:) 40 Dia
(Stored:) 40.0 (0.01, -0.01)
d67 = (Displayed:) 5
(Stored:) 5.0 (0.01, -0.01)
d68 = (Displayed:) 6.5
(Stored:) 6.5 (0.01, -0.01)
d69 = (Displayed:) 44.13 Dia (weak)
(Stored:) 44.12914247082 (0.01, -0.01)
END ADD
END IF

ADD FEATURE
INTERNAL FEATURE ID 898
PARENTS = 39(#8) 1(#1) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0019

FEATURE'S DIMENSIONS:
d70 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d71 = (Displayed:) 39.2 Dia
(Stored:) 39.2 (0.01, -0.01)
d72 = (Displayed:) 5.5
(Stored:) 5.5 (0.01, -0.01)
d73 = (Displayed:) 1
(Stored:) 1.0 (0.01, -0.01)

```
   d74 = (Displayed:) 1R
       (   Stored:) 1.0 ( 0.01, -0.01 )
   d75 = (Displayed:) 30
       (   Stored:) 30.0 ( 0.5, -0.5 )
   END ADD
END IF

IF BJ_14H==27
   IF BJ_1H=="METAL"

      ADD FEATURE
      INTERNAL FEATURE ID  967
      PARENTS = 1(#1) 39(#8) 5(#3)

CUT: Revolve

 NO.    ELEMENT NAME        INFO
 ---    -------------       -------------
  1     Feature Name        Defined
  2     Extrude Feat type   Solid
  3     Material            Remove
  4     Section             Defined
  4.1    Setup Plane        Defined
  4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2  View Direction     Side 1
  4.1.3  Orientation        Right
  4.1.4  Reference          RIGHT:F1(DATUM PLANE)
  4.2    Sketch             Defined
  5     Feature Form        Solid
  6     Material Side       Side Two
  7     Revolve Axis        Defined
  8     Revolve Axis Option  Internal Centerline
  9     Direction           Side 2
  10     Angle              Defined
  10.1   Side One           Defined
  10.1.1  Side One Angle     None
  10.2   Side Two           Defined
```

10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

SECTION NAME = S2D0020

FEATURE'S DIMENSIONS:
d76 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d77 = (Displayed:) 5.5
(Stored:) 5.5 (0.01, -0.01)
d78 = (Displayed:) 34 Dia
(Stored:) 34.0 (0.01, -0.01)
d97 = (Displayed:) 10
(Stored:) 10.0 (0.01, -0.01)
END ADD
END IF

ADD FEATURE
INTERNAL FEATURE ID 1021
PARENTS = 1(#1) 39(#8) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two

7 Revolve Axis Defined
8 Revolve Axis Option Internal Centerline
9 Direction Side 2
10 Angle Defined
10.1 Side One Defined
10.1.1 Side One Angle None
10.2 Side Two Defined
10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

SECTION NAME = S2D0021

FEATURE'S DIMENSIONS:
d79 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d80 = (Displayed:) 30
(Stored:) 30.0 (0.5, -0.5)
d81 = (Displayed:) 36 Dia
(Stored:) 36.0 (0.01, -0.01)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 1065
PARENTS = 1(#1) 39(#8) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right

4.1.4 Reference RIGHT:F1(DATUM PLANE)
4.2 Sketch Defined
5 Feature Form Solid
6 Material Side Side Two
7 Revolve Axis Defined
8 Revolve Axis Option Internal Centerline
9 Direction Side 2
10 Angle Defined
10.1 Side One Defined
10.1.1 Side One Angle None
10.2 Side Two Defined
10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

SECTION NAME = S2D0022

FEATURE'S DIMENSIONS:
d82 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d83 = (Displayed:) 15
(Stored:) 15.0 (0.5, -0.5)
d84 = (Displayed:) 35 Dia
(Stored:) 35.0 (0.01, -0.01)
d85 = (Displayed:) 1
(Stored:) 1.0 (0.01, -0.01)
d86 = (Displayed:) 1
(Stored:) 1.0 (0.01, -0.01)
d87 = (Displayed:) 37.5 Dia
(Stored:) 37.5 (0.01, -0.01)
d88 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
d89 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
d90 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
d91 = (Displayed:) 5.5
(Stored:) 5.5 (0.01, -0.01)

```
   d92 = (Displayed:) 3.4
        (   Stored:) 3.4 ( 0.01, -0.01 )
   d93 = (Displayed:) 36.4 Dia (weak)
        (   Stored:) 36.4 ( 0.01, -0.01 )
   d94 = (Displayed:) 3.4 (weak)
        (   Stored:) 3.4 ( 0.01, -0.01 )
   END ADD
END IF

IF BJ_14H==40
   IF BJ_1H=="METAL"

      ADD FEATURE (initial number 13)
      INTERNAL FEATURE ID  1251
      PARENTS = 1(#1) 39(#8) 122(#10) 5(#3)

PROTRUSION: Revolve

 NO.    ELEMENT NAME        INFO
 ---    -------------       -------------
  1     Feature Name        Defined
  2     Extrude Feat type   Solid
  3     Material            Add
  4     Section             Defined
  4.1    Setup Plane        Defined
  4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2  View Direction     Side 1
  4.1.3  Orientation        Right
  4.1.4  Reference          RIGHT:F1(DATUM PLANE)
  4.2    Sketch             Defined
  5     Feature Form        Solid
  6     Revolve Axis        Defined
  7     Revolve Axis Option  Internal Centerline
  8     Direction           Side 2
  9     Angle               Defined
  9.1    Side One           Defined
  9.1.1  Side One Angle     None
```

9.2 Side Two Defined

9.2.1 Side Two Angle Variable

9.2.2 Value 360.00

SECTION NAME = S2D0001

FEATURE'S DIMENSIONS:

d99 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

d114 = (Displayed:) 40 Dia (weak)

(Stored:) 40.0 (0.01, -0.01)

d115 = (Displayed:) 40.3 Dia (weak)

(Stored:) 40.29771707969 (0.01, -0.01)

d116 = (Displayed:) 46.55 Dia (weak)

(Stored:) 46.55321635431 (0.01, -0.01)

d117 = (Displayed:) 70 (weak)

(Stored:) 70.0 (0.5, -0.5)

d118 = (Displayed:) 6.76 (weak)

(Stored:) 6.761195155516 (0.01, -0.01)

END ADD

ADD FEATURE (initial number 14)

INTERNAL FEATURE ID 1305

PARENTS = 39(#8) 494(#12)

ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	1 Set
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined

2.1.3.1 Reference type Edge Chain
2.1.3.2 Curve Collection 2 Selections
2.1.4 Spine Defined
2.1.4.1 Ball/Spine Rolling Ball
2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined
2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 5.00
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

```
    FEATURE'S DIMENSIONS:
    d100 = (Displayed:) 5R
         (   Stored:) 5.0 ( 0.01, -0.01 )
    END ADD
  END IF
END IF

IF BJ_14H==35
  IF BJ_1H=="METAL"

    ADD FEATURE
    INTERNAL FEATURE ID  1369
    PARENTS = 1(#1) 39(#8) 122(#10) 734(*) 5(#3)
```

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined

4.1.1 Sketching Plane FRONT:F3(DATUM PLANE)
4.1.2 View Direction Side 1
4.1.3 Orientation Right
4.1.4 Reference RIGHT:F1(DATUM PLANE)
4.2 Sketch Defined
5 Feature Form Solid
6 Material Side Side Two
7 Revolve Axis Defined
8 Revolve Axis Option Internal Centerline
9 Direction Side 2
10 Angle Defined
10.1 Side One Defined
10.1.1 Side One Angle None
10.2 Side Two Defined
10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

```
      SECTION NAME = S2D0001

      FEATURE'S DIMENSIONS:
      d103 = (Displayed:) 360
             (   Stored:) 360.0 ( 0.5, -0.5 )
      d104 = (Displayed:) 44.1 Dia
             (   Stored:) 44.1 ( 0.01, -0.01 )
      d105 = (Displayed:) 6.5
             (   Stored:) 6.5 ( 0.01, -0.01 )
      END ADD
   END IF
END IF

IF BJ_14H==35
   IF BJ_1H=="METAL"

      ADD FEATURE
      INTERNAL FEATURE ID  1423
      FEATURE WAS CREATED IN ASSEMBLY BALL_JOINT
      PARENTS = 1(#1) 5(#3)
```

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0001

FEATURE'S DIMENSIONS:
d106 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d107 = (Displayed:) 3.22
(Stored:) 3.22 (0.01, -0.01)
d108 = (Displayed:) 37.92 Dia (weak)
(Stored:) 37.92 (0.01, -0.01)
d109 = (Displayed:) 47.4 Dia (weak)
(Stored:) 47.4 (0.01, -0.01)

END ADD

ADD FEATURE
INTERNAL FEATURE ID 1476
PARENTS = 1(#1) 734(*) 1423(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0002

FEATURE'S DIMENSIONS:
d110 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)

```
      d111 = (Displayed:) 43.86 Dia (weak)
           (   Stored:) 43.86 ( 0.01, -0.01 )
      d112 = (Displayed:) 42.77 Dia (weak)
           (   Stored:) 42.77426490801 ( 0.01, -0.01 )
      d113 = (Displayed:) 49.72 Dia (weak)
           (   Stored:) 49.72 ( 0.01, -0.01 )
      d131 = (Displayed:) 3.44R (weak)
           (   Stored:) 3.437186383879 ( 0.01, -0.01 )
      END ADD
   END IF
END IF

IF BJ_14H==30
   IF BJ_1H=="METAL"

      ADD FEATURE
      INTERNAL FEATURE ID  1560
      FEATURE WAS CREATED IN ASSEMBLY BALL_JOINT
      PARENTS = 1(#1) 5(#3)

PROTRUSION: Revolve

 NO.    ELEMENT NAME        INFO
 ---    -------------       -------------
  1     Feature Name        Defined
  2     Extrude Feat type   Solid
  3     Material            Add
  4     Section             Defined
  4.1    Setup Plane        Defined
  4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2  View Direction     Side 1
  4.1.3  Orientation        Right
  4.1.4  Reference          RIGHT:F1(DATUM PLANE)
  4.2    Sketch             Defined
  5     Feature Form        Solid
  6     Revolve Axis        Defined
  7     Revolve Axis Option  Internal Centerline
```

8 Direction Side 2
9 Angle Defined
9.1 Side One Defined
9.1.1 Side One Angle None
9.2 Side Two Defined
9.2.1 Side Two Angle Variable
9.2.2 Value 360.00

SECTION NAME = S2D0003

FEATURE'S DIMENSIONS:
d119 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d122 = (Displayed:) 2.23 (weak)
(Stored:) 2.234809340911 (0.01, -0.01)
d123 = (Displayed:) 33.28 Dia (weak)
(Stored:) 33.27971625399 (0.01, -0.01)
d124 = (Displayed:) 40.76 Dia (weak)
(Stored:) 40.75918427624 (0.01, -0.01)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 1613
PARENTS = 1(#1) 39(#8) 789(*) 843(*) 1560(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right

4.1.4 Reference RIGHT:F1(DATUM PLANE)
4.2 Sketch Defined
5 Feature Form Solid
6 Material Side Side Two
7 Revolve Axis Defined
8 Revolve Axis Option Internal Centerline
9 Direction Side 1
10 Angle Defined
10.1 Side One Defined
10.1.1 Side One Angle None
10.2 Side Two Defined
10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

```
    SECTION NAME = S2D0005

    FEATURE'S DIMENSIONS:
    d125 = (Displayed:) 360
          (   Stored:) 360.0 ( 0.5, -0.5 )
    d128 = (Displayed:) 37.09 Dia (weak)
          (   Stored:) 37.09258078532 ( 0.01, -0.01 )
    d129 = (Displayed:) 37.41 Dia (weak)
          (   Stored:) 37.41211268047 ( 0.01, -0.01 )
    d130 = (Displayed:) 41.83 Dia (weak)
          (   Stored:) 41.82787074057 ( 0.01, -0.01 )
    END ADD
  END IF
END IF

IF BJ_14H==27
  IF BJ_1H=="METAL"

    ADD FEATURE
    INTERNAL FEATURE ID  1705
    FEATURE WAS CREATED IN ASSEMBLY BALL_JOINT
    PARENTS = 1(#1) 5(#3)
```

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0001

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d132 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d133 = (Displayed:) 30.79 Dia (weak)
(Stored:) 30.79372070463 (0.01, -0.01)
d134 = (Displayed:) 35.84 Dia (weak)
(Stored:) 35.83825194079 (0.01, -0.01)

d135 = (Displayed:) 36.46 Dia (weak)

(Stored:) 36.45848119114 (0.01, -0.01)

d136 = (Displayed:) 12.3 (weak)

(Stored:) 12.29212522752 (0.5, -0.5)

END ADD

ADD FEATURE

INTERNAL FEATURE ID 1758

PARENTS = 1(#1) 39(#8) 967(*) 1021(*) 1705(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0002

```
        FEATURE'S DIMENSIONS:
        d137 = (Displayed:) 360
              (   Stored:) 360.0 ( 0.5, -0.5 )
        d138 = (Displayed:) 25.99 Dia (weak)
              (   Stored:) 25.99258476724 ( 0.01, -0.01 )
        d139 = (Displayed:) 33.23 Dia (weak)
              (   Stored:) 33.22768133695 ( 0.01, -0.01 )
        d140 = (Displayed:) 37.75 Dia (weak)
              (   Stored:) 37.7457272668 ( 0.01, -0.01 )
        END ADD
    END IF
END IF

IF BJ_1H=="PLASTIC"
    IF BJ_14H==27

        ADD FEATURE
        INTERNAL FEATURE ID  1942
        FEATURE WAS CREATED IN ASSEMBLY BALL_JOINT
        PARENTS = 1(#1) 39(#8) 5(#3)
CUT: Revolve

 NO.     ELEMENT NAME        INFO
 ---     -------------       -------------
  1      Feature Name        Defined
  2      Extrude Feat type   Solid
  3      Material            Remove
  4      Section             Defined
  4.1     Setup Plane        Defined
  4.1.1   Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2   View Direction     Side 1
  4.1.3   Orientation        Right
  4.1.4   Reference          RIGHT:F1(DATUM PLANE)
  4.2     Sketch             Defined
  5      Feature Form        Solid
  6      Material Side       Side Two
```

7 Revolve Axis Defined

8 Revolve Axis Option Internal Centerline

9 Direction Side 2

10 Angle Defined

10.1 Side One Defined

10.1.1 Side One Angle None

10.2 Side Two Defined

10.2.1 Side Two Angle Variable

10.2.2 Value 360.00

SECTION NAME = S2D0001

FEATURE'S DIMENSIONS:
d141 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d142 = (Displayed:) 45
(Stored:) 45.0 (0.5, -0.5)
d143 = (Displayed:) 5
(Stored:) 5.0 (0.01, -0.01)
d145 = (Displayed:) 36 (weak)
(Stored:) 36.0 (0.01, -0.01)
d146 = (Displayed:) 42.43 Dia (weak)
(Stored:) 42.42891868459 (0.01, -0.01)
END ADD
END IF
END IF

IF BJ_1H=="PLASTIC"
IF BJ_14H==27
IF BJ_3H=="ELLIPTICAL"

ADD FEATURE
INTERNAL FEATURE ID 2043
PARENTS = 39(#8) 1942(*) 1(#1) 5(#3)

PROTRUSION: Revolve

```
NO.    ELEMENT NAME        INFO
---    -------------       -------------
 1     Feature Name        Defined
 2     Extrude Feat type   Solid
 3     Material            Add
 4     Section             Defined
 4.1    Setup Plane        Defined
 4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
 4.1.2  View Direction     Side 1
 4.1.3  Orientation        Right
 4.1.4  Reference          RIGHT:F1(DATUM PLANE)
 4.2    Sketch             Defined
 5     Feature Form        Solid
 6     Revolve Axis        Defined
 7     Revolve Axis Option  Internal Centerline
 8     Direction           Side 2
 9     Angle               Defined
 9.1    Side One           Defined
 9.1.1  Side One Angle     None
 9.2    Side Two           Defined
 9.2.1  Side Two Angle     Variable
 9.2.2  Value              360.00

        SECTION NAME = S2D0002

        FEATURE'S DIMENSIONS:
        d147 = (Displayed:) 360
             (   Stored:) 360.0 ( 0.5, -0.5 )
        END ADD
     END IF
  END IF
END IF

IF BJ_1H=="PLASTIC"
  IF BJ_14H==27
```

ADD FEATURE
INTERNAL FEATURE ID 2112
PARENTS = 1(#1) 39(#8) 1021(*) 1942(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0003

FEATURE'S DIMENSIONS:
d148 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d149 = (Displayed:) 35
(Stored:) 35.0 (0.5, -0.5)

d150 = (Displayed:) 1

(Stored:) 1.0 (0.01, -0.01)

d151 = (Displayed:) 39.71 Dia (weak)

(Stored:) 39.71 (0.01, -0.01)

d152 = (Displayed:) 42.51 Dia (weak)

(Stored:) 42.51 (0.01, -0.01)

END ADD

ADD FEATURE

INTERNAL FEATURE ID 2176

PARENTS = 1(#1) 39(#8) 1942(*) 2112(*) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0004

FEATURE'S DIMENSIONS:
d153 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d154 = (Displayed:) 2.3
(Stored:) 2.3 (0.01, -0.01)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 2229
PARENTS = 1(#1) 1942(*) 2112(*) 2176(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined

10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

SECTION NAME = S2D0005

FEATURE'S DIMENSIONS:
d156 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d157 = (Displayed:) 30.18 Dia (weak)
(Stored:) 30.1804485578 (0.01, -0.01)
d158 = (Displayed:) 30.27 Dia (weak)
(Stored:) 30.27070120661 (0.01, -0.01)
d159 = (Displayed:) 30.48 Dia (weak)
(Stored:) 30.47788267953 (0.01, -0.01)
END ADD
END IF
END IF

IF BJ_1H=="PLASTIC"
IF BJ_14H==30

ADD FEATURE
INTERNAL FEATURE ID 2315
FEATURE WAS CREATED IN ASSEMBLY BALL_JOINT
PARENTS = 39(#8) 898(*) 1(#1) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1

```
 4.1.3   Orientation          Right
 4.1.4   Reference            RIGHT:F1(DATUM PLANE)
 4.2     Sketch               Defined
 5       Feature Form         Solid
 6       Material Side        Side Two
 7       Revolve Axis         Defined
 8       Revolve Axis Option  Internal Centerline
 9       Direction            Side 2
 10      Angle                Defined
 10.1    Side One             Defined
 10.1.1  Side One Angle       None
 10.2    Side Two             Defined
 10.2.1  Side Two Angle       Variable
 10.2.2  Value                360.00

      SECTION NAME = S2D0006

      FEATURE'S DIMENSIONS:
      d160 = (Displayed:) 360
             (   Stored:) 360.0 ( 0.5, -0.5 )
      d161 = (Displayed:) 5
             (   Stored:) 5.0 ( 0.01, -0.01 )
      d162 = (Displayed:) 45
             (   Stored:) 45.0 ( 0.5, -0.5 )
      END ADD
   END IF
END IF

IF BJ_1H=="PLASTIC"
   IF BJ_14H==30

      ADD FEATURE
      INTERNAL FEATURE ID  2467
      PARENTS = 39(#8) 2315(*) 1(#1) 5(#3)

CUT: Revolve
 NO.     ELEMENT NAME        INFO
```

---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0008

FEATURE'S DIMENSIONS:
d166 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d172 = (Displayed:) 47.44 Dia (weak)
(Stored:) 47.44 (0.01, -0.01)
d173 = (Displayed:) 38.92 Dia (weak)
(Stored:) 38.92 (0.01, -0.01)
d174 = (Displayed:) 48.41 Dia (weak)
(Stored:) 48.41 (0.01, -0.01)
d175 = (Displayed:) 57.72 (weak)
(Stored:) 57.72 (0.5, -0.5)
d176 = (Displayed:) 141.74 (weak)

(Stored:) 141.74 (0.5, -0.5)

END ADD

ADD FEATURE

INTERNAL FEATURE ID 2416

PARENTS = 1(#1) 39(#8) 2315(*) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0007

FEATURE'S DIMENSIONS:

d163 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

d164 = (Displayed:) 2.5
(Stored:) 2.5 (0.01, -0.01)
d165 = (Displayed:) 29 Dia
(Stored:) 29.0 (0.01, -0.01)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 2533
PARENTS = 1(#1) 2315(*) 2416(*) 2467(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0009

```
        FEATURE'S DIMENSIONS:
        d177 = (Displayed:) 360
               (   Stored:) 360.0 ( 0.5, -0.5 )
        d178 = (Displayed:) 33.73 Dia (weak)
               (   Stored:) 33.72615554314 ( 0.01, -0.01 )
        d179 = (Displayed:) 33.76 Dia (weak)
               (   Stored:) 33.7577773356 ( 0.01, -0.01 )
        d180 = (Displayed:) 33.83 Dia (weak)
               (   Stored:) 33.826056402 ( 0.01, -0.01 )
        d181 = (Displayed:) 40.66 Dia (weak)
               (   Stored:) 40.66088718004 ( 0.01, -0.01 )
        END ADD
    END IF
END IF

IF BJ_1H=="PLASTIC"
    IF BJ_14H==35

        ADD FEATURE
        INTERNAL FEATURE ID  2599
        FEATURE WAS CREATED IN ASSEMBLY BALL_JOINT
        PARENTS = 1(#1) 39(#8) 663(*) 734(*) 5(#3)

CUT: Revolve

 NO.     ELEMENT NAME        INFO
 ---     -------------       -------------
  1      Feature Name        Defined
  2      Extrude Feat type   Solid
  3      Material            Remove
  4      Section             Defined
  4.1     Setup Plane        Defined
  4.1.1   Sketching Plane     FRONT:F3(DATUM PLANE)
  4.1.2   View Direction      Side 1
  4.1.3   Orientation        Right
  4.1.4   Reference          RIGHT:F1(DATUM PLANE)
```

```
 4.2    Sketch              Defined
 5      Feature Form        Solid
 6      Material Side       Side Two
 7      Revolve Axis        Defined
 8      Revolve Axis Option  Internal Centerline
 9      Direction           Side 2
 10     Angle               Defined
 10.1   Side One            Defined
 10.1.1  Side One Angle      None
 10.2   Side Two            Defined
 10.2.1  Side Two Angle      Variable
 10.2.2  Value              360.00

    SECTION NAME = S2D0010

    FEATURE'S DIMENSIONS:
    d182 = (Displayed:) 360
          (   Stored:) 360.0 ( 0.5, -0.5 )
    d183 = (Displayed:) 5.5
          (   Stored:) 5.5 ( 0.01, -0.01 )
    d184 = (Displayed:) 50
          (   Stored:) 50.0 ( 0.5, -0.5 )
    END ADD
  END IF
END IF

IF BJ_1H=="PLASTIC"
  IF BJ_14H==35
    IF BJ_3H=="ELLIPTICAL"

      ADD FEATURE
      INTERNAL FEATURE ID  2701
      PARENTS = 1(#1) 39(#8) 663(*) 2599(*) 5(#3)

PROTRUSION: Revolve

 NO.    ELEMENT NAME        INFO
```

```
 ---    -------------      -------------
  1     Feature Name        Defined
  2     Extrude Feat type    Solid
  3     Material            Add
  4     Section             Defined
  4.1    Setup Plane         Defined
  4.1.1  Sketching Plane      FRONT:F3(DATUM PLANE)
  4.1.2  View Direction       Side 1
  4.1.3  Orientation          Right
  4.1.4  Reference            RIGHT:F1(DATUM PLANE)
  4.2    Sketch              Defined
  5     Feature Form        Solid
  6     Revolve Axis        Defined
  7     Revolve Axis Option  Internal Centerline
  8     Direction           Side 2
  9     Angle               Defined
  9.1    Side One            Defined
  9.1.1  Side One Angle       None
  9.2    Side Two            Defined
  9.2.1  Side Two Angle       Variable
  9.2.2  Value               360.00

        SECTION NAME = S2D0011

        FEATURE'S DIMENSIONS:
        d185 = (Displayed:) 360
             (   Stored:) 360.0 ( 0.5, -0.5 )
        END ADD
      END IF
   END IF
END IF

IF BJ_1H=="PLASTIC"
   IF BJ_14H==35

      ADD FEATURE
      INTERNAL FEATURE ID  2752
```

PARENTS = 1(#1) 39(#8) 734(*) 2599(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0012

FEATURE'S DIMENSIONS:
d186 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d189 = (Displayed:) 51.83 Dia (weak)
(Stored:) 51.83056078856 (0.01, -0.01)
d190 = (Displayed:) 45.5 Dia (weak)

(Stored:) 45.5 (0.01, -0.01)
d191 = (Displayed:) 55.04 Dia (weak)
(Stored:) 55.03767725182 (0.01, -0.01)
d192 = (Displayed:) 133.1 (weak)
(Stored:) 133.0956889402 (0.5, -0.5)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 2854
PARENTS = 1(#1) 2599(*) 2752(*) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0013

FEATURE'S DIMENSIONS:
d193 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d194 = (Displayed:) 2.5
(Stored:) 2.5 (0.01, -0.01)
d195 = (Displayed:) 35 Dia
(Stored:) 35.0 (0.01, -0.01)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 2905
PARENTS = 1(#1) 2599(*) 2752(*) 2854(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined

```
 10.2.1  Side Two Angle       Variable
 10.2.2  Value                360.00

      SECTION NAME = S2D0014

      FEATURE'S DIMENSIONS:
      d196 = (Displayed:) 360
            (   Stored:) 360.0 ( 0.5, -0.5 )
      d197 = (Displayed:) 39.56 Dia (weak)
            (   Stored:) 39.56431911424 ( 0.01, -0.01 )
      d198 = (Displayed:) 39.74 Dia (weak)
            (   Stored:) 39.73566565472 ( 0.01, -0.01 )
      d199 = (Displayed:) 40.07 Dia (weak)
            (   Stored:) 40.06800784512 ( 0.01, -0.01 )
      d200 = (Displayed:) 47.16 Dia (weak)
            (   Stored:) 47.16039615974 ( 0.01, -0.01 )
      END ADD
   END IF
END IF

IF BJ_1H=="PLASTIC"
   IF BJ_3H=="ELLIPTICAL"
      IF BJ_14H==30

         ADD FEATURE
         INTERNAL FEATURE ID  2971
         PARENTS = 1(#1) 39(#8) 898(*) 2315(*) 5(#3)

PROTRUSION: Revolve

 NO.    ELEMENT NAME        INFO
 ---    -------------       -------------
  1     Feature Name        Defined
  2     Extrude Feat type   Solid
  3     Material            Add
  4     Section             Defined
  4.1   Setup Plane         Defined
```

```
 4.1.1  Sketching Plane      FRONT:F3(DATUM PLANE)
 4.1.2  View Direction       Side 1
 4.1.3  Orientation          Right
 4.1.4  Reference            RIGHT:F1(DATUM PLANE)
 4.2    Sketch               Defined
 5      Feature Form         Solid
 6      Revolve Axis         Defined
 7      Revolve Axis Option  Internal Centerline
 8      Direction            Side 2
 9      Angle                Defined
 9.1    Side One             Defined
 9.1.1  Side One Angle       None
 9.2    Side Two             Defined
 9.2.1  Side Two Angle       Variable
 9.2.2  Value                360.00

     SECTION NAME = S2D0015

     FEATURE'S DIMENSIONS:
     d201 = (Displayed:) 360
         (   Stored:) 360.0 ( 0.5, -0.5 )
     END ADD
   END IF
 END IF
END IF

IF BJ_1H=="PLASTIC"
 IF BJ_14H==40

   ADD FEATURE
   INTERNAL FEATURE ID  3022
   FEATURE WAS CREATED IN ASSEMBLY BALL_JOINT
   PARENTS = 1(#1) 39(#8) 423(#11) 494(#12) 5(#3)

CUT: Revolve

 NO.    ELEMENT NAME       INFO
```

```
---      -------------        -------------
 1       Feature Name          Defined
 2       Extrude Feat type     Solid
 3       Material              Remove
 4       Section               Defined
 4.1     Setup Plane           Defined
 4.1.1   Sketching Plane       FRONT:F3(DATUM PLANE)
 4.1.2   View Direction        Side 1
 4.1.3   Orientation           Right
 4.1.4   Reference             RIGHT:F1(DATUM PLANE)
 4.2     Sketch                Defined
 5       Feature Form          Solid
 6       Material Side         Side Two
 7       Revolve Axis          Defined
 8       Revolve Axis Option  Internal Centerline
 9       Direction             Side 2
 10      Angle                 Defined
 10.1    Side One              Defined
 10.1.1  Side One Angle        None
 10.2    Side Two              Defined
 10.2.1  Side Two Angle        Variable
 10.2.2  Value                 360.00

    SECTION NAME = S2D0016

    FEATURE'S DIMENSIONS:
    d202 = (Displayed:) 360
        (   Stored:) 360.0 ( 0.5, -0.5 )
    d203 = (Displayed:) 55
        (   Stored:) 55.0 ( 0.5, -0.5 )
    END ADD
  END IF
END IF

IF BJ_1H=="PLASTIC"
  IF BJ_14H==40
    IF BJ_3H=="ELLIPTICAL"
```

ADD FEATURE
INTERNAL FEATURE ID 3107
PARENTS = 1(#1) 39(#8) 423(#11) 3022(*) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0017

FEATURE'S DIMENSIONS:
d204 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
END ADD
END IF

```
   END IF
END IF

IF BJ_1H=="PLASTIC"
   IF BJ_14H==40

      ADD FEATURE
      INTERNAL FEATURE ID  3158
      PARENTS = 1(#1) 39(#8) 494(#12) 3022(*) 5(#3)

PROTRUSION: Revolve

 NO.    ELEMENT NAME        INFO
 ---    -------------       -------------
  1     Feature Name        Defined
  2     Extrude Feat type   Solid
  3     Material            Add
  4     Section             Defined
  4.1    Setup Plane        Defined
  4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
  4.1.2  View Direction     Side 1
  4.1.3  Orientation        Right
  4.1.4  Reference          RIGHT:F1(DATUM PLANE)
  4.2    Sketch             Defined
  5     Feature Form        Solid
  6     Revolve Axis        Defined
  7     Revolve Axis Option  Internal Centerline
  8     Direction           Side 2
  9     Angle               Defined
  9.1    Side One           Defined
  9.1.1  Side One Angle     None
  9.2    Side Two           Defined
  9.2.1  Side Two Angle     Variable
  9.2.2  Value              360.00

      SECTION NAME = S2D0018
```

FEATURE'S DIMENSIONS:

d205 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

d206 = (Displayed:) 2.5

(Stored:) 2.5 (0.01, -0.01)

d207 = (Displayed:) 40 Dia

(Stored:) 40.0 (0.01, -0.01)

END ADD

ADD FEATURE

INTERNAL FEATURE ID 3210

PARENTS = 1(#1) 39(#8) 494(#12) 3158(*) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined

```
 10.2.1  Side Two Angle       Variable
 10.2.2  Value              360.00

     SECTION NAME = S2D0019

     FEATURE'S DIMENSIONS:
     d208 = (Displayed:) 360
           (   Stored:) 360.0 ( 0.5, -0.5 )
     d209 = (Displayed:) 44.99 Dia (weak)
           (   Stored:) 44.98512095674 ( 0.01, -0.01 )
     d210 = (Displayed:) 45.03 Dia (weak)
           (   Stored:) 45.03122182243 ( 0.01, -0.01 )
     d211 = (Displayed:) 44.93 Dia (weak)
           (   Stored:) 44.92530949391 ( 0.01, -0.01 )
     d212 = (Displayed:) 51.12 Dia (weak)
           (   Stored:) 51.1214839474 ( 0.01, -0.01 )
     END ADD
   END IF
END IF

IF BJ_4H=="INNER_THREAD"

   ADD FEATURE (initial number 15)
   INTERNAL FEATURE ID  3379
   PARENTS = 3(#2) 90(#9) 1(#1) 5(#3)

CUT: Revolve

 NO.     ELEMENT NAME        INFO
 ---     -------------       -------------
  1      Feature Name        Defined
  2      Extrude Feat type   Solid
  3      Material            Remove
  4      Section             Defined
  4.1     Setup Plane         Defined
  4.1.1   Sketching Plane     FRONT:F3(DATUM PLANE)
  4.1.2   View Direction      Side 1
```

```
4.1.3   Orientation          Right
4.1.4   Reference            RIGHT:F1(DATUM PLANE)
4.2     Sketch               Defined
5       Feature Form         Solid
6       Material Side        Side Two
7       Revolve Axis         Defined
8       Revolve Axis Option  Internal Centerline
9       Direction            Side 2
10      Angle                Defined
10.1    Side One             Defined
10.1.1  Side One Angle       None
10.2    Side Two             Defined
10.2.1  Side Two Angle       Variable
10.2.2  Value                360.00

  SECTION NAME = S2D0003

  FEATURE IS IN LAYER(S) :
   02___PRT_ALL_AXES - OPERATION = SHOWN

  FEATURE'S DIMENSIONS:
  d217 = (Displayed:) 360
       (   Stored:) 360.0 ( 0.5, -0.5 )
  h_37 = (Displayed:) 16 Dia
       (   Stored:) 16.0 ( 0.01, -0.01 )
  d219 = (Displayed:) 118
       (   Stored:) 118.0 ( 0.5, -0.5 )
  h_36 = (Displayed:) 60
       (   Stored:) 60.0 ( 0.01, -0.01 )
  d221 = (Displayed:) 45
       (   Stored:) 45.0 ( 0.5, -0.5 )
  h_37_1 = (Displayed:) 17 Dia
         (   Stored:) 17.0 ( 0.01, -0.01 )
  END ADD
END IF

ADD FEATURE (initial number 16)
```

INTERNAL FEATURE ID 3707

PARENTS = 90(#9)

COSMETIC: Thread

```
NO. ELEMENT NAME    INFO                              STATUS
--- ------------ -----------                          ------
 1  Thread Surf                                       Defined
 2  Start Surf                                        Defined
 3  Direction                                         Defined
 4  ThreadLength Blind, depth = 50                    Defined
 5  Major Diam    Value = 21.6000                     Defined
 6  Note Params                                       Defined
```

FEATURE'S DIMENSIONS:

d230 = (Displayed:) 50

(Stored:) 50.0 (0.00000000001, -0.00000000001)

d231 = (Displayed:) 21.6

(Stored:) 21.6 (0.01, -0.01)

END ADD

MASSPROP

END MASSPROP

APPENDIX B

Ball Pin Program

```
VERSION 5.0
REVNUM 1831
LISTING FOR PART BALL_PIN

INPUT
 BJ_1 STRING
 BJ_2 STRING
 BJ_3 STRING
 BJ_4 STRING
 BJ_5 NUMBER
 BJ_6 NUMBER
 BJ_7 NUMBER
 BJ_8 NUMBER
 BJ_9 NUMBER
 BJ_10 NUMBER
 BJ_11 NUMBER
 BJ_12 NUMBER
 BJ_13 NUMBER
 BJ_14 NUMBER
 BJ_15 NUMBER
 BJ_16 NUMBER
 BJ_17 NUMBER
 BJ_18 NUMBER
 BJ_19 NUMBER
 BJ_20 NUMBER
 BJ_21 NUMBER
 BJ_22 NUMBER
 BJ_23 STRING
END INPUT

RELATIONS
BP_15=BJ_11
BP_16=BJ_10
BP_17=BJ_20
```

```
IF BJ_14==40
BP_1=40
BP_2=18.5
BP_3=BJ_6+BJ_7
BP_4=BJ_7+5+15.5
BP_5=BJ_7+5
BP_6=BJ_7
BP_7=4.3
BP_8=18.5+BJ_6+BJ_7+BJ_8
BP_9=24
BP_10=BJ_13
BP_11=BJ_13-(BJ_7/BJ_12)
BP_12=BJ_9-2
BP_13=BJ_9-2
BP_14=BJ_9
BP_18=3
ENDIF

IF BJ_14==35
BP_1=35
BP_2=16
BP_3=BJ_6+BJ_7
BP_4=BJ_7+5+13.5
BP_5=BJ_7+5
BP_6=BJ_7
BP_7=4.3
BP_8=16+BJ_6+BJ_7+BJ_8
BP_9=21
BP_10=BJ_13
BP_11=BJ_13-(BJ_7/BJ_12)
BP_12=BJ_9-2
BP_13=BJ_9-2
BP_14=BJ_9
BP_18=2
ENDIF
```

```
IF BJ_14==30
BP_1=30
BP_2=13
BP_3=BJ_6+BJ_7
BP_4=BJ_7+5+12.5
BP_5=BJ_7+4
BP_6=BJ_7
BP_7=4.3
BP_8=13+BJ_6+BJ_7+BJ_8
BP_9=18
BP_10=BJ_13
BP_11=BJ_13-(BJ_7/BJ_12)
BP_12=BJ_9-2
BP_13=BJ_9-2
BP_14=BJ_9
BP_18=1.5
ENDIF

IF BJ_14==27
BP_1=27
BP_2=12.2
BP_3=BJ_6+BJ_7
BP_4=BJ_7+5+11.5
BP_5=BJ_7+4
BP_6=BJ_7
BP_7=4
BP_8=12.2+BJ_6+BJ_7+BJ_8
BP_9=16
BP_10=BJ_13
BP_11=BJ_13-(BJ_7/BJ_12)
BP_12=BJ_9-2
BP_13=BJ_9-2
BP_14=BJ_9
BP_18=1
ENDIF
END RELATIONS
```

ADD FEATURE (initial number 1)
INTERNAL FEATURE ID 1

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	X Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = RIGHT

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 2)
INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis
3	Flip Datum Dir	Defined
4	Fit	Defined

4.1 Fit Type Default

NAME = TOP

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD
ADD FEATURE (initial number 4)
INTERNAL FEATURE ID 7
TYPE = COORDINATE SYSTEM
NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :
05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN
05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 5)
INTERNAL FEATURE ID 39
PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0005

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d0 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
bp_1 = (Displayed:) 40 Dia
(Stored:) 40.0 (0.001, -0.001)
BP_18 = (Displayed:) 3bp_18R
(Stored:) 3.0 (0.01, -0.01)
d3 = (Displayed:) 25
(Stored:) 25.0 (0.5, -0.5)
bp_14 = (Displayed:) 24 Dia
(Stored:) 24.0 (0.001, -0.001)
d5 = (Displayed:) 1.5
(Stored:) 1.5 (0.001, -0.001)
d6 = (Displayed:) 45
(Stored:) 45.0 (0.5, -0.5)
bp_13 = (Displayed:) 22 Dia
(Stored:) 22.0 (0.001, -0.001)
bp_9 = (Displayed:) 24 Dia
(Stored:) 24.0 (0.001, -0.001)
bp_7 = (Displayed:) 4.3@bp_7D
(Stored:) 4.3 (0.001, -0.001)
bp_11 = (Displayed:) 26.83 Dia
(Stored:) 26.8333333333 (0.001, -0.001)
d11 = (Displayed:) 30
(Stored:) 30.0 (0.5, -0.5)
bp_12 = (Displayed:) 22 Dia
(Stored:) 22.0 (0.001, -0.001)
bp_5 = (Displayed:) 24bp_5
(Stored:) 24.0 (0.01, -0.01)
bp_6 = (Displayed:) 19bp_6
(Stored:) 19.0 (0.01, -0.01)
bp_10 = (Displayed:) 30 Dia
(Stored:) 30.0 (0.01, -0.01)
bp_2 = (Displayed:) 18.5bp_2

```
    (   Stored:) 18.5 ( 0.01, -0.01 )
bp_3 = (Displayed:) 48 bp_3
    (   Stored:) 48.0 ( 0.01, -0.01 )
bp_4 = (Displayed:) 39.5bp_4
    (   Stored:) 39.5 ( 0.01, -0.01 )
bp_8 = (Displayed:) 92.5bp_8
    (   Stored:) 92.5 ( 0.01, -0.01 )
END ADD

IF BJ_2=="CASTLE_NUT"

  ADD FEATURE (initial number 6)
  INTERNAL FEATURE ID  170
  PARENTS = 1(#1) 39(#5) 5(#3)


CUT: Extrude

 NO.   ELEMENT NAME      INFO
 ---    -------------     -------------
  1     Feature Name      Defined
  2     Extrude Feat type  Solid
  3     Material          Remove
  4     Section           Defined
  4.1    Setup Plane       Defined
  4.1.1  Sketching Plane   FRONT:F3(DATUM PLANE)
  4.1.2  View Direction    Side 1
  4.1.3  Orientation       Right
  4.1.4  Reference         RIGHT:F1(DATUM PLANE)
  4.2    Sketch            Defined
  5     Feature Form      Solid
  6     Material Side     Side Two
  7     Direction         Side 2
  8     Depth             Defined
  8.1    Side One          Defined
  8.1.1  Side One Depth    None
  8.2    Side Two          Defined
```

8.2.1 Side Two Depth   Symmetric
8.2.2 Value   60.00

SECTION NAME = S2D0005

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d21 = (Displayed:) 60
(Stored:) 60.0 (0.01, -0.01)
BP_16 = (Displayed:) 6 Dia
(Stored:) 6.0 (0.01, -0.01)
BP_15 = (Displayed:) 19bp_15
(Stored:) 19.0 (0.01, -0.01)
END ADD
END IF

IF BJ_2=="SELF_LOCKING"

ADD FEATURE
INTERNAL FEATURE ID 198
PARENTS = 1(#1) 39(#5) 5(#3)

CUT: Revolve

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)

4.2 Sketch Defined
5 Feature Form Solid
6 Material Side Side Two
7 Revolve Axis Defined
8 Revolve Axis Option Internal Centerline
9 Direction Side 2
10 Angle Defined
10.1 Side One Defined
10.1.1 Side One Angle None
10.2 Side Two Defined
10.2.1 Side Two Angle Variable
10.2.2 Value 360.00

SECTION NAME = S2D0006

FEATURE'S DIMENSIONS:
d24 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
BP_17 = (Displayed:) 8bp_17
(Stored:) 8.0 (0.01, -0.01)
d26 = (Displayed:) 120
(Stored:) 120.0 (0.5, -0.5)
d27 = (Displayed:) 5
(Stored:) 5.0 (0.01, -0.01)
END ADD

ADD FEATURE
INTERNAL FEATURE ID 247
PARENTS = 5(#3) 198(*) 39(#5)

CUT: Extrude

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove

4 Section Defined
4.1 Setup Plane Defined
4.1.1 Sketching Plane Surf:F5(REVOLVE_1)
4.1.2 View Direction Side 1
4.1.3 Orientation Top
4.1.4 Reference Surf:F5(REVOLVE_1)
4.2 Sketch Defined
5 Feature Form Solid
6 Material Side Side Two
7 Direction Side 1
8 Depth Defined
8.1 Side One Defined
8.1.1 Side One Depth None
8.2 Side Two Defined
8.2.1 Side Two Depth To Reference
8.2.2 Reference Edge

SECTION NAME = S2D0007

FEATURE'S DIMENSIONS:
d29 = (Displayed:) 120
(Stored:) 120.0 (0.5, -0.5)
d30 = (Displayed:) 120
(Stored:) 120.0 (0.5, -0.5)
END ADD
END IF

ADD FEATURE (initial number 7)
INTERNAL FEATURE ID 500
PARENTS = 1(#1) 3(#2)

DATUM AXIS

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined

2.1 Constraint #1 Defined
2.1.1 Type Through
2.1.2 Reference RIGHT:F1(DATUM PLANE)
2.2 Constraint #2 Defined
2.2.1 Type Through
2.2.2 Reference TOP:F2(DATUM PLANE)
3 Fit Defined
3.1 Fit Type Default
NAME = A_4
FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN
END ADD

ADD FEATURE (initial number 8)
INTERNAL FEATURE ID 508
PARENTS = 3(#2) 5(#3)

DATUM AXIS

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Type	Through
2.1.2	Reference	TOP:F2(DATUM PLANE)
2.2	Constraint #2	Defined
2.2.1	Type	Through
2.2.2	Reference	FRONT:F3(DATUM PLANE)
3	Fit	Defined
3.1	Fit Type	Default

NAME = A_5

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

END ADD

MASSPROP
END MASSPROP

APPENDIX C

Metal Bearing Program

```
VERSION 5.0
REVNUM 404
LISTING FOR PART METAL_BEARING_27_30_35

INPUT
 BJ_14MB NUMBER
 "ENTER SPHERE DIAMETER"
END INPUT

RELATIONS
IF BJ_14MB==35
MB1=40
MB2=1.5
MB3=17.5
MB4=6.5
MB5=35
MB6=8
ENDIF

IF BJ_14MB==30
MB1=35.5
MB2=1.5
MB3=15.5
MB4=6.5
MB5=30
MB6=8
ENDIF

IF BJ_14MB!=35
IF BJ_14MB!=30
MB1=32.5
MB2=1.5
MB3=14
MB4=5
```

MB5=27

MB6=8

ENDIF

ENDIF

END RELATIONS

ADD FEATURE (initial number 1)

INTERNAL FEATURE ID 1

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	X Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = RIGHT

FEATURE IS IN LAYER(S) :

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 2)

INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined

2.1 Constraint #1 Defined
2.1.1 Constr Type Y Axis
3 Flip Datum Dir Defined
4 Fit Defined
4.1 Fit Type Default

NAME = TOP

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5
DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD
ADD FEATURE (initial number 4)
INTERNAL FEATURE ID 7

TYPE = COORDINATE SYSTEM
NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :
05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN
05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD
ADD FEATURE (initial number 5)
INTERNAL FEATURE ID 39
PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0002

FEATURE IS IN LAYER(S) :

02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:

d0 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

mb3 = (Displayed:) 14

(Stored:) 14.0 (0.01, -0.01)

mb5 = (Displayed:) 27 Dia

(Stored:) 27.0 (0.01, -0.01)

mb1 = (Displayed:) 32.5 Dia

(Stored:) 32.5 (0.01, -0.01)

mb2 = (Displayed:) 1.5

(Stored:) 1.5 (0.01, -0.01)

mb4 = (Displayed:) 5

(Stored:) 5.0 (0.01, -0.01)

mb6 = (Displayed:) 8 Dia

(Stored:) 8.0 (0.01, -0.01)

END ADD

ADD FEATURE (initial number 6)

INTERNAL FEATURE ID 114

PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Sets	4 Sets
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain

2.1.3.2 Curve Collection 2 Selections

2.1.4 Spine Defined

2.1.4.1 Ball/Spine Rolling Ball

2.1.5 Extend Surfaces Disable

2.1.6 Radii 1 Points

2.1.6.1 Rad 0 Defined

2.1.6.1.1 D1 Defined

2.1.6.1.1.1 Distance type Enter Value

2.1.6.1.1.2 Distance value 1.00

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

2.2 Set 1 Defined

2.2.1 Shape options Constant

2.2.2 Conic Defined

2.2.2.1 Conic Type Plain

2.2.3 References Defined

2.2.3.1 Reference type Edge Chain

2.2.3.2 Curve Collection 2 Selections

2.2.4 Spine Defined

2.2.4.1 Ball/Spine Rolling Ball

2.2.5 Extend Surfaces Disable

2.2.6 Radii 1 Points

2.2.6.1 Rad 0 Defined

2.2.6.1.1 D1 Defined

2.2.6.1.1.1 Distance type Enter Value

2.2.6.1.1.2 Distance value 1.00

2.2.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

2.3 Set 2 Defined

2.3.1 Shape options Constant

2.3.2 Conic Defined

2.3.2.1 Conic Type Plain

2.3.3 References Defined

2.3.3.1 Reference type Edge Chain

2.3.3.2 Curve Collection 2 Selections

2.3.4 Spine Defined

2.3.4.1 Ball/Spine Rolling Ball

2.3.5 Extend Surfaces Disable

2.3.6 Radii 1 Points

2.3.6.1 Rad 0 Defined
2.3.6.1.1 D1 Defined
2.3.6.1.1.1 Distance type Enter Value
2.3.6.1.1.2 Distance value 1.00
2.3.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
2.4 Set 3 Defined
2.4.1 Shape options Constant
2.4.2 Conic Defined
2.4.2.1 Conic Type Plain
2.4.3 References Defined
2.4.3.1 Reference type Edge Chain
2.4.3.2 Curve Collection 2 Selections
2.4.4 Spine Defined
2.4.4.1 Ball/Spine Rolling Ball
2.4.5 Extend Surfaces Disable
2.4.6 Radii 1 Points
2.4.6.1 Rad 0 Defined
2.4.6.1.1 D1 Defined
2.4.6.1.1.1 Distance type Enter Value
2.4.6.1.1.2 Distance value 1.00
2.4.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d10 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
d11 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
d12 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
d13 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
END ADD

MASSPROP
END MASSPROP

APPENDIX D

Spring Program

```
VERSION 5.0
REVNUM 1393
LISTING FOR PART SPRING

INPUT
 BJ_14B NUMBER
END INPUT

RELATIONS
IF BJ_14B==40
D17=6
D18=12.1
D19=31
D20=38.3
D21=5
D24=4.5
D81=6.1
D82=6.1
ENDIF
IF BJ_14B==35
D17=7
D18=19
D19=17
D20=38
D21=5
D24=4
D81=7
D82=7
ENDIF
IF BJ_14B==30
D17=4.5
D18=9
D19=17.5
D20=31
```

```
D21=5
D24=3.5
D81=4.5
D82=4.5
ENDIF
IF BJ_14B==27
D17=3.75
D18=7
D19=16
D20=28
D21=5
D24=3
D81=3.75
D82=3.75
ENDIF
END RELATIONS
 ADD FEATURE (initial number 1)
 INTERNAL FEATURE ID  1
 DATUM PLANE

  NO.     ELEMENT NAME    INFO
  ---     -------------   -------------
   1      Feature Name    Defined
   2      Constraints     Defined
   2.1    Constraint #1   Defined
   2.1.1  Constr Type     X Axis
   3      Flip Datum Dir  Defined
   4      Fit             Defined
   4.1    Fit Type        Default

 NAME = RIGHT

   FEATURE IS IN LAYER(S) :
     01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
     01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

 END ADD
```

ADD FEATURE (initial number 2)
INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = TOP

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)

INTERNAL FEATURE ID 7

TYPE = COORDINATE SYSTEM

NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :

05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN

05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 5)

INTERNAL FEATURE ID 102

PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Helical Sweep

NO.	ELEMENT NAME	INFO	STATUS
---	------------	-----------	------
1	Attributes	Constant Pitch, Right Handed, Thru Axis	Defined
2	Swp Profile	Sk. plane - Surface FRONT of feat #3 (DATUM PLANE)	Defined
3	Pitch	Pitch = 5	Defined
4	Section		Defined

SECTION NAME = S2D0002

SECTION NAME = S2D0002

FEATURE IS IN LAYER(S) :
7_ALL_FEATURES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d17 = (Displayed:) 6
(Stored:) 6.0 (0.001, -0.001)
d18 = (Displayed:) 12.1
(Stored:) 12.1 (0.001, -0.001)
d19 = (Displayed:) 31 Dia
(Stored:) 31.0 (0.001, -0.001)
d20 = (Displayed:) 38.3 Dia
(Stored:) 38.3 (0.001, -0.001)
d21 = (Displayed:) 5PITCH
(Stored:) 5.0 (0.001, -0.001)
d24 = (Displayed:) 4.5 Dia
(Stored:) 4.5 (0.001, -0.001)
END ADD
ADD FEATURE (initial number 6)
INTERNAL FEATURE ID 697
PARENTS = 3(#2)

DATUM PLANE

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Offset
2.1.2	Constr References	Surface TOP of feat #2 (DATUM PLANE)
2.1.3	Constr Ref Offset	Value = 6.1000
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = DTM1

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
7_ALL_FEATURES - OPERATION = SHOWN
1_ALL_PLANES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d81 = (Displayed:) 6.1
(Stored:) 6.1 (0.001, -0.001)
END ADD

ADD FEATURE (initial number 7)
INTERNAL FEATURE ID 700
PARENTS = 3(#2)

DATUM PLANE

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Offset
2.1.2	Constr References	Surface TOP of feat #2 (DATUM PLANE)
2.1.3	Constr Ref Offset	Value = -6.1000
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = DTM2

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
7_ALL_FEATURES - OPERATION = SHOWN
1_ALL_PLANES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d82 = (Displayed:) 6.1

(Stored:) -6.1 (0.001, -0.001)
END ADD

ADD FEATURE (initial number 8)
INTERNAL FEATURE ID 515
PARENTS = 1(#1) 3(#2) 5(#3) 697(#6) 700(#7)

CUT: Extrude

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Top
4.1.4	Reference	TOP:F2(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Direction	Side 1
8	Depth	Defined
8.1	Side One	Defined
8.1.1	Side One Depth	None
8.2	Side Two	Defined
8.2.1	Side Two Depth	Symmetric
8.2.2	Value	54.00

SECTION NAME = S2D0001

FEATURE IS IN LAYER(S) :
7_ALL_FEATURES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:

d83 = (Displayed:) 4.673 (weak)
(Stored:) 4.672727097234 (0.001, -0.001)
d84 = (Displayed:) 9.343 (weak)
(Stored:) 9.343102569345 (0.001, -0.001)
d85 = (Displayed:) 52.432 (weak)
(Stored:) 52.43165933316 (0.001, -0.001)
d86 = (Displayed:) 25.27 (weak)
(Stored:) 25.26986899216 (0.001, -0.001)
d90 = (Displayed:) 54
(Stored:) 54.0 (0.01, -0.01)
END ADD
ADD FEATURE (initial number 9)
INTERNAL FEATURE ID 745
PARENTS = 1(#1) 5(#3)
DATUM AXIS

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Type	Through
2.1.2	Reference	RIGHT:F1(DATUM PLANE)
2.2	Constraint #2	Defined
2.2.1	Type	Through
2.2.2	Reference	FRONT:F3(DATUM PLANE)
3	Fit	Defined
3.1	Fit Type	Default

NAME = A_11

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN
7_ALL_FEATURES - OPERATION = SHOWN
3_ALL_AXES - OPERATION = SHOWN

END ADD

MASSPROP
END MASSPROP

APPENDIX E

End Cap Program

```
VERSION 5.0
REVNUM 407
LISTING FOR PART END_CAP_MB_27_30_35

INPUT
 BJ_14ECMB NUMBER
END INPUT

RELATIONS
IF BJ_14ECMB==35
ECMB1=43.9
ECMB2=22.1
ECMB3=2.5
ECMB4=9
ECMB5=40
ENDIF

IF BJ_14ECMB==30
ECMB1=37.9
ECMB2=21
ECMB3=3
ECMB4=6.5
ECMB5=45
ENDIF

IF BJ_14ECMB!=35
IF BJ_14ECMB!=30
ECMB1=33.9
ECMB2=21
ECMB3=2.5
ECMB4=6
ECMB5=30
ENDIF
ENDIF
```

END RELATIONS

ADD FEATURE (initial number 1)

INTERNAL FEATURE ID 1

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	X Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = RIGHT

FEATURE IS IN LAYER(S) :

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 2)

INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = TOP

FEATURE IS IN LAYER(S) :

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)

INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)

INTERNAL FEATURE ID 7

TYPE = COORDINATE SYSTEM

NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :

05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN
05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 5)
INTERNAL FEATURE ID 39
PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable
9.2.2	Value	360.00

SECTION NAME = S2D0001

FEATURE IS IN LAYER(S) :

02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:

d0 = (Displayed:) 360

(Stored:) 360.0 (0.5, -0.5)

ecmb1 = (Displayed:) 33.9 Dia

(Stored:) 33.9 (0.01, -0.01)

ecmb3 = (Displayed:) 2.5

(Stored:) 2.5 (0.01, -0.01)

ecmb4 = (Displayed:) 6

(Stored:) 6.0 (0.01, -0.01)

ecmb5 = (Displayed:) 30

(Stored:) 30.0 (0.5, -0.5)

ecmb2 = (Displayed:) 21 Dia

(Stored:) 21.0 (0.01, -0.01)

END ADD

ADD FEATURE (initial number 6)

INTERNAL FEATURE ID 114

PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Sets	4 Sets
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined

2.1.4.1 Ball/Spine Rolling Ball
2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined
2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 3.00
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
2.2 Set 1 Defined
2.2.1 Shape options Constant
2.2.2 Conic Defined
2.2.2.1 Conic Type Plain
2.2.3 References Defined
2.2.3.1 Reference type Edge Chain
2.2.3.2 Curve Collection 2 Selections
2.2.4 Spine Defined
2.2.4.1 Ball/Spine Rolling Ball
2.2.5 Extend Surfaces Disable
2.2.6 Radii 1 Points
2.2.6.1 Rad 0 Defined
2.2.6.1.1 D1 Defined
2.2.6.1.1.1 Distance type Enter Value
2.2.6.1.1.2 Distance value 3.00
2.2.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
2.3 Set 2 Defined
2.3.1 Shape options Constant
2.3.2 Conic Defined
2.3.2.1 Conic Type Plain
2.3.3 References Defined
2.3.3.1 Reference type Edge Chain
2.3.3.2 Curve Collection 2 Selections
2.3.4 Spine Defined
2.3.4.1 Ball/Spine Rolling Ball
2.3.5 Extend Surfaces Disable
2.3.6 Radii 1 Points
2.3.6.1 Rad 0 Defined
2.3.6.1.1 D1 Defined

2.3.6.1.1.1 Distance type Enter Value
2.3.6.1.1.2 Distance value 3.00
2.3.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
2.4 Set 3 Defined
2.4.1 Shape options Constant
2.4.2 Conic Defined
2.4.2.1 Conic Type Plain
2.4.3 References Defined
2.4.3.1 Reference type Edge Chain
2.4.3.2 Curve Collection 2 Selections
2.4.4 Spine Defined
2.4.4.1 Ball/Spine Rolling Ball
2.4.5 Extend Surfaces Disable
2.4.6 Radii 1 Points
2.4.6.1 Rad 0 Defined
2.4.6.1.1 D1 Defined
2.4.6.1.1.1 Distance type Enter Value
2.4.6.1.1.2 Distance value 3.00
2.4.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d8 = (Displayed:) 3R
(Stored:) 3.0 (0.01, -0.01)
d9 = (Displayed:) 3R
(Stored:) 3.0 (0.01, -0.01)
d10 = (Displayed:) 3R
(Stored:) 3.0 (0.01, -0.01)
d11 = (Displayed:) 3R
(Stored:) 3.0 (0.01, -0.01)
END ADD

MASSPROP
END MASSPROP

APPENDIX F

Clamping Ring Sheet Program

```
VERSION 5.0
REVNUM 876
LISTING FOR PART CLAMPING_RING_SHEET

INPUT
 BJ_14CRS NUMBER
END INPUT

RELATIONS
IF BJ_14CRS==40
CRS_1=60.6
CRS_2=50.2
CRS_3=5.75
CRS_4=25
CRS_5=1.66
CRS_6=0.64
ENDIF

IF BJ_14CRS==35
CRS_1=52.2
CRS_2=44.2
CRS_3=5
CRS_4=25
CRS_5=1.5
CRS_6=0.64
ENDIF

IF BJ_14CRS==30
CRS_1=45.2
CRS_2=39.2
CRS_3=4.5
CRS_4=25
CRS_5=1
CRS_6=0.5
```

```
ENDIF
END RELATIONS

 ADD FEATURE (initial number 1)
 INTERNAL FEATURE ID  1

 DATUM PLANE

  NO.    ELEMENT NAME    INFO
  ---    -------------   -------------
   1     Feature Name    Defined
   2     Constraints     Defined
   2.1    Constraint #1   Defined
   2.1.1  Constr Type     X Axis
   3     Flip Datum Dir  Defined
   4     Fit             Defined
   4.1    Fit Type        Default

 NAME = RIGHT

   FEATURE IS IN LAYER(S) :
     01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
     01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

 END ADD

 ADD FEATURE (initial number 2)
 INTERNAL FEATURE ID  3

 DATUM PLANE

  NO.    ELEMENT NAME    INFO
  ---    -------------   -------------
   1     Feature Name    Defined
   2     Constraints     Defined
   2.1    Constraint #1   Defined
   2.1.1  Constr Type     Y Axis
```

3 Flip Datum Dir Defined
4 Fit Defined
4.1 Fit Type Default

NAME = TOP

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)
INTERNAL FEATURE ID 7

TYPE = COORDINATE SYSTEM
NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :
05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN
05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 5)
INTERNAL FEATURE ID 39
PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined
7	Revolve Axis Option	Internal Centerline
8	Direction	Side 2
9	Angle	Defined
9.1	Side One	Defined
9.1.1	Side One Angle	None
9.2	Side Two	Defined
9.2.1	Side Two Angle	Variable

9.2.2 Value 360.00

SECTION NAME = S2D0016

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d0 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
crs_2 = (Displayed:) 50.2 Dia
(Stored:) 50.2 (0.01, -0.01)
crs_6 = (Displayed:) 0.64
(Stored:) 0.64 (0.01, -0.01)
crs_1 = (Displayed:) 60.6 Dia
(Stored:) 60.6 (0.01, -0.01)
crs_5 = (Displayed:) 1.66
(Stored:) 1.66 (0.01, -0.01)
crs_3 = (Displayed:) 5.75
(Stored:) 5.75 (0.01, -0.01)
crs_4 = (Displayed:) 25
(Stored:) 25.0 (0.5, -0.5)
END ADD

IF BJ_14CRS==40

ADD FEATURE (initial number 6)
INTERNAL FEATURE ID 114
PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	1 Set
2.1	Set 0	Defined

2.1.1 Shape options Constant

2.1.2 Conic Defined

2.1.2.1 Conic Type Plain

2.1.3 References Defined

2.1.3.1 Reference type Edge Chain

2.1.3.2 Curve Collection 2 Selections

2.1.4 Spine Defined

2.1.4.1 Ball/Spine Rolling Ball

2.1.5 Extend Surfaces Disable

2.1.6 Radii 1 Points

2.1.6.1 Rad 0 Defined

2.1.6.1.1 D1 Defined

2.1.6.1.1.1 Distance type Enter Value

2.1.6.1.1.2 Distance value 2.00

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

3 Attach type Make Solid

4 Transitions Defined

FEATURE'S DIMENSIONS:

d9 = (Displayed:) 2R

(Stored:) 2.0 (0.01, -0.01)

END ADD

ADD FEATURE (initial number 7)

INTERNAL FEATURE ID 154

PARENTS = 39(#5)

ROUND: General

NO. ELEMENT NAME INFO

--- ------------- -------------

1 Feature Name Defined

2 Sets 1 Set

2.1 Set 0 Defined

2.1.1 Shape options Constant

2.1.2 Conic Defined

2.1.2.1 Conic Type Plain

2.1.3 References Defined

2.1.3.1 Reference type Edge Chain

2.1.3.2 Curve Collection 2 Selections

2.1.4 Spine Defined

2.1.4.1 Ball/Spine Rolling Ball

2.1.5 Extend Surfaces Disable

2.1.6 Radii 1 Points

2.1.6.1 Rad 0 Defined

2.1.6.1.1 D1 Defined

2.1.6.1.1.1 Distance type Enter Value

2.1.6.1.1.2 Distance value 1.00

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

3 Attach type Make Solid

4 Transitions Defined

FEATURE'S DIMENSIONS:

d11 = (Displayed:) 1R

(Stored:) 1.0 (0.01, -0.01)

END ADD

ADD FEATURE (initial number 8)

INTERNAL FEATURE ID 174

PARENTS = 39(#5)

ROUND: General

NO. ELEMENT NAME INFO

--- ------------- -------------

1 Feature Name Defined

2 Sets 1 Set

2.1 Set 0 Defined

2.1.1 Shape options Constant

2.1.2 Conic Defined

2.1.2.1 Conic Type Plain

2.1.3 References Defined

2.1.3.1 Reference type Edge Chain

2.1.3.2 Curve Collection 2 Selections

2.1.4 Spine Defined
2.1.4.1 Ball/Spine Rolling Ball
2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined
2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 1.70
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d13 = (Displayed:) 1.7R
(Stored:) 1.7 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 9)
INTERNAL FEATURE ID 294
PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	1 Set
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined
2.1.4.1	Ball/Spine	Rolling Ball

2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined
2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 2.00
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d17 = (Displayed:) 2R
(Stored:) 2.0 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 10)
INTERNAL FEATURE ID 318
PARENTS = 39(#5)

ROUND: General

NO. ELEMENT NAME INFO
--- ------------- -------------
1 Feature Name Defined
2 Sets 2 Sets
2.1 Set 0 Defined
2.1.1 Shape options Constant
2.1.2 Conic Defined
2.1.2.1 Conic Type Plain
2.1.3 References Defined
2.1.3.1 Reference type Edge Chain
2.1.3.2 Curve Collection 2 Selections
2.1.4 Spine Defined
2.1.4.1 Ball/Spine Rolling Ball
2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined

2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 1.00
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
2.2 Set 1 Defined
2.2.1 Shape options Constant
2.2.2 Conic Defined
2.2.2.1 Conic Type Plain
2.2.3 References Defined
2.2.3.1 Reference type Edge Chain
2.2.3.2 Curve Collection 2 Selections
2.2.4 Spine Defined
2.2.4.1 Ball/Spine Rolling Ball
2.2.5 Extend Surfaces Disable
2.2.6 Radii 1 Points
2.2.6.1 Rad 0 Defined
2.2.6.1.1 D1 Defined
2.2.6.1.1.1 Distance type Enter Value
2.2.6.1.1.2 Distance value 1.00
2.2.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined
FEATURE'S DIMENSIONS:
d18 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
d19 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
END ADD
END IF

IF BJ_14CRS==35

ADD FEATURE
INTERNAL FEATURE ID 398
PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Sets	4 Sets
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined
2.1.4.1	Ball/Spine	Rolling Ball
2.1.5	Extend Surfaces	Disable
2.1.6	Radii	1 Points
2.1.6.1	Rad 0	Defined
2.1.6.1.1	D1	Defined
2.1.6.1.1.1	Distance type	Enter Value
2.1.6.1.1.2	Distance value	1.50
2.1.7	Pieces	1 of 1 Included, 0 Trimmed, 0 Extended
2.2	Set 1	Defined
2.2.1	Shape options	Constant
2.2.2	Conic	Defined
2.2.2.1	Conic Type	Plain
2.2.3	References	Defined
2.2.3.1	Reference type	Edge Chain
2.2.3.2	Curve Collection	2 Selections
2.2.4	Spine	Defined
2.2.4.1	Ball/Spine	Rolling Ball
2.2.5	Extend Surfaces	Disable
2.2.6	Radii	1 Points
2.2.6.1	Rad 0	Defined
2.2.6.1.1	D1	Defined
2.2.6.1.1.1	Distance type	Enter Value
2.2.6.1.1.2	Distance value	1.50
2.2.7	Pieces	1 of 1 Included, 0 Trimmed, 0 Extended

2.3 Set 2 Defined
2.3.1 Shape options Constant
2.3.2 Conic Defined
2.3.2.1 Conic Type Plain
2.3.3 References Defined
2.3.3.1 Reference type Edge Chain
2.3.3.2 Curve Collection 2 Selections
2.3.4 Spine Defined
2.3.4.1 Ball/Spine Rolling Ball
2.3.5 Extend Surfaces Disable
2.3.6 Radii 1 Points
2.3.6.1 Rad 0 Defined
2.3.6.1.1 D1 Defined
2.3.6.1.1.1 Distance type Enter Value
2.3.6.1.1.2 Distance value 1.50
2.3.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
2.4 Set 3 Defined
2.4.1 Shape options Constant
2.4.2 Conic Defined
2.4.2.1 Conic Type Plain
2.4.3 References Defined
2.4.3.1 Reference type Edge Chain
2.4.3.2 Curve Collection 2 Selections
2.4.4 Spine Defined
2.4.4.1 Ball/Spine Rolling Ball
2.4.5 Extend Surfaces Disable
2.4.6 Radii 1 Points
2.4.6.1 Rad 0 Defined
2.4.6.1.1 D1 Defined
2.4.6.1.1.1 Distance type Enter Value
2.4.6.1.1.2 Distance value 1.50
2.4.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

```
   FEATURE'S DIMENSIONS:
   d20 = (Displayed:) 1.5R
        (   Stored:) 1.5 ( 0.01, -0.01 )
   d21 = (Displayed:) 1.5R
        (   Stored:) 1.5 ( 0.01, -0.01 )
   d22 = (Displayed:) 1.5R
        (   Stored:) 1.5 ( 0.01, -0.01 )
   d23 = (Displayed:) 1.5R
        (   Stored:) 1.5 ( 0.01, -0.01 )
   END ADD
END IF

IF BJ_14CRS==30

   ADD FEATURE
   INTERNAL FEATURE ID  466
   PARENTS = 39(#5)


ROUND: General

 NO.           ELEMENT NAME      INFO
 ---           -------------     -------------
  1            Feature Name      Defined
  2            Sets              3 Sets
  2.1           Set 0            Defined
  2.1.1         Shape options     Constant
  2.1.2         Conic             Defined
  2.1.2.1        Conic Type        Plain
  2.1.3         References                              Defined
  2.1.3.1        Reference type    Edge Chain
  2.1.3.2        Curve Collection  2 Selections
  2.1.4         Spine             Defined
  2.1.4.1        Ball/Spine        Rolling Ball
  2.1.5         Extend Surfaces   Disable
  2.1.6         Radii                                   1 Points
  2.1.6.1        Rad 0                                         Defined
```

2.1.6.1.1 D1 Defined

2.1.6.1.1.1 Distance type Enter Value

2.1.6.1.1.2 Distance value 1.00

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

2.2 Set 1 Defined

2.2.1 Shape options Constant

2.2.2 Conic Defined

2.2.2.1 Conic Type Plain

2.2.3 References Defined

2.2.3.1 Reference type Edge Chain

2.2.3.2 Curve Collection 2 Selections

2.2.4 Spine Defined

2.2.4.1 Ball/Spine Rolling Ball

2.2.5 Extend Surfaces Disable

2.2.6 Radii 1 Points

2.2.6.1 Rad 0 Defined

2.2.6.1.1 D1 Defined

2.2.6.1.1.1 Distance type Enter Value

2.2.6.1.1.2 Distance value 1.00

2.2.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

2.3 Set 2 Defined

2.3.1 Shape options Constant

2.3.2 Conic Defined

2.3.2.1 Conic Type Plain

2.3.3 References Defined

2.3.3.1 Reference type Edge Chain

2.3.3.2 Curve Collection 2 Selections

2.3.4 Spine Defined

2.3.4.1 Ball/Spine Rolling Ball

2.3.5 Extend Surfaces Disable

2.3.6 Radii 1 Points

2.3.6.1 Rad 0 Defined

2.3.6.1.1 D1 Defined

2.3.6.1.1.1 Distance type Enter Value

2.3.6.1.1.2 Distance value 1.00

2.3.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

3 Attach type Make Solid

4 Transitions Defined

FEATURE'S DIMENSIONS:

d24 = (Displayed:) 1R

(Stored:) 1.0 (0.01, -0.01)

d25 = (Displayed:) 1R

(Stored:) 1.0 (0.01, -0.01)

d26 = (Displayed:) 1R

(Stored:) 1.0 (0.01, -0.01)

END ADD

ADD FEATURE

INTERNAL FEATURE ID 520

PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	2 Sets
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined
2.1.4.1	Ball/Spine	Rolling Ball
2.1.5	Extend Surfaces	Disable
2.1.6	Radii	1 Points
2.1.6.1	Rad 0	Defined

2.1.6.1.1 D1 Defined

2.1.6.1.1.1 Distance type Enter Value

2.1.6.1.1.2 Distance value 1.00

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

2.2 Set 1 Defined

2.2.1 Shape options Constant

2.2.2 Conic Defined

2.2.2.1 Conic Type Plain

2.2.3 References Defined

2.2.3.1 Reference type Edge Chain

2.2.3.2 Curve Collection 2 Selections

2.2.4 Spine Defined

2.2.4.1 Ball/Spine Rolling Ball

2.2.5 Extend Surfaces Disable

2.2.6 Radii 1 Points

2.2.6.1 Rad 0 Defined

2.2.6.1.1 D1 Defined

2.2.6.1.1.1 Distance type Enter Value

2.2.6.1.1.2 Distance value 1.00

2.2.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

3 Attach type Make Solid

4 Transitions Defined

FEATURE'S DIMENSIONS:

d27 = (Displayed:) 1R

(Stored:) 1.0 (0.01, -0.01)

d28 = (Displayed:) 1R

(Stored:) 1.0 (0.01, -0.01)

END ADD

END IF

MASSPROP

END MASSPROP

APPENDIX G

Clamping Ring Wire Program

VERSION 5.0

REVNUM 253

LISTING FOR PART CLAMPING_RING_WIRE_UP

INPUT

BJ_13CRWUP NUMBER

END INPUT

RELATIONS

CRWUP_1=BJ_13CRWUP+3

END RELATIONS

ADD FEATURE (initial number 1)

INTERNAL FEATURE ID 1

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	X Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = RIGHT

FEATURE IS IN LAYER(S) :

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 2)
INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = TOP

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------

1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)

INTERNAL FEATURE ID 7

TYPE = COORDINATE SYSTEM

NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :

05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN

05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 5)

INTERNAL FEATURE ID 39

PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

```
NO.  ELEMENT NAME      INFO
---  -------------     -------------
 1   Feature Name      Defined
 2   Extrude Feat type  Solid
 3   Material          Add
 4   Section           Defined
 4.1   Setup Plane     Defined
 4.1.1  Sketching Plane    FRONT:F3(DATUM PLANE)
 4.1.2  View Direction     Side 1
 4.1.3  Orientation        Right
 4.1.4  Reference          RIGHT:F1(DATUM PLANE)
 4.2   Sketch          Defined
 5   Feature Form      Solid
 6   Revolve Axis      Defined
 7   Revolve Axis Option  Internal Centerline
 8   Direction         Side 2
 9   Angle             Defined
 9.1   Side One        Defined
 9.1.1  Side One Angle     None
 9.2   Side Two        Defined
 9.2.1  Side Two Angle     Variable
 9.2.2  Value          360.00

SECTION NAME = S2D0003

  FEATURE IS IN LAYER(S) :
   02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d0 = (Displayed:) 360
   (   Stored:) 360.0 ( 0.5, -0.5 )
d1 = (Displayed:) 1.8 Dia
   (   Stored:) 1.8 ( 0.01, -0.01 )
crwup_1 = (Displayed:) 33 Dia
        (   Stored:) 33.0 ( 0.01, -0.01 )
END ADD
```

MASSPROP
END MASSPROP

APPENDIX H

Dust Boot Design for Ø30, Ø35 and Ø40 mm sphere diameters program

```
VERSION 5.0
REVNUM 1174
LISTING FOR PART DUST_BOOT_30_35_40

INPUT
 BJ_6DB NUMBER
 BJ_13DB NUMBER
 BJ_14DB NUMBER
END INPUT

RELATIONS
IF BJ_14DB==40
DB_1=51
DB_2=4.55
DB_3=49
DB_4=50
DB_5=40
DB_6=50
DB_7=BJ_13DB
DB_8=BJ_13DB+3
DB_9=BJ_13DB+7
DB_10=BJ_13DB+8
DB_11=56
DB_12=3.4
DB_13=4.86
DB_14=BJ_6DB-23
DB_15=1.66
DB_16=2.5
DB_17=4
DB_18=4
DB_19=3
DB_20=1.8
DB_21=3.5
ENDIF
```

```
IF BJ_14DB==35
DB_1=45.5
DB_2=3.36
DB_3=42
DB_4=43
DB_5=30
DB_6=40
DB_7=BJ_13DB
DB_8=BJ_13DB+3
DB_9=BJ_13DB+7
DB_10=BJ_13DB+8
DB_11=51
DB_12=2.75
DB_13=4.1
DB_14=BJ_6DB-21.65
DB_15=1.4
DB_16=2.5
DB_17=4
DB_18=4
DB_19=3
DB_20=1.8
DB_21=3.5
ENDIF
IF BJ_14DB==30
DB_1=40.2
DB_2=2.5
DB_3=37
DB_4=38
DB_5=27
DB_6=39
DB_7=BJ_13DB
DB_8=BJ_13DB+3
DB_9=BJ_13DB+7
DB_10=BJ_13DB+8
DB_11=44
DB_12=2.9
DB_13=4
```

```
DB_14=BJ_6DB-19
DB_15=1.2
DB_16=1.5
DB_17=3
DB_18=3
DB_19=2
DB_20=1.8
DB_21=3.5
ENDIF
END RELATIONS
 ADD FEATURE (initial number 1)
 INTERNAL FEATURE ID  1

 DATUM PLANE

  NO.     ELEMENT NAME    INFO
  ---     -------------   -------------
   1      Feature Name    Defined
   2      Constraints     Defined
   2.1    Constraint #1   Defined
   2.1.1  Constr Type     X Axis
   3      Flip Datum Dir  Defined
   4      Fit             Defined
   4.1    Fit Type        Default

 NAME = RIGHT

   FEATURE IS IN LAYER(S) :
     01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
     01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

 END ADD

 ADD FEATURE (initial number 2)
 INTERNAL FEATURE ID  3
```

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = TOP

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD
ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)
INTERNAL FEATURE ID 7
TYPE = COORDINATE SYSTEM
NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :
05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN
05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 5)
INTERNAL FEATURE ID 39
PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Revolve Axis	Defined

7 Revolve Axis Option Internal Centerline
8 Direction Side 2
9 Angle Defined
9.1 Side One Defined
9.1.1 Side One Angle None
9.2 Side Two Defined
9.2.1 Side Two Angle Variable
9.2.2 Value 360.00

SECTION NAME = S2D0002

FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d0 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
db_17 = (Displayed:) 4 Dia
(Stored:) 4.0 (0.01, -0.01)
db_18 = (Displayed:) 4 Dia
(Stored:) 4.0 (0.01, -0.01)
db_20 = (Displayed:) 1.8 Dia
(Stored:) 1.8 (0.01, -0.01)
db_21 = (Displayed:) 3.5
(Stored:) 3.5 (0.01, -0.01)
db_19 = (Displayed:) 3 Dia
(Stored:) 3.0 (0.01, -0.01)
d18 = (Displayed:) 75.5
(Stored:) 75.5 (0.5, -0.5)
db_5 = (Displayed:) 40 Dia
(Stored:) 40.0 (0.01, -0.01)
db_6 = (Displayed:) 50 Dia
(Stored:) 50.0 (0.01, -0.01)
db_7 = (Displayed:) 30 Dia
(Stored:) 30.0 (0.01, -0.01)
db_8 = (Displayed:) 33 Dia
(Stored:) 33.0 (0.01, -0.01)

db_10 = (Displayed:) 38 Dia
(Stored:) 38.0 (0.01, -0.01)
d39 = (Displayed:) 135
(Stored:) 135.0 (0.5, -0.5)
db_1 = (Displayed:) 51 Dia
(Stored:) 51.0 (0.01, -0.01)
db_2 = (Displayed:) 4.55
(Stored:) 4.55 (0.01, -0.01)
db_12 = (Displayed:) 3.4
(Stored:) 3.4 (0.01, -0.01)
db_15 = (Displayed:) 1.66
(Stored:) 1.66 (0.01, -0.01)
db_13 = (Displayed:) 4.86
(Stored:) 4.86 (0.01, -0.01)
db_16 = (Displayed:) 2.5R
(Stored:) 2.5 (0.01, -0.01)
db_4 = (Displayed:) 50 Dia
(Stored:) 50.0 (0.01, -0.01)
db_11 = (Displayed:) 56 Dia
(Stored:) 56.0 (0.01, -0.01)
db_14 = (Displayed:) 6
(Stored:) 6.0 (0.01, -0.01)
db_9 = (Displayed:) 37 Dia
(Stored:) 37.0 (0.01, -0.01)
db_3 = (Displayed:) 49 Dia
(Stored:) 49.0 (0.01, -0.01)
END ADD

IF BJ_14DB==40

ADD FEATURE (initial number 6)
INTERNAL FEATURE ID 478
PARENTS = 39(#5)

ROUND: General

NO. ELEMENT NAME INFO

---	-------------	-------------
1	Feature Name	Defined
2	Sets	2 Sets
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined
2.1.4.1	Ball/Spine	Rolling Ball
2.1.5	Extend Surfaces	Disable
2.1.6	Radii	1 Points
2.1.6.1	Rad 0	Defined
2.1.6.1.1	D1	Defined
2.1.6.1.1.1	Distance type	Enter Value
2.1.6.1.1.2	Distance value	2.00
2.1.7	Pieces	1 of 1 Included, 0 Trimmed, 0 Extended
2.2	Set 1	Defined
2.2.1	Shape options	Constant
2.2.2	Conic	Defined
2.2.2.1	Conic Type	Plain
2.2.3	References	Defined
2.2.3.1	Reference type	Edge Chain
2.2.3.2	Curve Collection	2 Selections
2.2.4	Spine	Defined
2.2.4.1	Ball/Spine	Rolling Ball
2.2.5	Extend Surfaces	Disable
2.2.6	Radii	1 Points
2.2.6.1	Rad 0	Defined
2.2.6.1.1	D1	Defined
2.2.6.1.1.1	Distance type	Enter Value
2.2.6.1.1.2	Distance value	2.00
2.2.7	Pieces	1 of 1 Included, 0 Trimmed, 0 Extended
3	Attach type	Make Solid
4	Transitions	Defined

```
   FEATURE'S DIMENSIONS:
   d57 = (Displayed:) 2R
        (   Stored:) 2.0 ( 0.01, -0.01 )
   d58 = (Displayed:) 2R
        (   Stored:) 2.0 ( 0.01, -0.01 )
   END ADD
END IF

ADD FEATURE (initial number 7)
INTERNAL FEATURE ID  526
PARENTS = 39(#5)

DATUM PLANE

 NO.    ELEMENT NAME       INFO
 ---    -------------      -------------
  1     Feature Name       Defined
  2     Constraints        Defined
  2.1    Constraint #1      Defined
  2.1.1  Constr Type        Through
  2.1.2  Constr References  edge of feat #5 (PROTRUSION)
  3     Flip Datum Dir     Defined
  4     Fit                Defined
  4.1    Fit Type           Default

NAME = DTM1

  FEATURE IS IN LAYER(S) :
    01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

END ADD
ADD FEATURE (initial number 8)
INTERNAL FEATURE ID  528
PARENTS = 526(#7)

DATUM PLANE
```

NO. ELEMENT NAME INFO
--- ------------- -------------
1 Feature Name Defined
2 Constraints Defined
2.1 Constraint #1 Defined
2.1.1 Constr Type Offset
2.1.2 Constr References Surface DTM1 of feat #7 (DATUM PLANE)
2.1.3 Constr Ref Offset Value = -3.1300
3 Flip Datum Dir Defined
4 Fit Defined
4.1 Fit Type Default

NAME = DTM2

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d60 = (Displayed:) 3.13
(Stored:) -3.13 (0.01, -0.01)
END ADD

IF BJ_14DB==35

ADD FEATURE
INTERNAL FEATURE ID 537
PARENTS = 39(#5)

ROUND: General

NO. ELEMENT NAME INFO
--- ------------- -------------
1 Feature Name Defined
2 Sets 2 Sets
2.1 Set 0 Defined
2.1.1 Shape options Constant

2.1.2 Conic Defined

2.1.2.1 Conic Type Plain

2.1.3 References Defined

2.1.3.1 Reference type Edge Chain

2.1.3.2 Curve Collection 2 Selections

2.1.4 Spine Defined

2.1.4.1 Ball/Spine Rolling Ball

2.1.5 Extend Surfaces Disable

2.1.6 Radii 1 Points

2.1.6.1 Rad 0 Defined

2.1.6.1.1 D1 Defined

2.1.6.1.1.1 Distance type Enter Value

2.1.6.1.1.2 Distance value 1.50

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

2.2 Set 1 Defined

2.2.1 Shape options Constant

2.2.2 Conic Defined

2.2.2.1 Conic Type Plain

2.2.3 References Defined

2.2.3.1 Reference type Edge Chain

2.2.3.2 Curve Collection 2 Selections

2.2.4 Spine Defined

2.2.4.1 Ball/Spine Rolling Ball

2.2.5 Extend Surfaces Disable

2.2.6 Radii 1 Points

2.2.6.1 Rad 0 Defined

2.2.6.1.1 D1 Defined

2.2.6.1.1.1 Distance type Enter Value

2.2.6.1.1.2 Distance value 1.50

2.2.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

3 Attach type Make Solid

4 Transitions Defined

FEATURE'S DIMENSIONS:

d61 = (Displayed:) 1.5R

(Stored:) 1.5 (0.01, -0.01)

d62 = (Displayed:) 1.5R

(Stored:) 1.5 (0.01, -0.01)
END ADD
END IF

IF BJ_14DB==30

ADD FEATURE
INTERNAL FEATURE ID 580
PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	2 Sets
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined
2.1.4.1	Ball/Spine	Rolling Ball
2.1.5	Extend Surfaces	Disable
2.1.6	Radii	1 Points
2.1.6.1	Rad 0	Defined
2.1.6.1.1	D1	Defined
2.1.6.1.1.1	Distance type	Enter Value
2.1.6.1.1.2	Distance value	1.00
2.1.7	Pieces	1 of 1 Included, 0 Trimmed, 0 Extended
2.2	Set 1	Defined
2.2.1	Shape options	Constant
2.2.2	Conic	Defined
2.2.2.1	Conic Type	Plain
2.2.3	References	Defined

2.2.3.1 Reference type Edge Chain
2.2.3.2 Curve Collection 2 Selections
2.2.4 Spine Defined
2.2.4.1 Ball/Spine Rolling Ball
2.2.5 Extend Surfaces Disable
2.2.6 Radii 1 Points
2.2.6.1 Rad 0 Defined
2.2.6.1.1 D1 Defined
2.2.6.1.1.1 Distance type Enter Value
2.2.6.1.1.2 Distance value 1.00
2.2.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d63 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
d64 = (Displayed:) 1R
(Stored:) 1.0 (0.01, -0.01)
END ADD
END IF

MASSPROP
END MASSPROP

APPENDIX I

Dust Boot Design for Ø27 mm sphere diameter program

```
VERSION 5.0
REVNUM 854
LISTING FOR PART DUST_BOOT_27

INPUT
 BJ_6DB NUMBER
 BJ_13DB NUMBER
 BJ_14DB NUMBER
END INPUT

RELATIONS
IF BJ_14DB==27
DB_4=BJ_13DB+3
DB_5=BJ_13DB
DB_6=BJ_6DB-6
ENDIF
END RELATIONS
 ADD FEATURE (initial number 1)
 INTERNAL FEATURE ID  1
 DATUM PLANE

  NO.    ELEMENT NAME    INFO
  ---    -------------   -------------
   1     Feature Name    Defined
   2     Constraints     Defined
   2.1    Constraint #1   Defined
   2.1.1  Constr Type     X Axis
   3     Flip Datum Dir  Defined
   4     Fit             Defined
   4.1    Fit Type        Default

 NAME = RIGHT

   FEATURE IS IN LAYER(S) :
```

01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD
ADD FEATURE (initial number 2)
INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = TOP
FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5
DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined

4 Fit Defined
4.1 Fit Type Default

NAME = FRONT

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
01___PRT_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)
INTERNAL FEATURE ID 7
TYPE = COORDINATE SYSTEM
NAME = PRT_CSYS_DEF

FEATURE IS IN LAYER(S) :
05___PRT_ALL_DTM_CSYS - OPERATION = SHOWN
05___PRT_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 5)
INTERNAL FEATURE ID 39
PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)

4.1.2 View Direction Side 1
4.1.3 Orientation Right
4.1.4 Reference RIGHT:F1(DATUM PLANE)
4.2 Sketch Defined
5 Feature Form Solid
6 Revolve Axis Defined
7 Revolve Axis Option Internal Centerline
8 Direction Side 2
9 Angle Defined
9.1 Side One Defined
9.1.1 Side One Angle None
9.2 Side Two Defined
9.2.1 Side Two Angle Variable
9.2.2 Value 360.00

SECTION NAME = S2D0015
FEATURE IS IN LAYER(S) :
02___PRT_ALL_AXES - OPERATION = SHOWN
FEATURE'S DIMENSIONS:
d0 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
db_15 = (Displayed:) 42 Dia
(Stored:) 42.0 (0.01, -0.01)
db_13 = (Displayed:) 35 Dia
(Stored:) 35.0 (0.01, -0.01)
db_9 = (Displayed:) 15
(Stored:) 15.0 (0.5, -0.5)
db_12 = (Displayed:) 3.2
(Stored:) 3.2 (0.01, -0.01)
db_14 = (Displayed:) 38.5 Dia
(Stored:) 38.5 (0.01, -0.01)
db_16 = (Displayed:) 1.5
(Stored:) 1.5 (0.01, -0.01)
db_3 = (Displayed:) 27.5 Dia
(Stored:) 27.5 (0.01, -0.01)
d8 = (Displayed:) 2
(Stored:) 2.0 (0.01, -0.01)
db_18 = (Displayed:) 1.85 Dia

(Stored:) 1.85 (0.01, -0.01)
db_4 = (Displayed:) 23 Dia
(Stored:) 23.0 (0.01, -0.01)
d11 = (Displayed:) 75
(Stored:) 75.0 (0.5, -0.5)
db_20 = (Displayed:) 25.5 Dia
(Stored:) 25.5 (0.01, -0.01)
db_1 = (Displayed:) 43.5 Dia
(Stored:) 43.5 (0.01, -0.01)
db_8 = (Displayed:) 9 Dia
(Stored:) 9.0 (0.01, -0.01)
db_11 = (Displayed:) 6.9
(Stored:) 6.9 (0.01, -0.01)
db_10 = (Displayed:) 7.2
(Stored:) 7.2 (0.01, -0.01)
db_2 = (Displayed:) 40 Dia
(Stored:) 40.0 (0.01, -0.01)
db_7 = (Displayed:) 6R
(Stored:) 6.0 (0.01, -0.01)
db_17 = (Displayed:) 1.8
(Stored:) 1.8 (0.01, -0.01)
db_19 = (Displayed:) 3.5
(Stored:) 3.5 (0.01, -0.01)
db_5 = (Displayed:) 20 Dia
(Stored:) 20.0 (0.01, -0.01)
db_6 = (Displayed:) 19
(Stored:) 19.0 (0.01, -0.01)
END ADD
ADD FEATURE (initial number 6)
INTERNAL FEATURE ID 362
PARENTS = 39(#5)
ROUND: General

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Sets	1 Set

2.1 Set 0 Defined
2.1.1 Shape options Constant
2.1.2 Conic Defined
2.1.2.1 Conic Type Plain
2.1.3 References Defined
2.1.3.1 Reference type Edge Chain
2.1.3.2 Curve Collection 2 Selections
2.1.4 Spine Defined
2.1.4.1 Ball/Spine Rolling Ball
2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined
2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 0.50
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d33 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 7)
INTERNAL FEATURE ID 382
PARENTS = 39(#5)
ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	1 Set
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined

2.1.3.1 Reference type Edge Chain
2.1.3.2 Curve Collection 2 Selections
2.1.4 Spine Defined
2.1.4.1 Ball/Spine Rolling Ball
2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined
2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 0.50
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d34 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 8)
INTERNAL FEATURE ID 402
PARENTS = 39(#5)

ROUND: General

NO. ELEMENT NAME INFO
--- ------------- -------------
1 Feature Name Defined
2 Sets 1 Set
2.1 Set 0 Defined
2.1.1 Shape options Constant
2.1.2 Conic Defined
2.1.2.1 Conic Type Plain
2.1.3 References Defined
2.1.3.1 Reference type Edge Chain
2.1.3.2 Curve Collection 2 Selections
2.1.4 Spine Defined

2.1.4.1 Ball/Spine Rolling Ball
2.1.5 Extend Surfaces Disable
2.1.6 Radii 1 Points
2.1.6.1 Rad 0 Defined
2.1.6.1.1 D1 Defined
2.1.6.1.1.1 Distance type Enter Value
2.1.6.1.1.2 Distance value 0.50
2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined

FEATURE'S DIMENSIONS:
d35 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 9)
INTERNAL FEATURE ID 424
PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	1 Set
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined
2.1.4.1	Ball/Spine	Rolling Ball
2.1.5	Extend Surfaces	Disable
2.1.6	Radii	1 Points

2.1.6.1 Rad 0 Defined

2.1.6.1.1 D1 Defined

2.1.6.1.1.1 Distance type Enter Value

2.1.6.1.1.2 Distance value 0.50

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended

3 Attach type Make Solid

4 Transitions Defined

FEATURE'S DIMENSIONS:

d36 = (Displayed:) 0.5R

(Stored:) 0.5 (0.01, -0.01)

END ADD

ADD FEATURE (initial number 10)

INTERNAL FEATURE ID 446

PARENTS = 39(#5)

ROUND: General

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Sets	1 Set
2.1	Set 0	Defined
2.1.1	Shape options	Constant
2.1.2	Conic	Defined
2.1.2.1	Conic Type	Plain
2.1.3	References	Defined
2.1.3.1	Reference type	Edge Chain
2.1.3.2	Curve Collection	2 Selections
2.1.4	Spine	Defined
2.1.4.1	Ball/Spine	Rolling Ball
2.1.5	Extend Surfaces	Disable
2.1.6	Radii	1 Points
2.1.6.1	Rad 0	Defined
2.1.6.1.1	D1	Defined
2.1.6.1.1.1	Distance type	Enter Value
2.1.6.1.1.2	Distance value	0.50

2.1.7 Pieces 1 of 1 Included, 0 Trimmed, 0 Extended
3 Attach type Make Solid
4 Transitions Defined
FEATURE'S DIMENSIONS:
d37 = (Displayed:) 0.5R
(Stored:) 0.5 (0.01, -0.01)
END ADD

ADD FEATURE (initial number 11)
INTERNAL FEATURE ID 484
PARENTS = 1(#1) 39(#5) 5(#3)
CUT: Revolve

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Right
4.1.4	Reference	RIGHT:F1(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Revolve Axis	Defined
8	Revolve Axis Option	Internal Centerline
9	Direction	Side 2
10	Angle	Defined
10.1	Side One	Defined
10.1.1	Side One Angle	None
10.2	Side Two	Defined
10.2.1	Side Two Angle	Variable
10.2.2	Value	360.00

SECTION NAME = S2D0004

FEATURE'S DIMENSIONS:
d38 = (Displayed:) 360
(Stored:) 360.0 (0.5, -0.5)
d39 = (Displayed:) 22.55 Dia (weak)
(Stored:) 22.54500475606 (0.01, -0.01)
d40 = (Displayed:) 28.3 Dia (weak)
(Stored:) 28.30418942452 (0.01, -0.01)
d41 = (Displayed:) 38.85 Dia (weak)
(Stored:) 38.84869907045 (0.01, -0.01)
d42 = (Displayed:) 39.79 Dia (weak)
(Stored:) 39.7879662983 (0.01, -0.01)
d43 = (Displayed:) 1.51R (weak)
(Stored:) 1.506311833181 (0.01, -0.01)
END ADD
ADD FEATURE (initial number 12)
INTERNAL FEATURE ID 549
PARENTS = 39(#5)
DATUM PLANE

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Through
2.1.2	Constr References	edge of feat #5 (PROTRUSION)
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = DTM1

FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN
END ADD

ADD FEATURE (initial number 13)

INTERNAL FEATURE ID 551
PARENTS = 549(#12)

DATUM PLANE

NO. ELEMENT NAME INFO
--- ------------- -------------
1 Feature Name Defined
2 Constraints Defined
2.1 Constraint #1 Defined
2.1.1 Constr Type Offset
2.1.2 Constr References Surface DTM1 of feat #12 (DATUM PLANE)
2.1.3 Constr Ref Offset Value = -2.9000
3 Flip Datum Dir Defined
4 Fit Defined
4.1 Fit Type Default
NAME = DTM2
FEATURE IS IN LAYER(S) :
01___PRT_ALL_DTM_PLN - OPERATION = SHOWN

FEATURE'S DIMENSIONS:
d44 = (Displayed:) 2.9
(Stored:) -2.9 (0.01, -0.01)
END ADD

MASSPROP
END MASSPROP

APPENDIX J

Assembly of ball joint program

```
VERSION 5.0
REVNUM 5424
LISTING FOR ASSEMBLY BALL_JOINT

INPUT
 BEARING_TYPE STRING
 CONNECTING_TYPE STRING
 HOUSING_TYPE STRING
 HOUSING_SHAFT_TYPE STRING
 DUST_BOOT_WORKING_TEMPERATURE NUMBER
 NECK_LENGTH NUMBER
 TAPER_LENGTH NUMBER
 BP_THREAD_LENGHT NUMBER
 BP_THREAD NUMBER
 PIN_DIAMETER NUMBER
 PIN_LENGTH NUMBER
 CONIC_RATIO NUMBER
 CONIC_DIAMETER NUMBER
 SPHERE_DIAMETER NUMBER
 HOUSING_SHAFT_LENGTH NUMBER
 HOUSING_THREAD_LENGTH NUMBER
 HOUSING_HOLE_LENGTH NUMBER
 HOUSING_HOLE_DIAMETER NUMBER
 HOUSING_SHAFT_DIAMETER NUMBER
 ALLEN_KEY_NUMBER NUMBER
 ANGLE_FOR_OVAL_HOUSING NUMBER
 ANGLE_FOR_ELLIPTIC_HOUSING NUMBER
 BP_MATERIAL STRING
END INPUT

RELATIONS
IF SPHERE_DIAMETER==27
    IF CONIC_DIAMETER>23
```

```
        NECK_LENGTH=25
        CONIC_DIAMETER=20
            TAPER_LENGTH=19
        BP_THREAD=16
    ENDIF
 ENDIF
 IF SPHERE_DIAMETER==40
    IF CONIC_DIAMETER<25
        NECK_LENGTH=29
        CONIC_DIAMETER=30
        BP_THREAD=24
    ENDIF
 ENDIF
IF SPHERE_DIAMETER==30
    IF CONIC_DIAMETER>25
        NECK_LENGTH=25
        CONIC_DIAMETER=20
        BP_THREAD=16
    ENDIF
 ENDIF
 IF SPHERE_DIAMETER==35
    IF CONIC_DIAMETER<26
        NECK_LENGTH=29
        CONIC_DIAMETER=30
        BP_THREAD=24
    ENDIF
 ENDIF

IF BEARING_TYPE=="METAL"
MSG_1="BEARING TYPE IS METAL"
ENDIF

IF BEARING_TYPE=="PLASTIC"
MSG_1="BEARING TYPE IS PLASTIC"
ENDIF

IF CONNECTING_TYPE=="SELF_LOCKING"
```

```
MSG_2="CONNECTIN TYPE IS SELF LOCKING (RPOVIDED WITH SELF LOCKING NUT)"
ENDIF

IF CONNECTING_TYPE=="CASTLE_NUT"
MSG_2="COONECTION TYPE IS CASTLE NUT (PROVIDED WITH CASTLE NUT)"
ENDIF

IF DUST_BOOT_WORKING_TEMPERATURE<(-30)
MSG_5="DUST BOOT METARIAL MUST BE POLYCHLROPEN"
ENDIF

IF DUST_BOOT_WORKING_TEMPERATURE>50
MSG_5="DUST BOOT METARIAL MUST BE POLYCHLROPEN"
ENDIF

IF (-30)<=DUST_BOOT_WORKING_TEMPERATURE
IF DUST_BOOT_WORKING_TEMPERATURE<=50
MSG_5="DUST BOOT METARIAL MUST BE POLYURETHANE"
ENDIF
ENDIF

IF HOUSING_TYPE=="ELLIPTICAL"
MSG_3="HOUSING TYPE IS ELLIPTICAL"
ENDIF

IF HOUSING_TYPE=="OVAL"
MSG_3="HOUSING TYPE IS OVAL"
ENDIF

IF HOUSING_SHAFT_TYPE=="INNER_THREAD"
MSG_4="HOUSING SHAFT TYPE IS INNER THREAD"
ENDIF

IF HOUSING_SHAFT_TYPE=="OUTER_THREAD"
MSG_4="HOUSING SHAFT TYPE IS OUTER THREAD"
ENDIF
```

```
IF SPHERE_DIAMETER==27
MSG_7="SPHERE DIAMETER=27"
ENDIF

IF SPHERE_DIAMETER==30
MSG_7="SPHERE DIAMETER=30"
ENDIF

IF SPHERE_DIAMETER==35
MSG_7="SPHERE DIAMETER=35"
ENDIF

IF SPHERE_DIAMETER==40
MSG_7="SPHERE DIAMETER=40"
ENDIF

IF BP_MATERIAL=="41CR4"
SUT=900
SUT2=130.4348
ENDIF

IF BP_MATERIAL=="42CRMO4"
SUT=1100
SUT2=159.42
ENDIF

IF BEARING_TYPE=="PLASTIC"
IF SPHERE_DIAMETER==27
F1=17000
F2=30000
T=10000
ND=16
ENDIF
IF SPHERE_DIAMETER==30
F1=30000
F2=35000
T=10000
```

```
ND=18
ENDIF
IF SPHERE_DIAMETER==35
F1=40000
F2=40000
T=15000
ND=21
ENDIF
IF SPHERE_DIAMETER==40
F1=40000
F2=45000
T=18000
ND=24
ENDIF
ENDIF

IF BEARING_TYPE=="METAL"
IF SPHERE_DIAMETER==27
F1=32000
F2=60000
T=10000
ND=16
ENDIF
IF SPHERE_DIAMETER==30
F1=40000
F2=80000
T=12000
ND=18
ENDIF
IF SPHERE_DIAMETER==35
F1=65000
F2=90000
T=15000
ND=21
ENDIF
IF SPHERE_DIAMETER==40
F1=90000
```

```
F2=120000
T=18000
ND=24
ENDIF
ENDIF

NL=NECK_LENGTH
TL=TAPER_LENGTH

NORMAL_STRESS=(F2/((3.14*(ND^2))/4))+(F1*(NL+(TL/2))*(ND/2))/(3.14*(ND^4)/64)

SHEAR_STRESS=(F1/((3.14*(ND^2))/4))+(T*(ND/2))/(3.14*(ND^4)/32)

VON_MISSES_STRESS=(NORMAL_STRESS+(3*SHEAR_STRESS))^0.5

ROOT_A_INC=0.245799-(0.00307794*SUT2)+(0.0000150874*SUT2^2)\
-(0.0000000266978*SUT2^3)

ROOT_A_MM=((ROOT_A_INC^2)*25.4)^0.5

KF=1+(1/(0.5+(ROOT_A_MM/(2^0.5))))

SE=SUT*(4.51*(SUT^(-0.265)))*(1.24*(ND^(-0.1)))*0.897

SAFETY_FACTOR=SE/(VON_MISSES_STRESS*KF)

IF SAFETY_FACTOR>0
MSG_6="UNSAFE DESIGN !!!!"
ENDIF

IF SAFETY_FACTOR>1
MSG_6="SAFETY FACTOR IS SMALLER THAN 1.5!!! DESIGN MUST BE IMPROVED"
ENDIF

IF SAFETY_FACTOR>1.5
MSG_6="SAFETY FACTOR IS SMALLER THAN 2!!! DESIGN MUST BE IMPROVED"
ENDIF
```

```
IF SAFETY_FACTOR>2
MSG_6="SAFETY FACTOR IS HIGHER THAN 2. DESIGN IS VALID"
ENDIF

IF SAFETY_FACTOR>3
MSG_6="SAFETY FACTOR IS HIGHER THAN 3. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>4
MSG_6="SAFETY FACTOR IS HIGHER THAN 4. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>5
MSG_6="SAFETY FACTOR IS HIGHER THAN 5. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>6
MSG_6="SAFETY FACTOR IS HIGHER THAN 6. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>7
MSG_6="SAFETY FACTOR IS HIGHER THAN 7. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>8
MSG_6="SAFETY FACTOR IS HIGHER THAN 8. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>9
MSG_6="SAFETY FACTOR IS HIGHER THAN 9. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>10
MSG_6="SAFETY FACTOR IS HIGHER THAN 10. GOOD DESIGN"
ENDIF
```

```
IF SAFETY_FACTOR>11
MSG_6="SAFETY FACTOR IS HIGHER THAN 11. GOOD DESIGN"
ENDIF

IF SAFETY_FACTOR>12
MSG_6="SAFETY FACTOR IS HIGHER THAN 12. GOOD DESIGN"
ENDIF
END RELATIONS

 EXECUTE PART BALL_PIN
 BJ_1 = BEARING_TYPE
 BJ_2 = CONNECTING_TYPE
 BJ_3 = HOUSING_TYPE
 BJ_4 = HOUSING_SHAFT_TYPE
 BJ_5 = DUST_BOOT_WORKING_TEMPERATURE
 BJ_6 = NECK_LENGTH
 BJ_7 = TAPER_LENGTH
 BJ_8 = BP_THREAD_LENGHT
 BJ_9 = BP_THREAD
 BJ_10 = PIN_DIAMETER
 BJ_11 = PIN_LENGTH
 BJ_12 = CONIC_RATIO
 BJ_13 = CONIC_DIAMETER
 BJ_14 = SPHERE_DIAMETER
 BJ_15 = HOUSING_SHAFT_LENGTH
 BJ_16 = HOUSING_THREAD_LENGTH
 BJ_17 = HOUSING_HOLE_LENGTH
 BJ_18 = HOUSING_HOLE_DIAMETER
 BJ_19 = HOUSING_SHAFT_DIAMETER
 BJ_20 = ALLEN_KEY_NUMBER
 BJ_21 = ANGLE_FOR_OVAL_HOUSING
 BJ_22 = ANGLE_FOR_ELLIPTIC_HOUSING
 BJ_23 = BP_MATERIAL
 END EXECUTE

 EXECUTE PART HOUSING
 BJ_1H = BEARING_TYPE
```

```
 BJ_2H = CONNECTING_TYPE
 BJ_3H = HOUSING_TYPE
 BJ_4H = HOUSING_SHAFT_TYPE
 BJ_5H = DUST_BOOT_WORKING_TEMPERATURE
!*** WARNG: invalid input variable in assignment
 BJ_6H = NECK_LENGTH
!*** WARNG: invalid input variable in assignment
 BJ_7H = TAPER_LENGTH
!*** WARNG: invalid input variable in assignment
 BJ_8H = BP_THREAD_LENGHT
!*** WARNG: invalid input variable in assignment
 BJ_9H = BP_THREAD
!*** WARNG: invalid input variable in assignment
 BJ_10H = PIN_DIAMETER
!*** WARNG: invalid input variable in assignment
 BJ_11H = PIN_LENGTH
!*** WARNG: invalid input variable in assignment
 BJ_12H = CONIC_RATIO
!*** WARNG: invalid input variable in assignment
 BJ_13H = CONIC_DIAMETER
!*** WARNG: invalid input variable in assignment
 BJ_14H = SPHERE_DIAMETER
 BJ_15H = HOUSING_SHAFT_LENGTH
 BJ_16H = HOUSING_THREAD_LENGTH
 BJ_17H = HOUSING_HOLE_LENGTH
 BJ_18H = HOUSING_HOLE_DIAMETER
 BJ_19H = HOUSING_SHAFT_DIAMETER
 BJ_20H = ALLEN_KEY_NUMBER
!*** WARNG: invalid input variable in assignment
 BJ_21H = ANGLE_FOR_OVAL_HOUSING
!*** WARNG: invalid input variable in assignment
 BJ_22H = ANGLE_FOR_ELLIPTIC_HOUSING
!*** WARNG: invalid input variable in assignment
 BJ_23H = BP_MATERIAL
!*** WARNG: invalid input variable in assignment
 END EXECUTE
```

```
 EXECUTE PART END_CAP_MB_27_30_35
 BJ_14ECMB = SPHERE_DIAMETER
 END EXECUTE
!*** WARNG: cannot find this model

 EXECUTE PART SPRING
 BJ_14B = SPHERE_DIAMETER
 END EXECUTE

 EXECUTE PART METAL_BEARING_27_30_35
 BJ_14MB = SPHERE_DIAMETER
 END EXECUTE
!*** WARNG: cannot find this model

 EXECUTE PART END_CAP_PB
 BJ_14ECPB = SPHERE_DIAMETER
 END EXECUTE
!*** WARNG: cannot find this model

 EXECUTE PART CLAMPING_RING_WIRE_UP
 BJ_13CRWUP = CONIC_DIAMETER
 END EXECUTE

 EXECUTE PART CLAMPING_RING_SHEET
 BJ_14CRS = SPHERE_DIAMETER
 END EXECUTE

 EXECUTE PART DUST_BOOT_30_35_40
 BJ_6DB = NECK_LENGTH
 BJ_13DB = CONIC_DIAMETER
 BJ_14DB = SPHERE_DIAMETER
 END EXECUTE

 EXECUTE PART DUST_BOOT_27
 BJ_6DB = NECK_LENGTH
 BJ_13DB = CONIC_DIAMETER
 BJ_14DB = SPHERE_DIAMETER
```

END EXECUTE
!*** WARNG: cannot find this model

ADD FEATURE (initial number 1)
INTERNAL FEATURE ID 1

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	X Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = ASM_RIGHT

FEATURE IS IN LAYER(S) :
01__ASM_ALL_DTM_PLN - OPERATION = SHOWN
01__ASM_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 2)
INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis

3 Flip Datum Dir Defined
4 Fit Defined
4.1 Fit Type Default

NAME = ASM_TOP

FEATURE IS IN LAYER(S) :
01__ASM_ALL_DTM_PLN - OPERATION = SHOWN
01__ASM_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 3)
INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = ASM_FRONT

FEATURE IS IN LAYER(S) :
01__ASM_ALL_DTM_PLN - OPERATION = SHOWN
01__ASM_DEF_DTM_PLN - OPERATION = SHOWN

END ADD

ADD FEATURE (initial number 4)

```
INTERNAL FEATURE ID  7
TYPE = COORDINATE SYSTEM
NAME = ASM_DEF_CSYS


  FEATURE IS IN LAYER(S) :
    05__ASM_ALL_DTM_CSYS - OPERATION = SHOWN
    05__ASM_DEF_DTM_CSYS - OPERATION = SHOWN

END ADD

ADD PART HOUSING
INTERNAL COMPONENT ID 39
END ADD

IF BEARING_TYPE=="METAL"
   ADD PART BALL_PIN
   INTERNAL COMPONENT ID 413
   PARENTS = 39(#5)
   END ADD
END IF

IF BEARING_TYPE=="METAL"
   IF SPHERE_DIAMETER==40
      ADD PART METAL_BEARING
      INTERNAL COMPONENT ID 471
      PARENTS = 413(#6)
      END ADD

      ADD PART SPRING
      INTERNAL COMPONENT ID 586
      PARENTS = 471(#7) 39(#5)
      END ADD

      ADD PART END_CAP_MB
      INTERNAL COMPONENT ID 611
      PARENTS = 413(#6)
```

```
    END ADD
  END IF

  IF SPHERE_DIAMETER!=40
    ADD PART METAL_BEARING_27_30_35
    INTERNAL COMPONENT ID 874
    PARENTS = 413(#6) 39(#5)
    END ADD

    ADD PART SPRING
    INTERNAL COMPONENT ID 1224
    PARENTS = 874(*) 39(#5)
    END ADD

    ADD PART END_CAP_MB_27_30_35
    INTERNAL COMPONENT ID 950
    PARENTS = 413(#6) 1224(*)
    END ADD
  END IF
END IF

IF BEARING_TYPE=="PLASTIC"
  IF SPHERE_DIAMETER==27
    ADD PART PLASTIC_BEARING_27
    INTERNAL COMPONENT ID 1537
    PARENTS = 39(#5)
    END ADD

    ADD PART END_CAP_PB
    INTERNAL COMPONENT ID 1643
    PARENTS = 1537(*) 39(#5)
    END ADD
  END IF
END IF

IF BEARING_TYPE=="PLASTIC"
  IF SPHERE_DIAMETER==30
```

```
      ADD PART PLASTIC_BEARING_30
      INTERNAL COMPONENT ID 1724
      PARENTS = 39(#5)
      END ADD

      ADD PART END_CAP_PB
      INTERNAL COMPONENT ID 1788
      PARENTS = 1724(*) 39(#5)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==35
   IF BEARING_TYPE=="PLASTIC"
      ADD PART PLASTIC_BEARING_35
      INTERNAL COMPONENT ID 1859
      PARENTS = 39(#5)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==27
   IF BEARING_TYPE=="PLASTIC"
      ADD PART BALL_PIN
      INTERNAL COMPONENT ID 1899
      PARENTS = 39(#5)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==30
   IF BEARING_TYPE=="PLASTIC"
      ADD PART BALL_PIN
      INTERNAL COMPONENT ID 1901
      PARENTS = 39(#5)
      END ADD
   END IF
```

```
END IF

IF SPHERE_DIAMETER==35
   IF BEARING_TYPE=="PLASTIC"
      ADD PART BALL_PIN
      INTERNAL COMPONENT ID 1902
      PARENTS = 39(#5)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==40
   IF BEARING_TYPE=="PLASTIC"
      ADD PART BALL_PIN
      INTERNAL COMPONENT ID 1909
      PARENTS = 39(#5)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==35
   IF BEARING_TYPE=="PLASTIC"
      ADD PART END_CAP_PB
      INTERNAL COMPONENT ID 2002
      PARENTS = 1859(*) 39(#5)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==40
   IF BEARING_TYPE=="PLASTIC"
      ADD PART PLASTIC_BEARING_40
      INTERNAL COMPONENT ID 2203
      PARENTS = 1909(*) 39(#5)
      END ADD
   END IF
END IF
```

```
IF SPHERE_DIAMETER==40
   IF BEARING_TYPE=="PLASTIC"
      ADD PART END_CAP_PB
      INTERNAL COMPONENT ID 2264
      PARENTS = 2203(*) 39(#5)
      END ADD
   END IF
END IF

IF BEARING_TYPE=="METAL"
   IF SPHERE_DIAMETER==40
      ADD PART CLAMPING_RING_SHEET
      INTERNAL COMPONENT ID 2323
      PARENTS = 39(#5) 413(#6)
      END ADD

      ADD PART DUST_BOOT_30_35_40
      INTERNAL COMPONENT ID 2942
      PARENTS = 2323(#10) 413(#6)
      END ADD

      ADD PART CLAMPING_RING_WIRE_UP
      INTERNAL COMPONENT ID 2998
      PARENTS = 2942(#11) 413(#6)
      END ADD
   END IF
END IF

IF BEARING_TYPE=="PLASTIC"
   IF SPHERE_DIAMETER==40
      ADD PART CLAMPING_RING_SHEET
      INTERNAL COMPONENT ID 3061
      PARENTS = 39(#5) 1909(*)
      END ADD

      ADD PART DUST_BOOT_30_35_40
```

```
      INTERNAL COMPONENT ID 3062
      PARENTS = 3061(*) 1909(*)
      END ADD

      ADD PART CLAMPING_RING_WIRE_UP
      INTERNAL COMPONENT ID 3063
      PARENTS = 3062(*) 1909(*)
      END ADD
   END IF
END IF

IF BEARING_TYPE=="METAL"
   IF SPHERE_DIAMETER==35
      ADD PART CLAMPING_RING_SHEET
      INTERNAL COMPONENT ID 3066
      PARENTS = 39(#5) 413(#6)
      END ADD

      ADD PART DUST_BOOT_30_35_40
      INTERNAL COMPONENT ID 3127
      PARENTS = 3066(*) 413(#6)
      END ADD

      ADD PART CLAMPING_RING_WIRE_UP
      INTERNAL COMPONENT ID 3230
      PARENTS = 3127(*) 413(#6)
      END ADD
   END IF
END IF

IF BEARING_TYPE=="PLASTIC"
   IF SPHERE_DIAMETER==35
      ADD PART CLAMPING_RING_SHEET
      INTERNAL COMPONENT ID 3243
      PARENTS = 39(#5) 1902(*)
      END ADD
```

```
    ADD PART DUST_BOOT_30_35_40
    INTERNAL COMPONENT ID 3245
    PARENTS = 3243(*) 1902(*)
    END ADD

    ADD PART CLAMPING_RING_WIRE_UP
    INTERNAL COMPONENT ID 3246
    PARENTS = 3245(*) 1902(*)
    END ADD
  END IF
END IF

IF BEARING_TYPE=="METAL"
  IF SPHERE_DIAMETER==30
    ADD PART CLAMPING_RING_SHEET
    INTERNAL COMPONENT ID 3364
    PARENTS = 39(#5) 413(#6)
    END ADD

    ADD PART DUST_BOOT_30_35_40
    INTERNAL COMPONENT ID 3395
    PARENTS = 3364(*) 413(#6)
    END ADD

    ADD PART CLAMPING_RING_WIRE_UP
    INTERNAL COMPONENT ID 3526
    PARENTS = 3395(*) 413(#6)
    END ADD
  END IF
END IF

IF BEARING_TYPE=="PLASTIC"
  IF SPHERE_DIAMETER==30
    ADD PART CLAMPING_RING_SHEET
    INTERNAL COMPONENT ID 3533
    PARENTS = 39(#5) 1901(*)
    END ADD
```

```
      ADD PART DUST_BOOT_30_35_40
      INTERNAL COMPONENT ID 3535
      PARENTS = 3533(*) 1901(*)
      END ADD

      ADD PART CLAMPING_RING_WIRE_UP
      INTERNAL COMPONENT ID 3536
      PARENTS = 3535(*) 1901(*)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==27
   IF BEARING_TYPE=="METAL"
      ADD PART DUST_BOOT_27
      INTERNAL COMPONENT ID 3725
      END ADD

      ADD PART CLAMPING_RING_WIRE_UP
      INTERNAL COMPONENT ID 3919
      PARENTS = 3725(*) 413(#6)
      END ADD

      ADD PART CLAMPING_RING_WIRE_B
      INTERNAL COMPONENT ID 3932
      PARENTS = 413(#6)
      END ADD
   END IF
END IF

IF SPHERE_DIAMETER==27
   IF BEARING_TYPE=="PLASTIC"
      ADD PART DUST_BOOT_27
      INTERNAL COMPONENT ID 3993
      PARENTS = 1899(*)
      END ADD
```

```
      ADD PART CLAMPING_RING_WIRE_B
      INTERNAL COMPONENT ID 3996
      PARENTS = 1899(*)
      END ADD

      ADD PART CLAMPING_RING_WIRE_UP
      INTERNAL COMPONENT ID 4000
      PARENTS = 3993(*) 1899(*)
      END ADD
   END IF
END IF
ADD FEATURE (initial number 13)
INTERNAL FEATURE ID  4585

ANNOTATION FEATURE

 NO.     ELEMENT NAME              INFO
 ---     -------------             -------------
  1      Feature Name              Defined
  2      Annotation Elements Array  Defined
  2.1      Annotation Element #1      Defined
  2.1.1    Annotation Element Name    AE_NOTE0
  2.1.2    Annotation                 Defined
  2.1.2.1  Annotation Type            Note
  2.1.2.2  Annotation Id              3
  2.1.2.3  Consumed                   Defined
  2.1.2.4  Annotation References      Defined
  2.1.3    User Defined References    Defined
  2.1.4    Application                Defined
  2.1.4.1  Application Type           Defined
  2.1.4.2  Application Data           Defined
  2.1.4.3  Consumed                   Defined
  2.1.4.4  Application References     Defined
  2.1.5    Copy Flag                  Defined

END ADD
```

ADD FEATURE (initial number 14)
INTERNAL FEATURE ID 4587

ANNOTATION FEATURE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Annotation Elements Array	Defined
2.1	Annotation Element #1	Defined
2.1.1	Annotation Element Name	AE_NOTE1
2.1.2	Annotation	Defined
2.1.2.1	Annotation Type	Note
2.1.2.2	Annotation Id	4
2.1.2.3	Consumed	Defined
2.1.2.4	Annotation References	Defined
2.1.3	User Defined References	Defined
2.1.4	Application	Defined
2.1.4.1	Application Type	Defined
2.1.4.2	Application Data	Defined
2.1.4.3	Consumed	Defined
2.1.4.4	Application References	Defined
2.1.5	Copy Flag	Defined

END ADD

ADD FEATURE (initial number 15)
INTERNAL FEATURE ID 4589
ANNOTATION FEATURE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Annotation Elements Array	Defined
2.1	Annotation Element #1	Defined
2.1.1	Annotation Element Name	AE_NOTE2
2.1.2	Annotation	Defined
2.1.2.1	Annotation Type	Note

2.1.2.2	Annotation Id	5
2.1.2.3	Consumed	Defined
2.1.2.4	Annotation References	Defined
2.1.3	User Defined References	Defined
2.1.4	Application	Defined
2.1.4.1	Application Type	Defined
2.1.4.2	Application Data	Defined
2.1.4.3	Consumed	Defined
2.1.4.4	Application References	Defined
2.1.5	Copy Flag	Defined

END ADD

ADD FEATURE (initial number 16)
INTERNAL FEATURE ID 4591

ANNOTATION FEATURE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Annotation Elements Array	Defined
2.1	Annotation Element #1	Defined
2.1.1	Annotation Element Name	AE_NOTE3
2.1.2	Annotation	Defined
2.1.2.1	Annotation Type	Note
2.1.2.2	Annotation Id	6
2.1.2.3	Consumed	Defined
2.1.2.4	Annotation References	Defined
2.1.3	User Defined References	Defined
2.1.4	Application	Defined
2.1.4.1	Application Type	Defined
2.1.4.2	Application Data	Defined
2.1.4.3	Consumed	Defined
2.1.4.4	Application References	Defined
2.1.5	Copy Flag	Defined

END ADD

ADD FEATURE (initial number 17)

INTERNAL FEATURE ID 4593

ANNOTATION FEATURE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Annotation Elements Array	Defined
2.1	Annotation Element #1	Defined
2.1.1	Annotation Element Name	AE_NOTE4
2.1.2	Annotation	Defined
2.1.2.1	Annotation Type	Note
2.1.2.2	Annotation Id	7
2.1.2.3	Consumed	Defined
2.1.2.4	Annotation References	Defined
2.1.3	User Defined References	Defined
2.1.4	Application	Defined
2.1.4.1	Application Type	Defined
2.1.4.2	Application Data	Defined
2.1.4.3	Consumed	Defined
2.1.4.4	Application References	Defined
2.1.5	Copy Flag	Defined

END ADD

ADD FEATURE (initial number 18)

INTERNAL FEATURE ID 4595

ANNOTATION FEATURE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Annotation Elements Array	Defined
2.1	Annotation Element #1	Defined
2.1.1	Annotation Element Name	AE_NOTE5
2.1.2	Annotation	Defined
2.1.2.1	Annotation Type	Note

2.1.2.2 Annotation Id 8
2.1.2.3 Consumed Defined
2.1.2.4 Annotation References Defined
2.1.3 User Defined References Defined
2.1.4 Application Defined
2.1.4.1 Application Type Defined
2.1.4.2 Application Data Defined
2.1.4.3 Consumed Defined
2.1.4.4 Application References Defined
2.1.5 Copy Flag Defined

END ADD
ADD FEATURE (initial number 19)
INTERNAL FEATURE ID 4610
ANNOTATION FEATURE

NO.	ELEMENT NAME	INFO
---	-------------	-------------
1	Feature Name	Defined
2	Annotation Elements Array	Defined
2.1	Annotation Element #1	Defined
2.1.1	Annotation Element Name	AE_NOTE6
2.1.2	Annotation	Defined
2.1.2.1	Annotation Type	Note
2.1.2.2	Annotation Id	9
2.1.2.3	Consumed	Defined
2.1.2.4	Annotation References	Defined
2.1.3	User Defined References	Defined
2.1.4	Application	Defined
2.1.4.1	Application Type	Defined
2.1.4.2	Application Data	Defined
2.1.4.3	Consumed	Defined
2.1.4.4	Application References	Defined
2.1.5	Copy Flag	Defined

END ADD
MASSPROP
END MASSPROP