İZMİR KÂTİP ÇELEBİ ÜNİVERSİTESİ
2010
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

Optimizing the Load Carrying Performance of the Hydraulic Guiding Elements with Finite Elements Method

Submitted to the Graduate School of Natural and Applied Sciences
in partial fulfillment of the requirements for the degree of

Master of Science in Department of Material Science and Engineering

by

Cem TANYERİ

ORCID 0000-0002-1290-2875

May, 2022

This is to certify that we have read the thesis **Optimizing the Load Carrying Performance of the Hydraulic Guiding Elements with Finite Elements Method** submitted by **Cem Tanyeri**, and it has been judged to be successful, in scope and in quality, at the defense exam and accepted by our jury as a MASTER'S THESIS.

APPROVED BY:

Advisor: **Prof. Dr. Mehmet Özgür SEYDİBEYOĞLU**
İzmir Kâtip Çelebi University

Committee Members:

Assist. Prof. Dr. S. Bahar BAŞTÜRK
Manisa Celal Bayar University

Assist. Prof. Dr. İsmail Doğan KÜLCÜ
İzmir Kâtip Çelebi University

Date of Defense: June 27, 2022

Declaration of Authorship

I, **Cem TANYERİ**, declare that this thesis titled **Optimizing the Load Carrying Performance of the Hydraulic Guiding Elements with Finite Elements Method** and the work presented in it are my own. I confirm that:

- This work was done wholly or mainly while in candidature for the Master's / Doctoral degree at this university.
- Where any part of this thesis has previously been submitted for a degree or any other qualification at this university or any other institution, this has been clearly stated.
- Where I have consulted the published work of others, this is always clearly attributed.
- Where I have quoted from the work of others, the source is always given. This thesis is entirely my own work, with the exception of such quotations.
- I have acknowledged all major sources of assistance.
- Where the thesis is based on work done by myself jointly with others, I have made clear exactly what was done by others and what I have contributed myself.

Date: 17.05.2022

Optimizing the Load Carrying Performance of the Hydraulic Guiding Elements with Finite Elements Method

Abstract

Guiding elements, which have an important effect on the long-term trouble-free operation of hydraulic and pneumatic systems, resist the forces perpendicular to the axis in hydraulic and pneumatic cylinders and are used to prevent metal-to-metal contact. In composite materials, which is one of the most frequently used materials in guiding elements, different parameters such as fiber orientations and reinforcement material ratios greatly affect the quality and strength of the product. This study, it is aimed to analyze the transition steps of composite guiding elements from material to the product by using the finite element method and simulation studies using Digimat, Moldex3D, and MSC Marc programs. The analysis studies were carried out in an integrated manner with Digimat used in composite material modeling, Moldex3D used in simulating plastic injection production, and MSC Marc used in the nonlinear analysis for the final product. As a result of the analysis, the filling status, shrinkage value, and pressure value of the product in the cavity were displayed in the Moldex3D program. The fiber orientations exported from the Moldex3D program were processed in the Digimat program and the material model generated. Then, the prepared material model transferred to the Marc program, nonlinear analysis made, and the results examined comparatively. Validation of these simulations provided by the tests carried out at the Test Center of Kastaş Sealing Technologies. As a result of optimization activities, the load carrying ability of the composite hydraulic bearing element has been increased.

Keywords: Composite material, guiding elements, simulation, FEA, modelling of plastic injection process, material modelling

Optimizing the Load Carrying Performance of the Hydraulic Guiding Elements with Finite Elements Method

Öz

Hidrolik ve pnömatik sistemlerin uzun süreli sorunsuz çalışmasında önemli bir etkiye sahip olan yataklama elemanları, hidrolik ve pnömatik silindirlerde eksene dik olan kuvvetleri taşıyarak metal-metal temasını önlemek için kullanılmaktadırlar. Yataklama elemanlarında en sık kullanılan malzemelerden biri olan kompozit malzemelerde, fiber yönelimleri ve takviye malzeme oranları gibi farklı parametreler, ürünün kalitesini ve mukavemetini büyük ölçüde etkilemektedir. Bu çalışmada, kompozit yataklama elemanlarının sonlu elemanlar yöntemi kullanılarak malzemeden ürüne geçiş aşamalarının ve Digimat, Moldex3D ve MSC Marc programları kullanılarak simülasyon çalışmaları yapılması amaçlanmıştır. Analiz çalışmaları, kompozit malzeme modellemesinde kullanılan Digimat, plastik enjeksiyon üretiminin simülasyonunda kullanılan Moldex3D ve nihai ürün için doğrusal olmayan analizde kullanılan MSC Marc ile entegre bir şekilde gerçekleştirilmiştir. Yapılan analiz sonucunda ürünün boşluktaki dolum durumu, çekme değeri ve basınç değeri Moldex3D programında incelenmiştir. Moldex3D programından dışa aktarılan fiber oryantasyonları Digimat programında işlendi ve malzeme modeli oluşturuldu. Daha sonra hazırlanan malzeme modeli Marc programına aktarılarak, doğrusal olmayan analizler yapılmış ve sonuçlar karşılaştırmalı olarak incelenmiştir. Bu simülasyonların doğrulaması, Kastaş Sızdırmazlık Teknolojileri Test Merkezi'nde yapılan testler ile sağlanmıştır. Optimizasyon çalışmaları sonucunda kompozit hidrolik yataklama elemanının yük taşıma kabiliyeti artırılmıştır.

Anahtar Kelimeler: Kompozit malzeme, yataklama elemanları, simülasyon, FEA, plastik enjeksiyon prosesinin modellenmesi, malzeme modellemesi

To my family,

Acknowledgment

I would like to express my gratitude to my advisor, Prof. Mehmet Özgür Seydibeyoğlu for his guidance and suggestions.

I present my thanks to M.Sc. Mechanical Engineer Ozan Devlen from Kastaş Sealing Technologies for their support and guidance during this study. I would like to present my thanks for supplying the materials and letting me use the test facilities in research and development test center of Kastaş Sealing Technologies. I would like to endless thanks for supporting and believing me during my thesis to my family.

I would like to thank to M.Sc. Seçkin Semiz for helping to perform the material characterization tests and criticism during my study.

17.05.2022 Cem TANYERİ

Table of Contents

List of Figures

List of Tables

List of Abbreviations

ANOVA	Analysis-of-Variance
DOE	Design of Experiment
FEA	Finite Element Analysis
MPa	Megapascal
PA	Polyamide
PEI	Polethylemine
POM	Polyoxymethylene
PP	Polypropylene
PTFE	Polytetraflouroethlyne

List of Symbols

N	Force
kgf	Kilogram force
a_{ij}	Fiber Orientation

Chapter 1

Introduction

1.1 Background

In hydraulic systems, vertical loads are carried by guiding elements. In case of metal-to-metal contact in hydraulic systems, damage occurs in the system and the system stops. Due to their higher load bearing capacity usage of composite guiding elements in hydraulic cylinders is increasing day by day while conventional guiding elements usage is decreasing.

Polymer composites are a type of construction material that are classified together. It has been found that adding fibers and particles to the polymer matrix can improve the properties of polymers [1].

Filler materials like glass fiber and graphite are most of added to Polyoxymethylene (POM) to improve mechanical strength and load carrying capacity [2]. The use of fiberglass in POM composites has led to a significant increase in the strength and intensity of the element, a reduction in wear and tear, but an increase in surface roughness and raises friction coefficients [3]. Many groups of researchers conduct research on polymer composites with glass fiber. It has been found that glass fibers can help to strengthen polymers, resulting in increased strength properties, resistance to creep, and fatigue, as well as improved dimensional stability [4-7].

The carbon fiber incorporated into the POM matrix provides better strength properties and also has a positive effect on the abrasion resistance. Many groups of researchers conduct research on polymer composites with carbon fiber. It has been found that carbon fiber incorporated into the POM matrix not only provides better strength properties but also positively effects on the abrasion resistance. With increasing fiber content, the coefficient of friction and wear are reduced [8-11].

Moldflow and Moldex3D flow analysis simulation programs have been used frequently in studies on short fiber reinforced polymer materials. The fiber orientation data obtained were transferred to the digimat software and a transition was made from the micro anisotropic model to the macro isotropic model. Finite element analysis software and Digimat software were used in coupled with the obtained material data [12, 17, 27].

In the literature review, no direct subject related to hydraulic short-reinforced fiber polymer hydraulic bearing elements has been found, and it is aimed to increase the load-bearing capability of the hydraulic pom composite guiding element in the light of the data collected from previous scientific studies in this field. In this study, which was carried out by examining the literature research, it was emphasized that the material load carrying capacity can be improved by optimizing the injection mold geometry in addition to improving the material strength by changing the composite additive ratio in the material. To successfully model short fiber reinforced POM material in FEA a multi-level approach similar and mean-field homogenization method used similar to other researchs [12, 27]. In addition to previous studies optimized mold geometry parameters for hydraulic guiding element were obtained by applying Design of Experiment (DOE) and Analysis of Variance (ANOVA).

1.2 Definitions

1.2.1 Hydraulic Cylinder

The hydraulic cylinder is a circuit element that converts hydraulic energy into mechanical energy and is used to provide linear motion. The hydraulic energy generated by the pump is converted into linear or angular motion with the help of a cylinder. Cylinders can be made in a variety of shapes and sizes, depending on their intended use. These are machine elements that are generally used in systems that require excessive forces and have a velocity that is below 0.5 m / s [13].

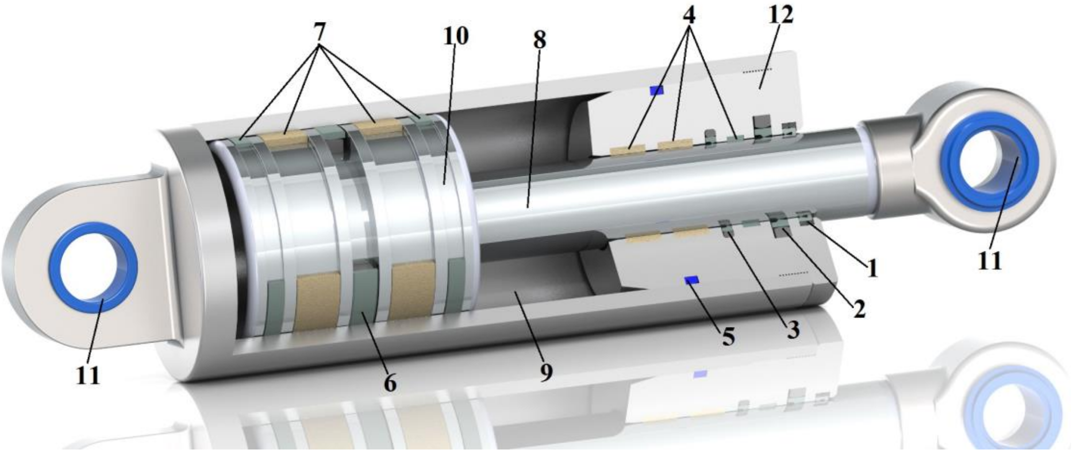


Figure 1.1: Hydraulic Cyinder

The cylinder located above, has the parts detailed below;

1: Wiper seal

2: Rod seal

3: Rod buffer seal

4: Rod guiding rings

5: Static seal

6: Piston seal

7: Piston guiding rings

8: Rod

9: Bore / Barel

10: Piston

11: Cylinder joints

12: Head [14]

1.2.2 Guiding Element

Guiding elements are used to prevent metal to metal contact in hydraulic cylinders. The cylinders help to carry the axial loads during the operation. By ensuring that the cylinder works in a centralized manner, the seal elements prevent crushing under load and create a safe working environment. The materials used in these pieces are typically very strong and able to withstand a lot of strain, without breaking or becoming distorted. The guiding elements do not act as a seal in the system; geometries of the guiding elements allow the passage of fluid.

Most of todays systems use thermoplastic, PTFE and polyester resin guiding elements.

Advantages of non-metal guiding elements stated below;

- Easy to assemble, easy to replace maintenance
- Low-cost solutions
- High-loading capacity
- High wear resistance and long service life
- The ability to suppress vibration systems
- Low friction
- Doesn’t create hydrodynamic pressure
- Work, no damage to metal surfaces

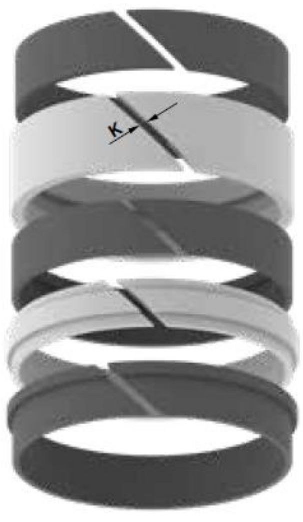


Figure 1.2: Guiding Elements

1.2.3 Guiding Element Materials

1.2.3.1 Thermoplastic Guiding Elements

POM or PA guiding elements are generally used as glass fiber reinforced or pure. POM and PA bearing elements are preferred because they are economical. At temperatures of 60°C and above, POM and PA guiding elements lose their load-carrying capacity and surface contact pressure, like other thermoplastics. They are suitable for use in light and medium duty applications.

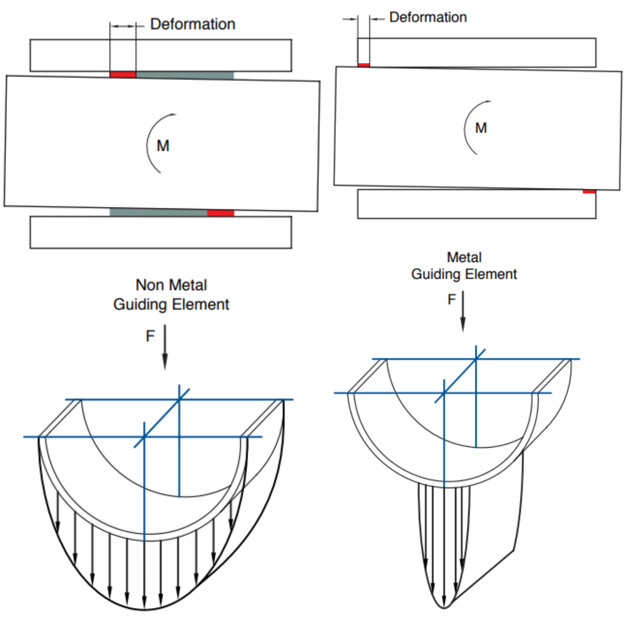
POM is a very crystalline plastic material that has a high degree of melting temperature. This polymer is made from formaldehyde by either polymerization into a single chain (POM-H) or blending with cyclic ethers (POM-C). The mechanical properties of acetal materials depend on the percentage of crystalline material. The value can be around 70% to 75%. The high degree of crystallinity in the polymer contributes to its high stiffness and resistance to fatigue and creep stresses. This feature enables polys (oxymethylene) to run under variable load conditions. The high percentage of crystals in the material makes it very hard and resistant to wear and tear. It also has good strength and resilience to high temperatures. POM is a very strong and stiff thermoplastic with good dimensional stability [15].

1.2.3.2 PTFE Guiding Elements

Polytetrafluoroethylene (PTFE) guiding elements are used in systems where high temperature, chemicals exist in the working environment and low friction forces are required. Properties of PTFE guiding elements; bronze, carbon and molybdenum disulfide additives can be used to make guiding elements more suitable for the system they will work in. Good elasticity properties of PTFE guiding elements are the main choice reasons for designs. In some applications, PTFE guiding elements are used with other guiding elements with higher load carrying capacity. In such applications, PTFE guiding element collects foreign particles in the environment and prevents these particles from sticking to the harder guiding element and damaging the cylinder or rod. They are used in light and medium duty applications due to low contact pressures.

1.2.3.3 Polyester Resin Guiding Elements

They are products that are composed of a combination of cloths such as cotton, polyester, aramid etc., resins and different filling materials. They have high load carrying capacities and can be used in heavy duty applications. Polyester resin guiding elements create a much better bearing area thanks to their elastic structure; thus, it is more successful in carrying the radial forces in the system. Load distribution is close to homogeneous in the polyester resin guiding elements, in this manner they prevent the problems caused by dry running because of misalignments in the system that may occur due to high elastic deformations. Resin prevents break apart of pieces from guiding elements. PTFE additive in polyester resin guiding elements reduce friction. Polyester resin guiding elements maintain their dimensional stability at high operating temperatures very well compared to other guiding elements. Nowadays, the load carrying capacities of the guiding elements, which are mostly preferred in the sectors where medium and heavy-duty cylinders are used, changes with the effect of temperature and speed. Load carrying capacity decreases as temperature and speed increase. Figure 1.3 shows pressure distribution on metal and non metal guiding elements.



(a) (b)

Figure 1.3: Pressure Distrubition on Guiding Element, (a) Non Metal Guiding Element, (b) Metal Guiding Element

Figure 1.4 shows surface pressure and operating speed relation on guiding elements.

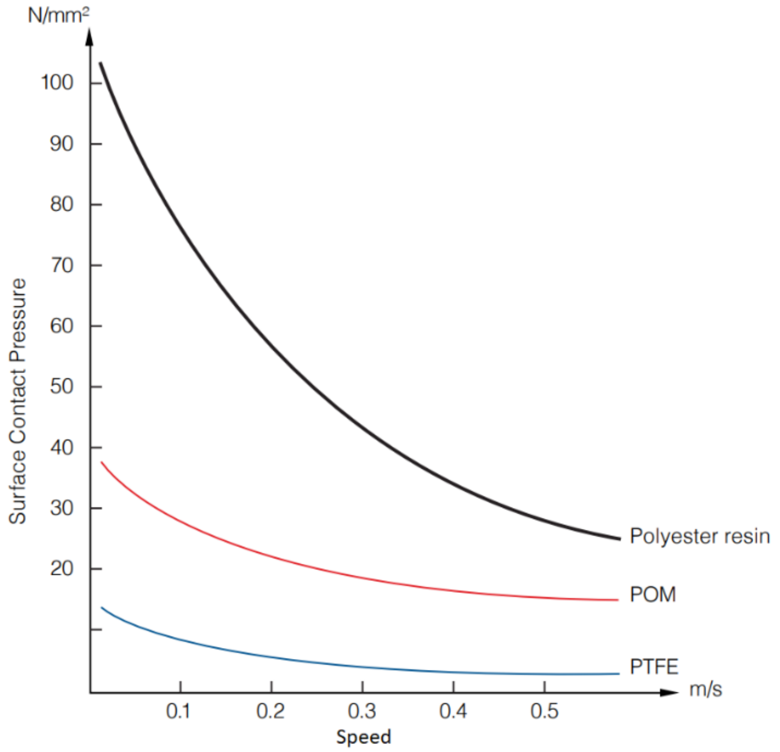


Figure 1.4: Surface Contact Pressure-Speed Relation on Guiding Element Materials

1.2.4 Guiding Element Forms

Guiding elements can be manufactured in various designs. It can be produced in the form of flat, «U», «L» or «T» depending on the applications and groove types they will be mounted on. «L» and «T» type guiding elements are generally seen in telescopic cylinder applications. Guiding element forms can be seen in Figure 1.5. They can be produced from glass fiber reinforced thermoplastic materials. Load carrying capacities can be increased with the glass fiber additive. In this study, flat form guiding element was used.

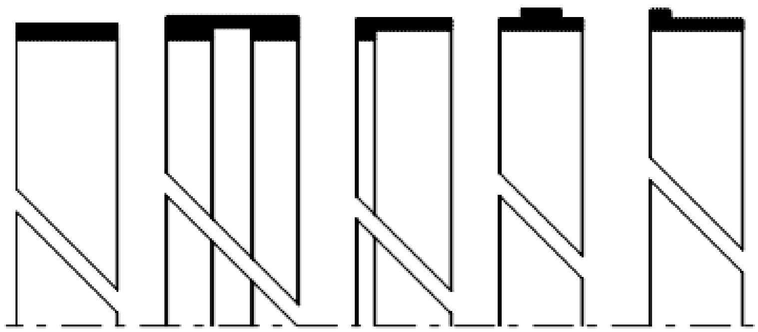


Figure 1.5: Guiding Element Forms

1.2.5 Finite Element Analysis

Finite Element Analysis (FEA) is a computer-aided engineering technique that predicts the behavior of different designs and materials under certain boundary conditions.

To evaluate the sealing performance of sealing elements thanks to FEA; it is possible to study the force and deformation reactions, predict friction forces, yield values and assembly forces.

FEA permits the production of sealing elements with better performance in the same conditions [13].

1.2.5.1 Moldex3D

Moldex3D program is software that provides product efficiency and optimization by simulating the plastic injection process, changing the process and mold design parameters. Below are sample images shown in fiber flow.

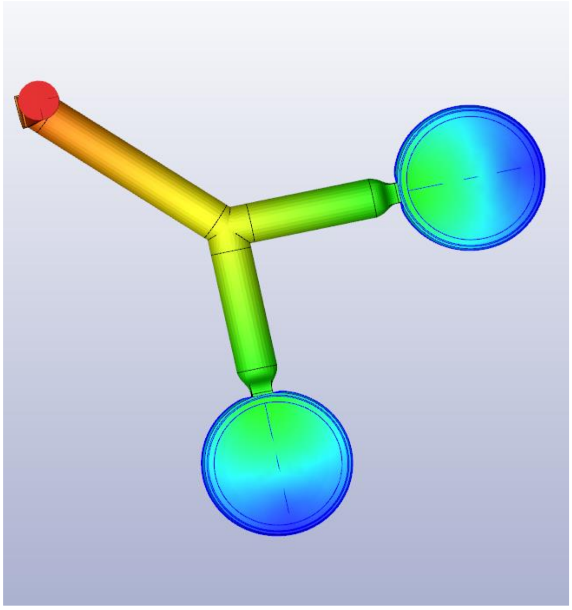


Figure 1.6: Moldex3D image

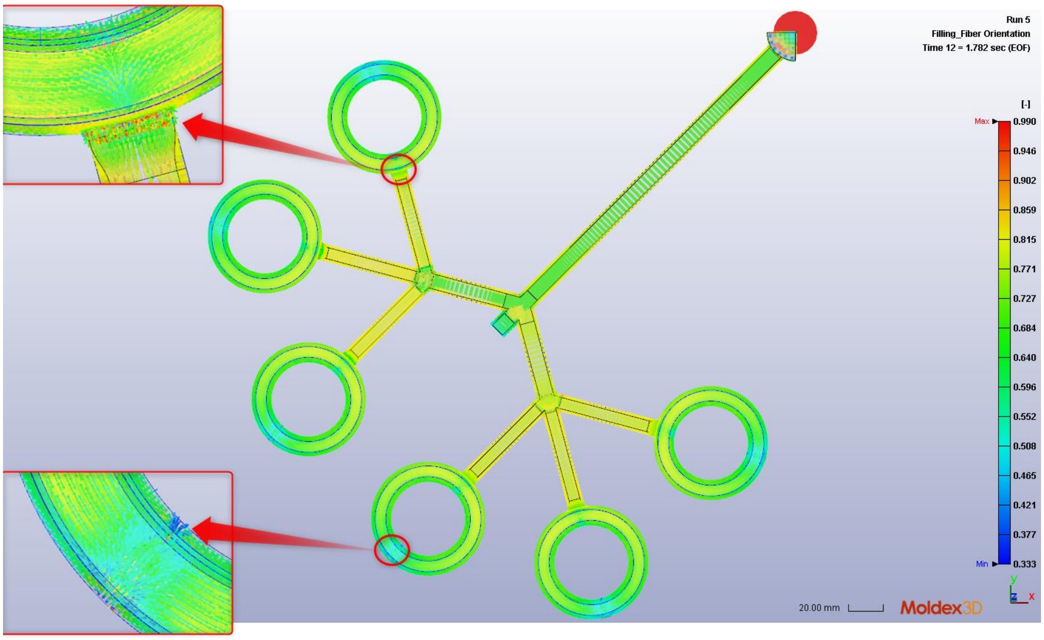


Figure 1.7: Fiber Orientation

Moldex3D commonly used to simulate the plastic, wax, and powder injection molding process. The software has an e-Designer module which is used to create a CAD model, design the injection gate, mould, cooling channels, etc. According to the requirements. The simulation results show how the injection moulding process changes as the parts move through four stages: filling, packing, cooling, and warpage [16]. Yan FY stated that usage of injection molding analysis softwares like Moldex3D or Moldflow helps to study the flow patterns of polymer melt and fibers inside the mould during injection, packing and cooling processes [2]. These softwares also helps to predict and overcome possible injection molding problems. With the use of Moldex3D software, the mold design process is shortened and the cost of the final product is reduced.

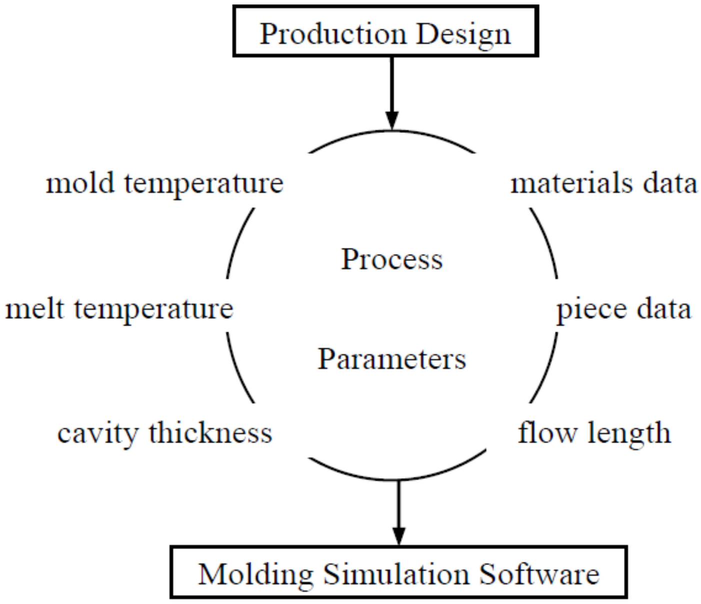


Figure 1.8: The process flow of injection molding design with Simulation Software [2]

1.2.5.2 Digimat

It is the software that enables the composite materials to be used in FEA software to be modeled more effectively and more accurately, enabling higher quality and more innovative products to be released in a shorter period, thus saving time and money. In this study, since composite bearing elements were developed, the Digimat RP module was used.

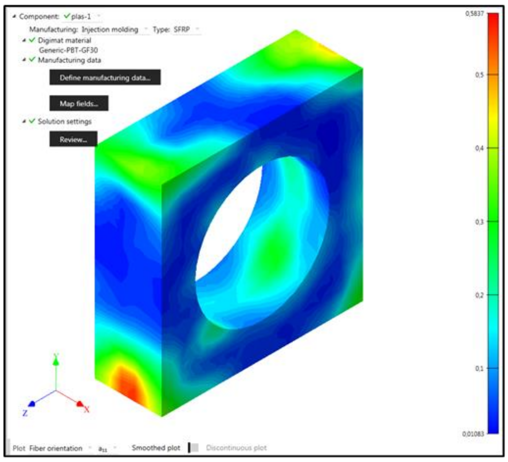


Figure 1.9: Digimat Image

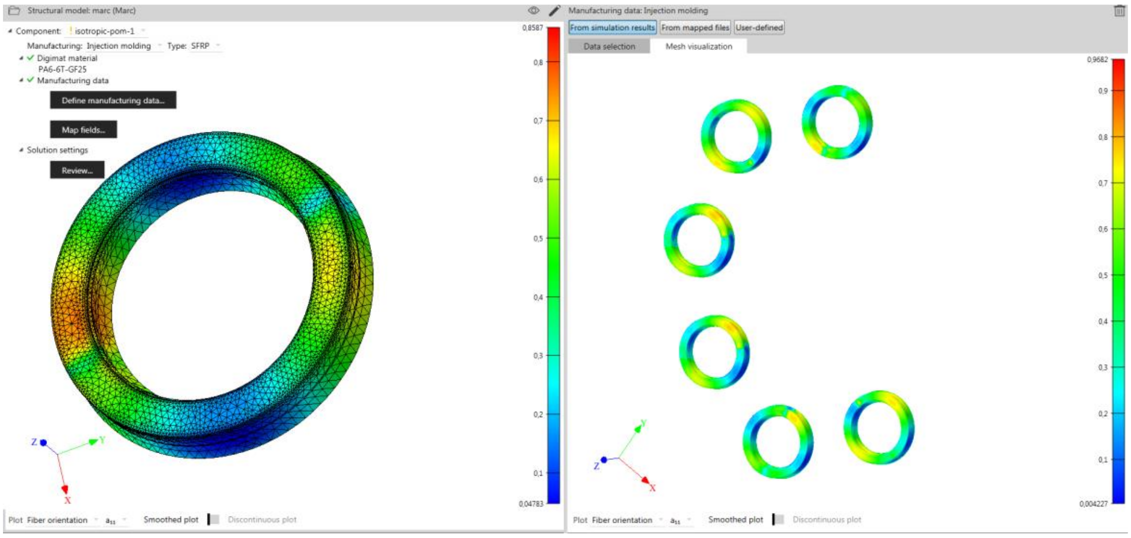


Figure 1.10: Material Mapping in Digimat

There are many stuidies which analyzes filled composite polymer matrixes with Digimat; for example, Anna K. studied on Basalt/Glass Fiber Polypropylene Hybrid Composites with Digimat and Moldex3D softwares [17]. Digimat is utilized to represent the anisotropic material in this study similar to the research done by Jan Anders Lindhult and Miranda Ljunberg [27], with fiber orientations from Moldex3D simulations mapped onto the structural mesh. Separate models for the matrix and fiber materials, as well as additional microstructure information, are used to represent the short fiber reinforced polymers in Digimat. The microstructure is homogenized in Digimat using the mean field homogenization approach in order to estimate the macroscopic material properties.

Basic computing workflow of Digimat software shown in Figure 1.11.

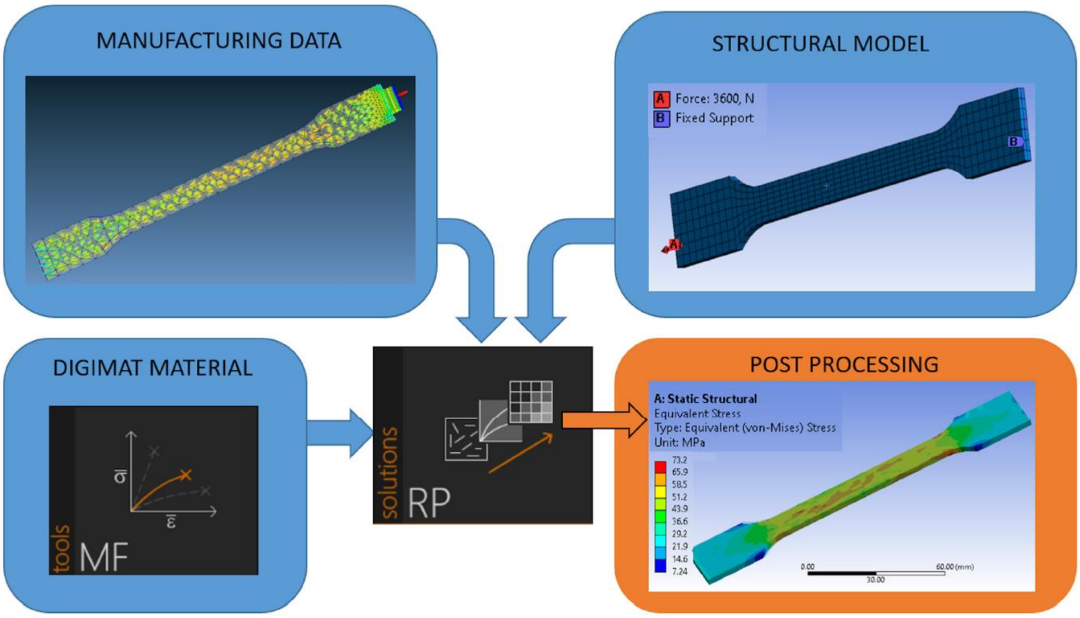


Figure 1.11: Digimat Computing Flow Chart

With the Digimat RP module, it is ensured that the production effects of easy-to-use reinforced plastic parts are reflected on the analysis modules, process simulation and optimization and finite element simulations are performed.

It is an easy and efficient solution for the design of fiber reinforced plastic parts [18].

Digimat RP module is especially used for chopped fiber reinforced plastics. Usable basic resins are polyamide (PA), polypropylene (PP), polyoxymethlyne (POM) and Polethylemine (PEI) with glass and carbon fiber materials [19].

1.2.5.3 Marc

Marc is a versatile, general-purpose FEA solver that can accurately model the product behavior under static, dynamic and multiple physics loading scenarios. Marc's skills in modeling nonlinear material behavior and transient environmental conditions make him the perfect person to help you solve complex design problems. [20]

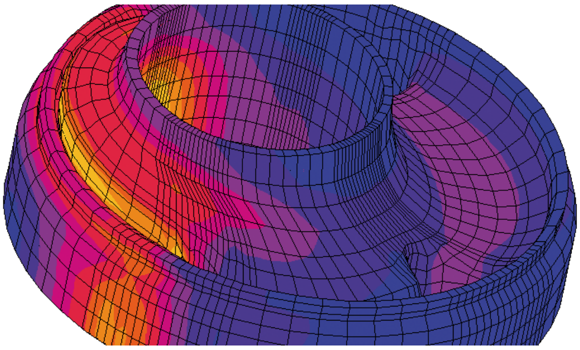


Figure 1.12: Msc Marc Image

In this study, fiber orientations were obtained with Moldex3D and transferred to digimat software. With the Digimat software, the data obtained from Moldex3D processed and the material generated through Digimat RP model. Later this material data mapped on the 3D material geometry and transferred to the solver FEA software.

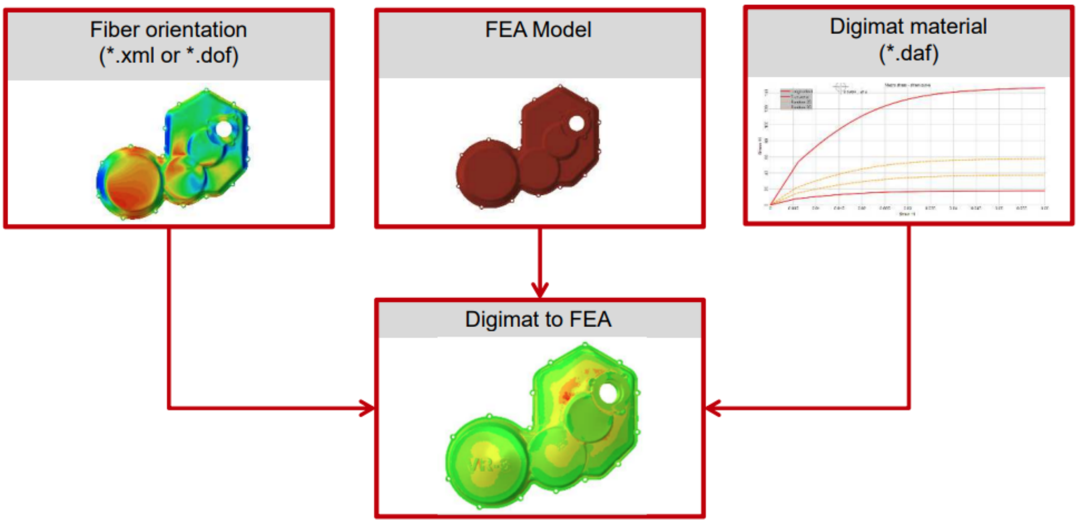


Figure 1.13 Applying Moldex and Digimat in FEA [19]

1.2.6 Injection Molding Systems

Injection molding is the most common manufacturing process for plastic products. The machine is well-suited for mass production of plastic parts with complex shapes that require precise tolerance. This process involves forcing hot polymer melt into a cold empty cavity, and then holding the resulting solidified product under a high pressure [21].

A typical injection molding machine has three main units: the clamping unit, the plasticating unit, and the drive unit. The clamping unit holds the injection mold in place. The tool is able to close, clamp, and open the mold. The main parts of the machine are the fixed and moving plates, the tie bars, and the mechanism for opening, closing and clamping.

Injection molding machines are generally classified by maximum clamp force. Clamp force is the force holds both halves of injection mold together to avoid opening two halves of the mold because of the internal pressure of the melt plastic material [22].

Figure 1.14: Plastic Injection Machine

1.2.6.1 Injection Molding Cycle

Injection molding cycle basically consists of 4 stages. In stage 1 both halves of the injection mold closes and melt plastic material injected into the mold. In stage 2 injection molding machine holds pressure to ensure minimal shirinkage. In stage 3 melt material coolsdown and solidifies. In stage 4 both mold halves open and the ejector pins of the injection machine pushes the material outside of the mold.

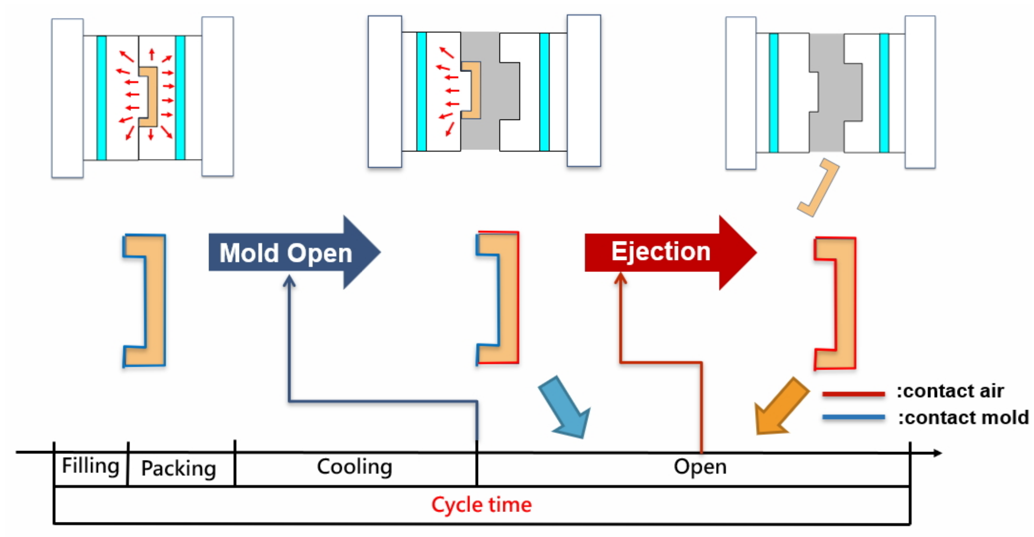


Figure 1.15: Injection Cycle [23]

1.2.6.2 The Runner System

Injection molding is a manufacturing process that produces plastic products with precise dimensions and a consistent structure. It is most commonly used in industrial settings. The gating system for injection moulding is also called the runner system. The feeding channels lead the plastic melt to the cavity from the injection machine nozzle. The main runner, branch runner, gate, and cold slag cavity are usually present in runner system. The quality of plastic products and production efficiency are closely related [24].

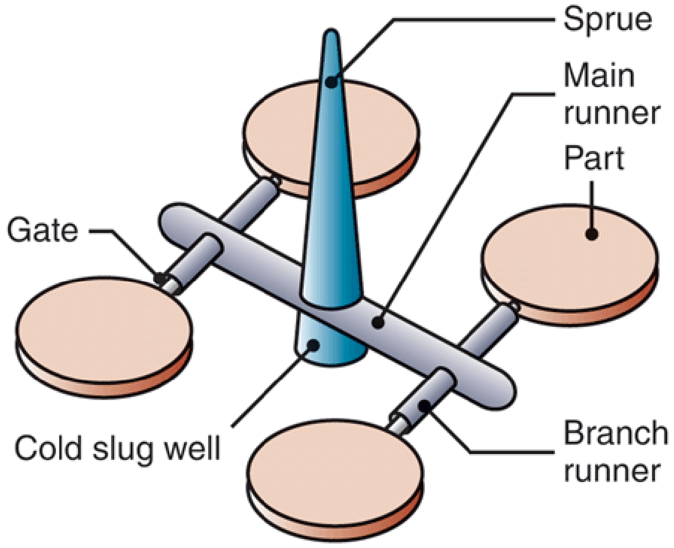


Figure 1.16: Injection Runner System

The gate is also known as the feeding port, which is the narrow channel between the branch runner and the mould cavity, and Injection Mould Gates also the shortest and thinnest part of the process. Its purpose is to use the tightening flow surface to make the plastic faster. The high shear rate makes the plastic fluidity good (due to the plastic shear thinning characteristics); The heating effect of viscous heating also has the function of raising temperature and reducing viscosity. After the moulding is complete, the gate is cured to seal the cavity and prevent the reverse flow of plastic. This will help to keep the forming products compact and ensure that the sinking function is functional. It is handy to cut off the runner system and plastic parts after they have been formed.

Main runner is the part of the injection machine that connects to the sub-runner and the cavity. The top of the main runner is concave, making it convenient to connect with the nozzle. The inlet diameter of the main runner should be slightly larger than the nozzle diameter (0.8 mm) to prevent the overflow and prevent the block due to the wrong connection. The inlet diameter is determined according to the product size, generally 4-8 mm. The main runner's diameter should expand by about three to five degrees inwards to make demoulding the runner slag easier.

Branch runner is the channel which connects the main runner and each cavity in the multi-groove mould. The arrangement and distribution of the sub-runners in the plastic mould should be symmetrical and equidistant in order to fill the cavity with equal velocity.

Cold slag cavity is a hole or cavity located at the end of the main runner. The collector is used to collect the cold slag that is produced after two injections of the injection nozzle, which helps to prevent blockage of the sub-runner and the gate. When the cold slag is mixed into the cavity of the injection mould, it makes it easy to create internal stress in the products. The diameter of the cold slag cavity is about 8 to 10 mm and the depth is about 6 mm.

Chapter 2

Materials and Method

2.1 Materials

In this study Kastaş PM9902 (25% short glass fiber filled POM) is used as base material. PM9902 material was kindly provided by Kastas Sealing Technologies. (see Figure 2.1) This material will be referring to as POM (25% GF) throughout the entire study. PM9902 is a plastic made from polyoxyethylene (POM) and has excellent mechanical, thermal, and chemical properties. Additionally, it has great dimensional stability and resistance to wear and tear.

Figure 2.1: POM (%25GF)

Mechnical properties of POM (%25GF) presented below.

Table 2.1: Mechanical Properties of POM (%25GF)

Properties	Standard	Value	Unit
Specific Gravity	ASTM D792	1,59	g/cm^3
Molding Shrinkage (Flow), 3.2mm	ASTM D955	0,4 ~ 0,9	%
Melt flow rate	ASTM D1238	10	g/10min
Tensile Strength	ASTM D638	128	N/mm^2
Elongation at Break	ASTM D638	3	%

In this study, FEA were carried out with 20% and 25% glass fiber added POM materials. K68 guiding elements are injected at injection molding machines with parameters suggested by raw material supplier.

Table 2.2: Injection Molding Parameters

Filling	**Item Data**	**Unit**
Filling Time	6	sec
Melt Temperature	200	°C
Mold Temperature	80	°C
Max Injection Pressure	200	MPa
Injection Volume	50.20	cc
Packing		
Packing Time	10	sec
Max Packing Pressure	200	MPa

K68 guiding element which used in this study is shown below.

Figure 2.2: K68 Guiding Element

2.2 Method

2.2.1 Analysis Flow Chart

In this study K68 guiding element was taken as a reference and accepted as the base product in the improvement processes. Analysis flow chart is shown in Figure 2.3. First 3D Model designed in Solidworks and exported to Msc Marc software to identify boundary conditions and create loadcases. Moldex3D software is used to simulate plastic injection process and observe fiber orientetation in the part. Data gathered from Moldex3D and Marc used in Digimat to successfully create composite material model for different glass fiber filler ratios and different gate, runner designs. Finally, with the obtained material data, FEA is performed in the Digimat program with MSC Marc as a cooperative software. Guiding element deformed 0,2 mm axially and estimated load bearing capacities measured for each design calculated with FEA and the consistency of the estimated results checked with guiding element test rig.

The same analysis flow processes were repeated for each different runner and gate designs and for each material with different fiber reinforcement ratios. The operation of the analysis flow chart is detailed in the rest of this study.

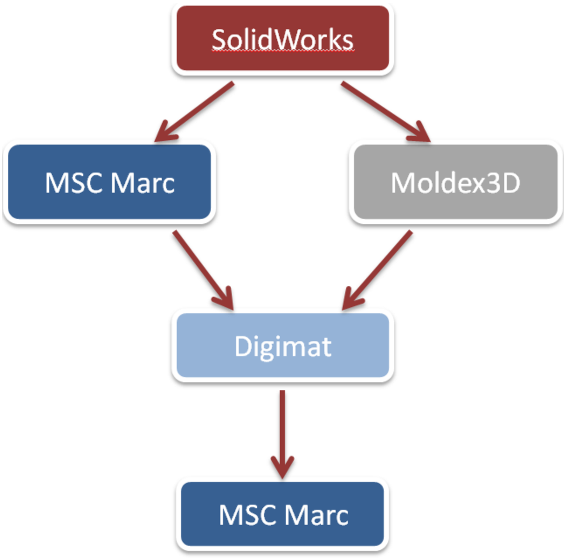


Figure 2.3: Analysis Flow Chart

2.2.2 Msc Marc Analysis

The product designed with this program is simulated according to the working conditions. In this study, the results of nonlinear analyzes in position and loading conditions determined due to usage conditions and tests carried out in the test center for validation. Outputs are given as distribution of Equivalent Cauchy Stress and Contact Stress.

2.2.2.1 CAD Model

Analyzes were made for the K68 Guide Element drawn in the Solidworks program. The model of the Guide Element is shown in Figure 2.4. Model CAD data has been transferred to MSC Marc environment.

Figure 2.4: Cad Model

2.2.2.2 Mesh

K68 is divided into finite elements as indicated in Figure 2.5 by using SimXpert 3D meshing tool.

The number of elements and the number of nodes are shown in Table 2.2.

Table 2.3: Mesh Element, Node Numbers Msc Marc

Product Code	**Number of Elements**	**Number of Nodes**
K68	4773	9899

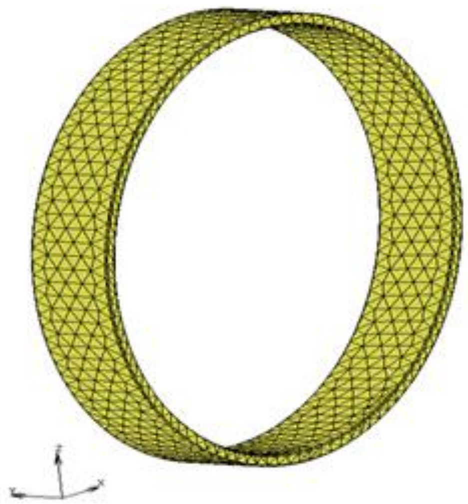


Figure 2.5: K68– Mesh Structure- Overview

Element Type 127 used in this study. This element is a second-order isoparametric three-dimensional tetrahedron. Each edge forms a parabola, with four nodes defining the corners of the element and six nodes defining the position of the "midpoint" of each edge. The stiffness of this element is computed using the four-point integration algorithm. The mass matrix for this element can be approximated using sixteen-point Gaussian integration [25]. Figure 2.6 shows the form of element 127.

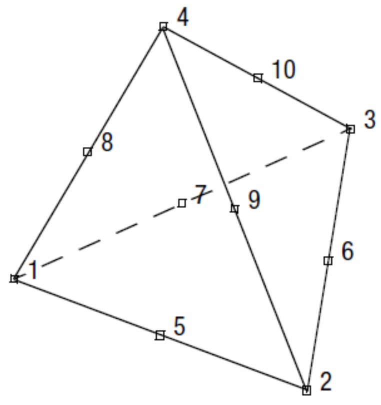


Figure 2.6: Form of Element 127

This allows for an accurate representation of the strain field in elastic analyses.

2.2.2.3 Material Model

In this study anisotropic chracteristics of short fiber reinforced plastic homogenized and converted to isotropic macro structure using Mori-Tanaka mean field homoeginzation method.

Elastic-Plastic Isotropic material model used to represent POM engineering material in Msc Marc. Hooke's Law states that the stress and strain in an object change in a linear fashion with each other. The figure below shows that stress is inversely proportional to strain in a uniaxial tension test. The modulus of elasticity is the well-known measure of the stiffness of a material. Uniaxial stress-strain relation of linear elastic material is shown on Figure 2.7.

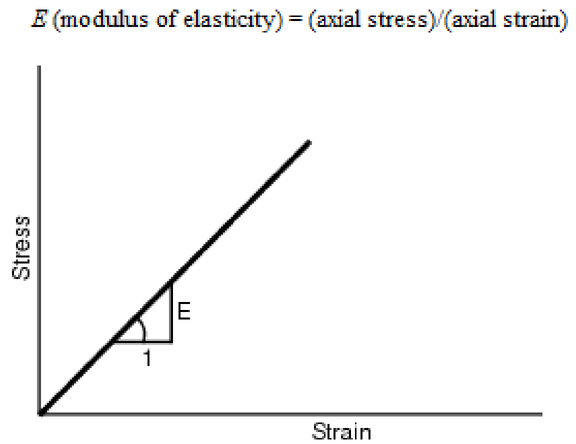


Figure 2.7: Uniaxial Stress-Strain Releation of Linear Elastic Material

During the coupled analysis, Msc Marc will interact with Digimat to compute the macroscopic stress response at each integration point for all composite elements. At each iteration of the analysis, Digimat assesses the macroscopic response using the Mori-Tanaka mean field homogenization model. The orientation data from each integration point in the elements is used to calculate the composite properties. [26].

2.2.2.4 Boundary Conditions

The data obtained from the nominal operating parameters of the product were used as the loadcase and boundary conditions. The analysis setup is as the first step is to place the guiding element into groove, the second step is to complete the assembly process by inserting the rod into the product, and as the last step, the force is applied perpendicular to the rod. The K68 guiding element, which is suitable to be used in construction machinery, cranes, injection benches, agricultural machinery, light and medium duty cylinders, is produced from glass fiber reinforced POM material. In this case, the fiber orientation of the glass fibers used during the production of the material is of great importance in terms of the strength of the products.

Boundary conditions determined considering these conditions were applied as seen in Figure 2.8.

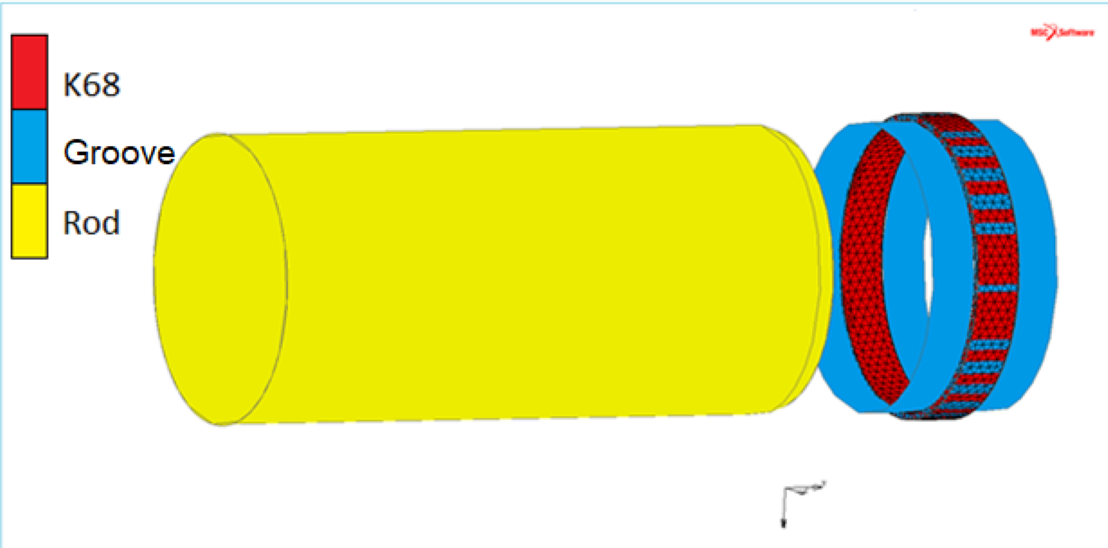


Figure 2.8: Boundary Conditions

2.2.3 Plastic Injection Process Analysis with Moldex3D

It is the program used to simulate the injection molding process with Moldex3D to optimize product design and manufacturability and maximize product quality. In this study, it is aimed to see and optimize the geometry of mold and gate. Glass fiber orientation is calculated with the help of Moldex3D which is critical for the performance of guiding elements. The parameters used in the current production are used for the analysis data for injection machine process parameters. The filling status of the product, the shrinkage value, and the pressure value it creates can be calculated and exported from Moldex3D. In addition, the fiber orientations and mesh structure resulting from the process will be exported in accordance with the MSC Marc program to be used as input in the Digimat Program.

2.2.3.1 CAD Model

Analyzes were made for the K68 Guide Element drawn in the Solidworks program. The CAD data of the model of the Guiding Element shown in Figure 2.7 has been transferred to the Moldex3D environment.

Table 2.4: Moldex3D Mold Design Variables

Program	Variables	Description	
Moldex3D	Melt Material Inlet	X1	Melt material inlet diameter
		X2	Melt material outlet diameter
	Runner	X3	Runner Width
		X4	Runner Angle
		X5	Runner Depth
	Gate	X6	Gate inlet Thickness
		X7	Gate outlet Thickness

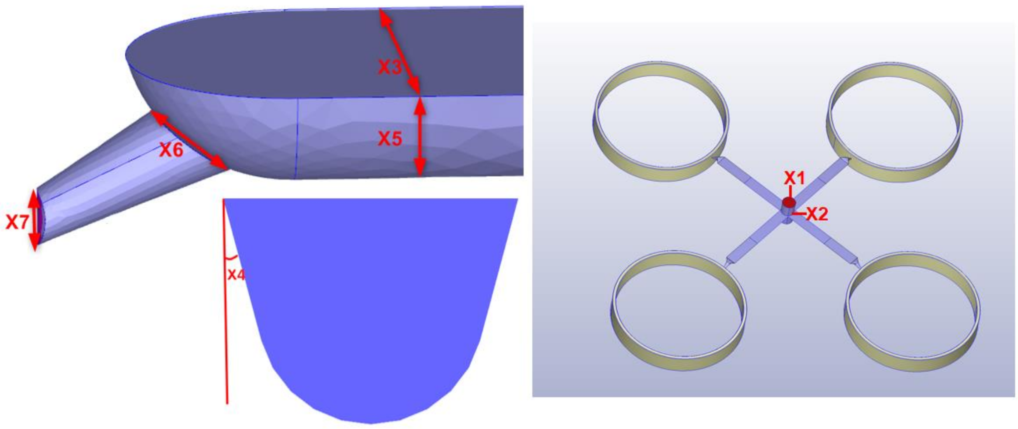
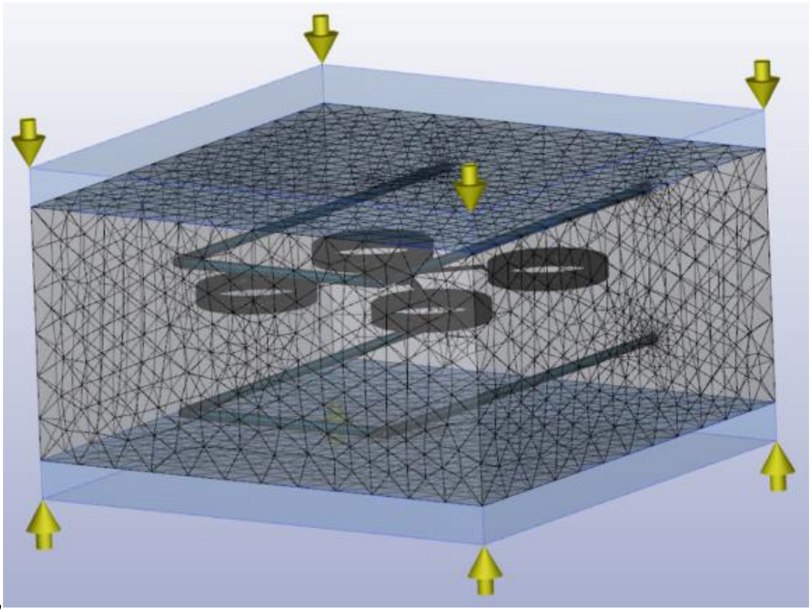


Figure 2.9: K68 Cad Model, Moldex3D

2.2.3.2 Mesh

The mold of the K68 bearing element is divided into elements as shown in Figure 2.8.



,

Figure 2.10: K68 Mesh Structure, General View

The element numbers and node numbers of K68 are shown in Table 2.

Table 2.5: Mesh Element, Node Numbers Moldex3D

Product Code	**Number of Elements**	**Number of Nodes**
K68	21092	41596

2.2.3.3 Boundary Conditions

Loadcase and boundary conditions were determined according to the parameters in Figure 2.10 taken from the plastic injection machine. Since the fiber orientation of the glass fibers used during the production of K68 products made of glass fiber added POM material is of great importance in terms of the strength of the products, the process parameters were taken from the plastic injection unit. According to these data, the analysis is defined as the modeling of the runner inlet, main mold, cooling channels used during production, and the input of molten material and the boundary conditions of the cooler inlet and outlet. According to the process parameters, the plastic injection process was designed.

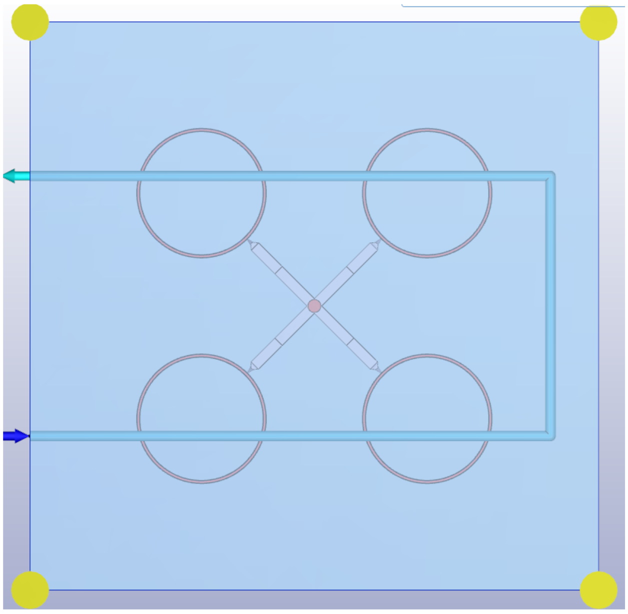


Figure 2.11: Boundary Conditions Moldex3D

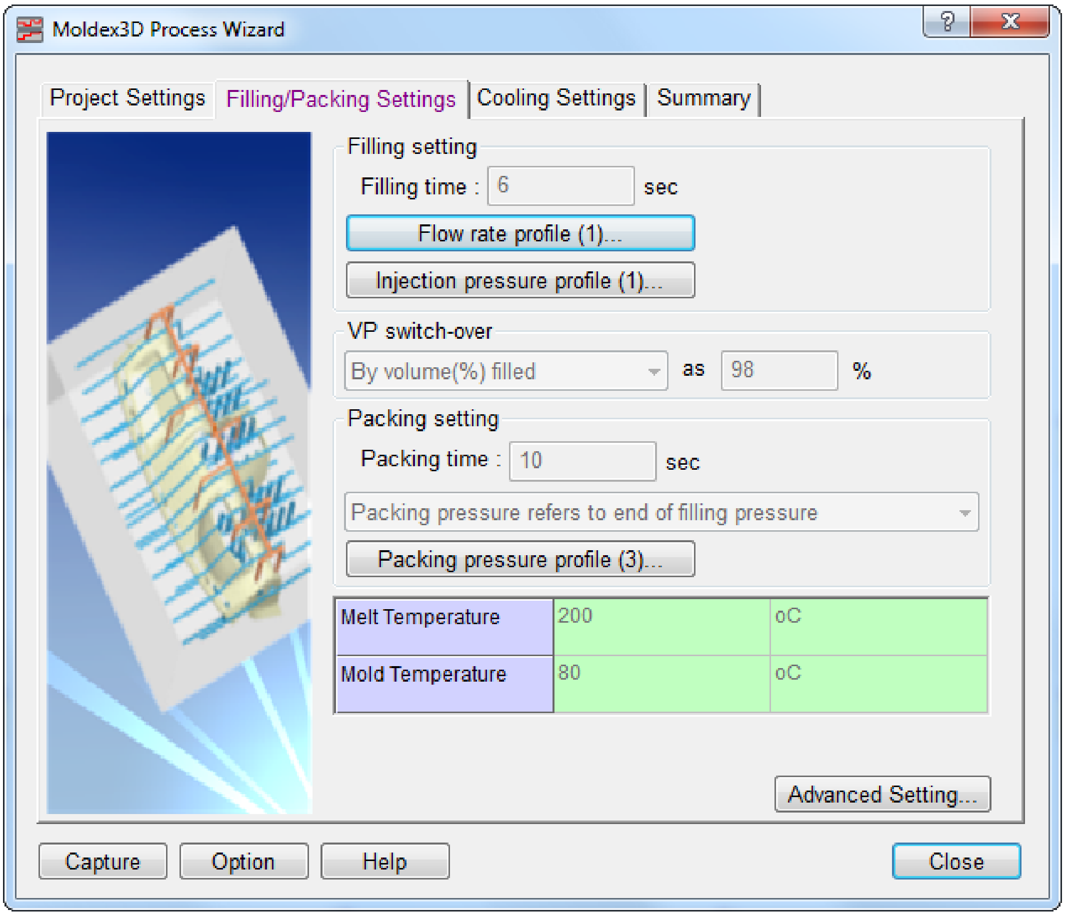


Figure 2.12: Defining Process Parameters to Moldex3D

The analysis results are exported to the Digimat software in the continuation of the process.

2.2.3.4 Fiber Orientations

The orientation of the fiber strands depends on the geometrical features of the component, such as thickness, ribs, and injection mould gate location. Other factors that can affect the quality of a injection-moulded product are processing variables such as the melt and mould temperature [27]. Because material properties vary along and across the fiber direction, a complex anisotropic material with varied stiffness, yield, ultimate, and fatigue strength in each material point and direction results. In several analysis tools, such as Moldex3d and Digimat, the fiber orientation is described by the orientation tensor, aij, as shown in Figure 2.13.

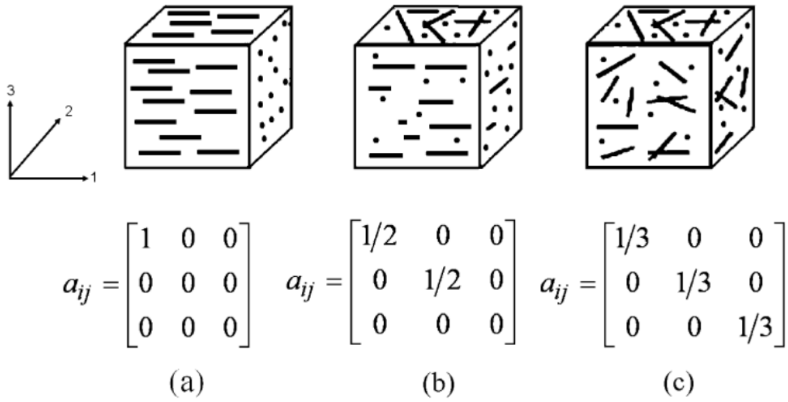


Figure 2.13: Examples of orientation tensors a) Unidirectional 1-direction b) Randomly oriented in 1-2 plane c) Randomly orientated in 3D

The orientation tensor is based on the orientation distribution function and defines the fiber orientation distribution for a single material point. The diagonal terms characterize the intensity of fiber orientation in the 1, 2, and 3 directions, but the offdiagonal terms represent a redistribution of intensities in angular space [26].

2.2.4 Material Modelling with Digimat

2.2.4.1 Model

The CAD data of the Guiding Element model shown in Figure 2.11 were transferred to the Digimat environment with the structural analysis data made in MSC Marc program.

Figure 2.14: K68 CAD Model –Digimat

2.2.4.2 Mapping

The mesh data that comes with the analysis data transferred from the MSC Marc program to the Digimat environment and the mesh files sent in the Moldex3D program are overlapped (mapping) in the Digimat environment.

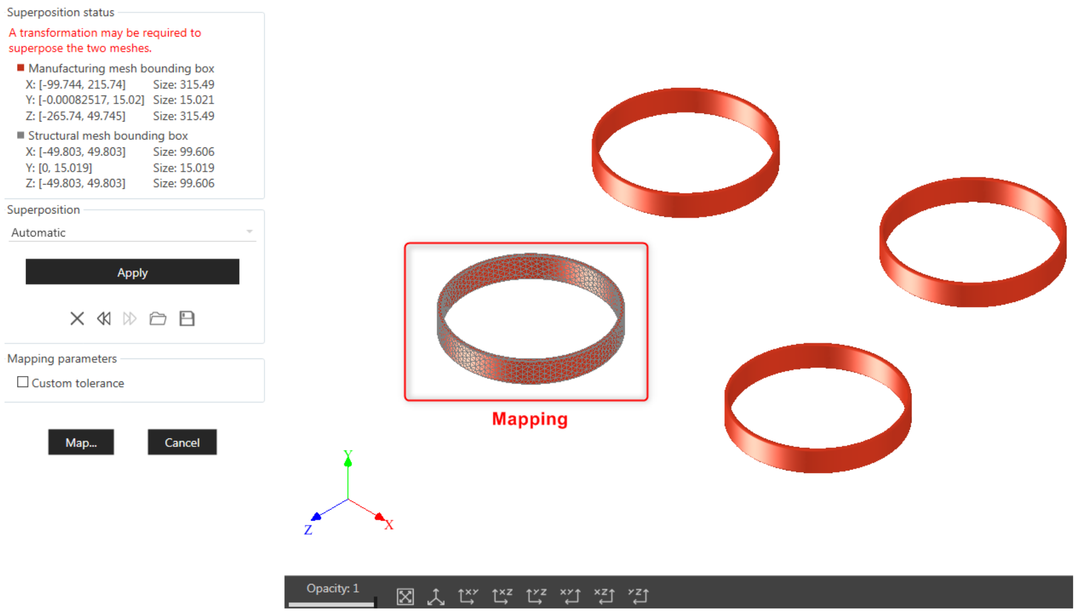


Figure 2.15: K68– Mapping- General View

After the mapping process, the image of the fiber orientations is as in Figure 2.13 and Figure 2.14.

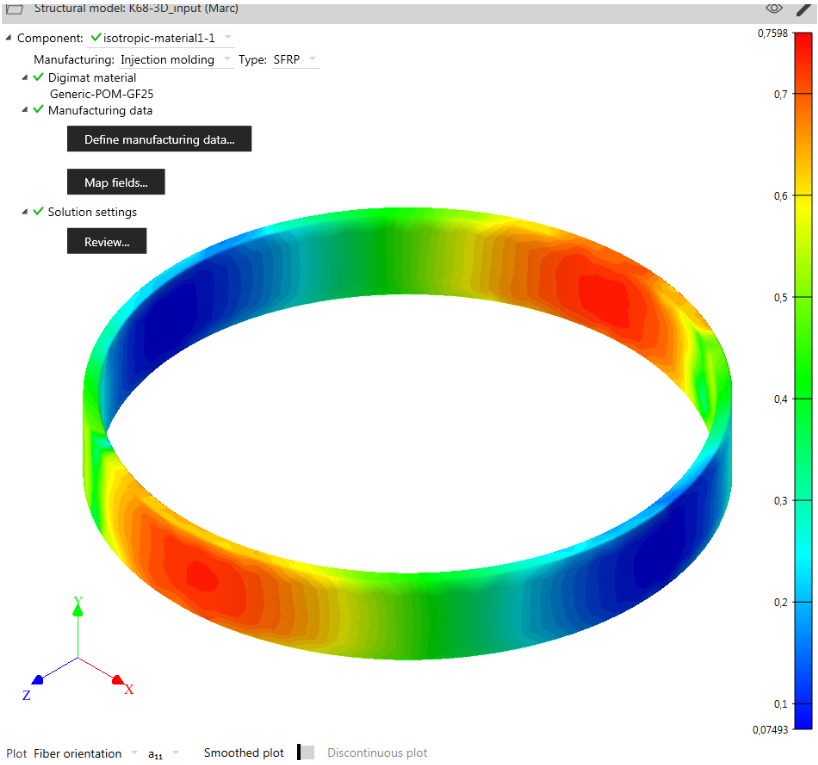


Figure 2.16: K68 Fiber Orientation- Color Scale View

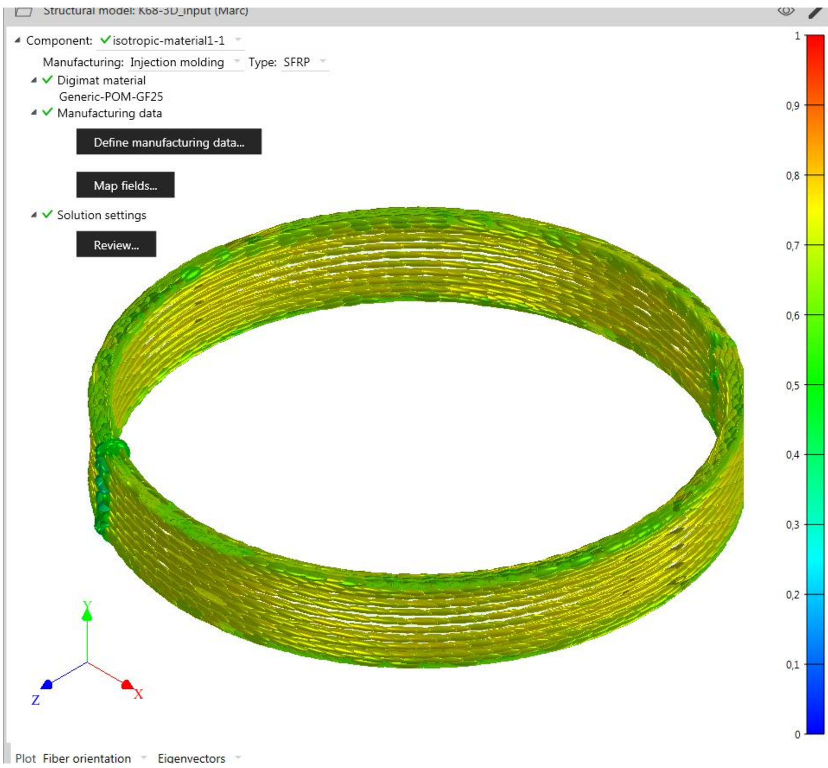


Figure 2.17: K68 Fiber Orientation- Vectoral View

2.2.4.3 Material Modeling

From the test results, the material model is created as in Figure 2.15 with the data created with the Digimat-MX module or transferred from the outsource. The outputs obtained as a result of the analysis are exported to be run in the MSC Marc program.

Figure 2.18: Material Data-Digimat

The Mori-Tanaka model will be used for the mean field homogenization in this study. The representative volume element is handled as a single inclusion issue subjected to a strain corresponding to the average strain in the matrix phase, and the model assumes that each inclusion behaves as if it were isolated in the matrix. This suggests that the model is appropriate for composites with lower fiber volume percentages, up to about 25% [26].

The representative volume element is divided into a number of pseudo-grains with unidirectional fiber alignment in the mean field homogenization method of an short fiber reinforced polymer. Each pseudo-grain represents a specific angular increment by expressing a unique segment in space. The number of angular increments permitted varies between 6 and 16, although Digimat suggests 12 for a reasonable balance of accuracy and computing time. In the short fiber reinforced polymer, all pseudo-grains are equally important in the case of randomly oriented fibers. When the fibers are orientated in a given direction, such as in injection moulded components, the pseudo-grains with corresponding directions have a greater impact on the homogenized response. As a result, the pseudo-grains are assigned varied weighting factors in the representative volume element, with greater weight given to pseudo-grains with the existent fiber alignment directions, as determined by the orientation tensor. This results in an representative volume element reaction that is reasonable [27].

2.2.5 FEA with Gathered Data

The analyzes were finalized with the integration of Digimat, Moldex3D and Msc Marc as stated in chapted 2.2.2.2. The force resulting from a 0,2 mm displacement on guiding element is calculated in this way (see Figure 2.16).

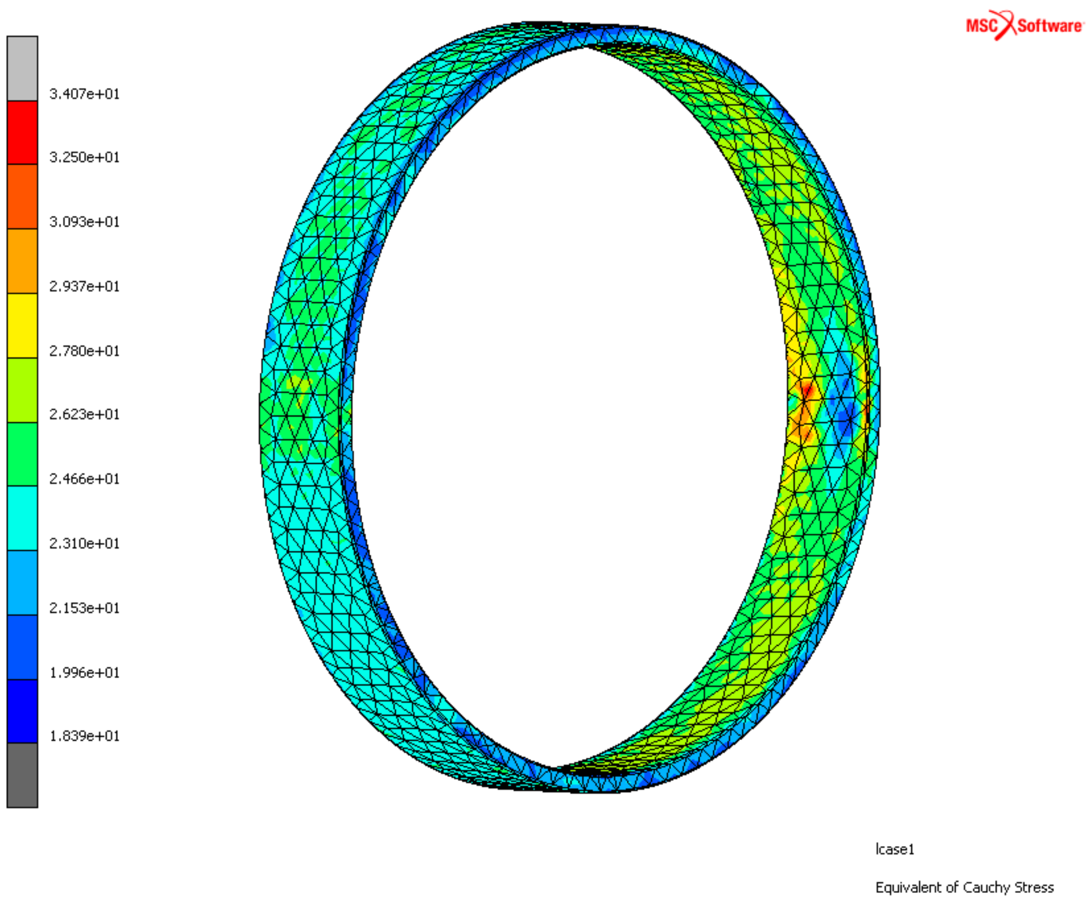


Figure 2.19: Analysis Result: Equivalent Cauchy Stress (MPa)

2.2.6 Validation with Guiding Element Test Rig

Purpose of this test rig (see Figure 2.17, Figure 2.18) is to measure load carrying capacity of guiding elements at certain displacement. In this study load carrying capacity of reference guiding element at 0,2 mm gathered from FEA compared with the load measured from test rig.

The working principle of the test; the guiding element cut in a semi-circular shape is placed in the circular test channel. Then, a test shaft is placed between the lower and upper apparatus, and the apparatus is closed so that a preload is given to the test device. With a force increase rate of 3 tons/min, the test continues until the maximum displacement of 0,2 mm is achieved and how much load can be safely carried is measured. With the test rig, the consistency of the estimated results obtained as a result of the FEA was checked.

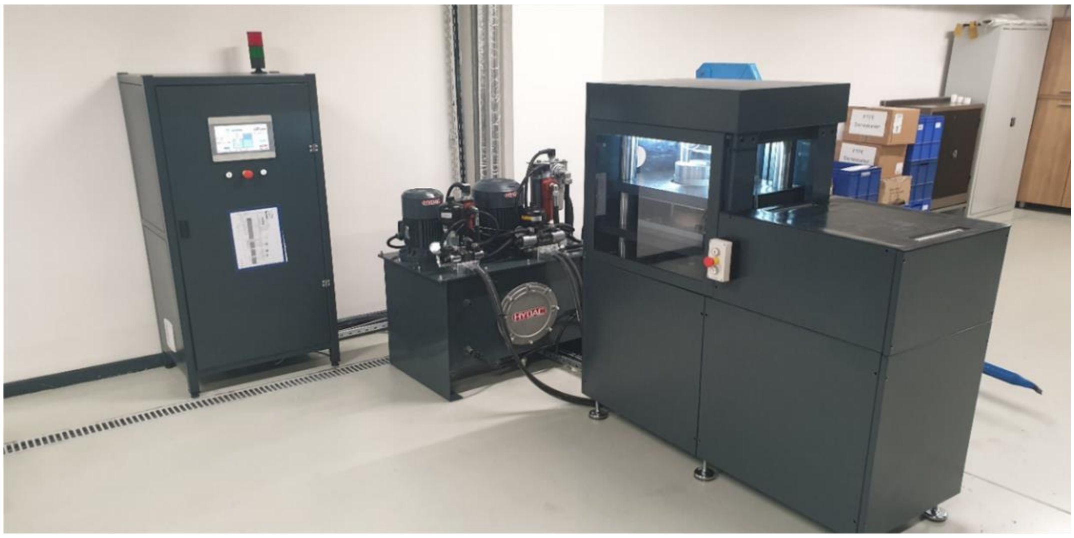


Figure 2.20: Guiding Element Test Rig

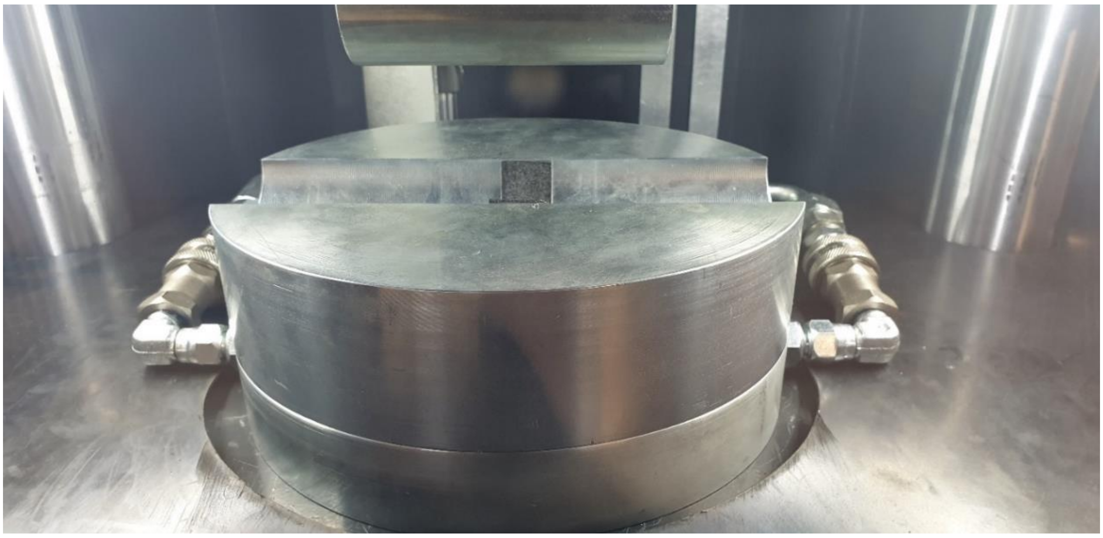


Figure 2.21: Guiding Element Test Apparatus

2.2.7 Design of Experiment

Analyzes performed with the integration of 3 different programs require long solution times. DOE studies were carried out to shorten the solution times and obtain the optimized model. As a result of regression analysis, optimized mold, runner geometry and optimum glass fiber additive ratio were determined.

Thanks to DOE, a revised mold design that will give the guiding element improved load carrying performance has been obtained and analysis flow chart repeated again for new K68 injection mold design.

2.2.8 Optimized Design Injection Machine Mold Production

Mold design was determined according to the analysis results made with the integration of Moldex3D, Digimat and MSC Marc programs. A new mold was produced according to the determined mold design.

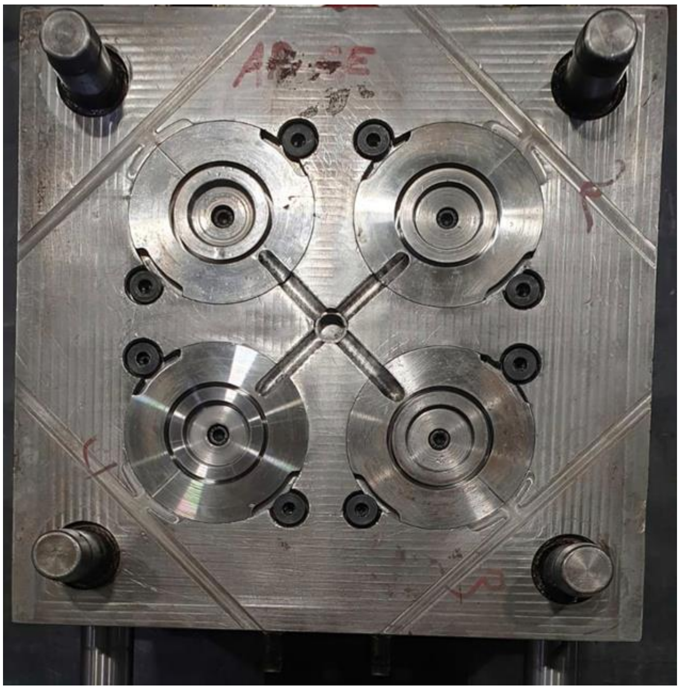


Figure 2.22: Revised mold design for improved performance

2.2.9 Injection Molding Revised Design K68 and Validation with Guiding Element Test Rig

Production was carried out on the plastic injection machine with the revised design K68 injection mold. The obtained product was tested with the Guiding Element Test Rig and the results were compared with the analysis results.

Chapter 3

Results and Discussions

3.1 FEA and Test Rig Results of Reference K68

The data obtained as a result of the tests made with the test rig on the reference K68 guiding element to be improved were compared with the data obtained as a result of the finite element analysis made in the computer environment.

The load on the product as a result of the 0,2 mm deformation of the guiding element is given in the table below.

Table 3.1: FEA and test rig test results at 0,2 mm deformation for reference K68

	Reference K68	
	Load (kgf)	Displacement (mm)
Test Rig	14399,96	0,2
FEA Result	15637,48	0,2

The force&displacement graph obtained as a result of the work done in the test device and analysis programs for reference K68 is presented below.

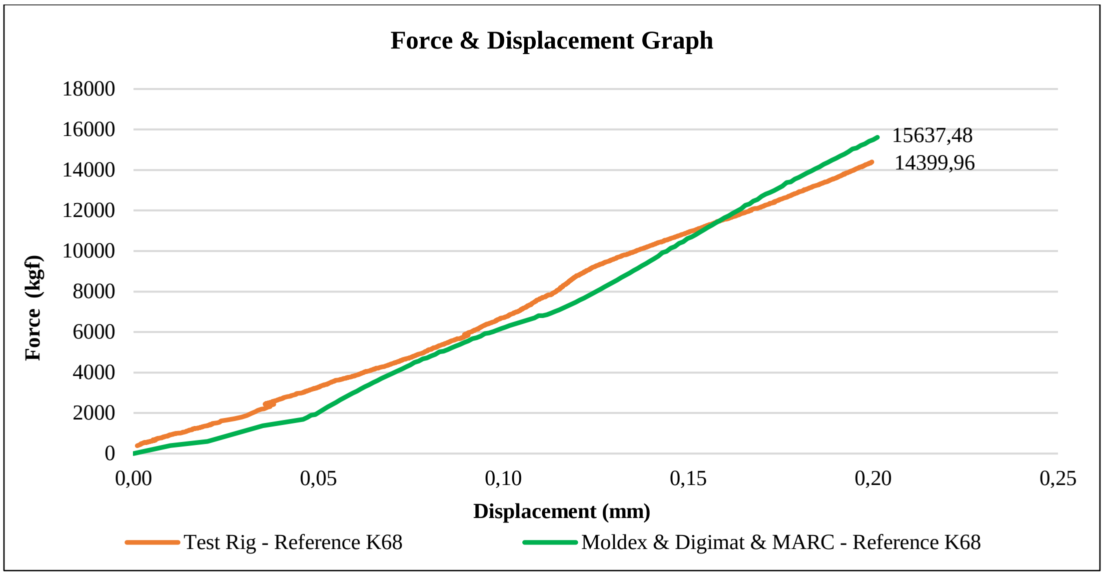


Figure 3.1: Predicted and Measured Force for Reference K68

The finite element analysis result and the result obtained from the test device show 92% consistency. This measurement success rate was found to be sufficient and optimized product mold design studies were initiated.

3.2 FEA and Test Rig Results of Revised K68

The loads on the guiding element as a result of the 0,2 mm deformation of the revised guiding element indicated in the table below.

Table 3.2: FEA and test rig test results at 0,2 mm deformation for revised K68

Revised K68		
	Load (kgf)	Displacement (mm)
Test Rig	16990,4	0,2
FEA Result	19255,86	0,2

The Force&Displacement graph obtained as a result of the work done in the test device and analysis programs for revised K68 is presented below.

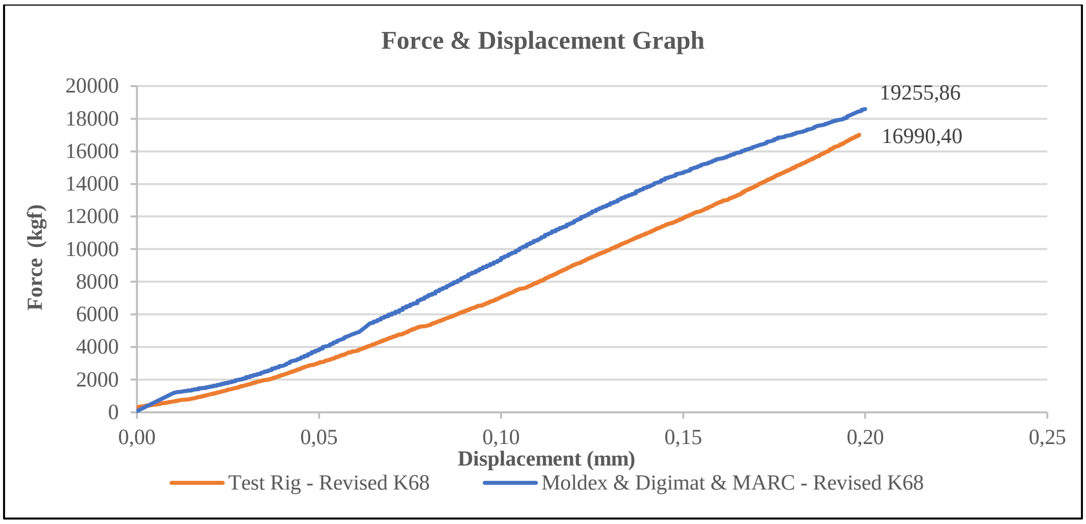


Figure 3.2: Predicted and Measured force for Revised K68

The finite element analysis result and the result obtained from the test device show 88% consistency. The mold geometry design for the revised product design specified in the table has been made with DOE. The study was concluded successfully, with the estimated data obtained as a result of FEA and the data compliance rate measured with the test device as sufficient.

3.3 K68 Mold Optimization DOE and FEA Results

Analyzes performed with the integration of 3 different programs require long solution times. Ramakrishnan and Mao found that the Taguchi optimization and ANOVA methods were found to be very useful in identifying the most important modelling process parameters for volumetric shrinkage and optimization of control parameters to achieve minimum partial shrinkage [28]. In this study ANOVA method performed to optimize mold geometry of K68 hydraulic guiding element to get maximum load carrying capacity. The data of 7 analyzes whose variables and results were listed at the beginning of the term are listed as in Table 3.3 in order to perform optimization studies with the experimental design method.

Table 3.3: Mold, runner parameters and FEA results

Software	**Moldex3D**			**Digimat**	**Marc (Results)**		
Variables/ Analysis Number	Melt Material Inlet	Runner	Gate	Fiber Ratio	Eqv. Cauchy Stress	Contact Stress	Force
Reference	10 10	4,99 mm 51,34 ° 2,5 mm	4 mm 1 mm	25% GF	88,96 MPa	179,9 MPa	1,534x10^5
1	10 7,5	5,99 mm 40 ° 3 mm	4 mm 1 mm	25% GF	87,81 MPa	180,5 MPa	1,534x10^5
2	12,5 10	5,99 mm 40 ° 3 mm	4 mm 0,5 mm	25% GF	88,32 MPa	178,6 Mpa	1,528x10^5
3	12,5 10	4,99 mm 40 ° 3 mm	4 mm 2 mm	25% GF	85,98 MPa	185,3 MPa	1,516x10^5
4	12,5 10	4,99 mm 40 ° 3 mm	4 mm 2 mm	20% GF	80,32 MPa	179,5 MPa	1,461x10^5
5	12,5 10	4,99 mm 60 ° 3 mm	4 mm 2 mm	25% GF	95,26 MPa	201,3 MPa	1,825x10^5
6	12,5 10	4,99 mm 60 ° 3 mm	4 mm 2 mm	20% GF	80,22 MPa	178,8 MPa	1,459x10^5

All these analysis data were used for optimization with the experimental design method. The data is divided into two as input and output as in Table 3.4.

Table 3.4: Inputs and Outputs

Input								Output		
Melt Material Inlet		Runner					Digimat	Eq. Cauchy	Contact Stress	Force
X1	X2	X3	X4	X5	X6	X7	X8	Y1	Y2	Y3
10	10	4,99	51,34	2,5	4	1	0,25	88,96	179,9	1,534
10	7,5	5,99	40	3	4	1	0,25	87,81	180,5	1,534
12,5	10	5,99	40	3	4	0,5	0,25	88,32	178,6	1,528
12,5	10	5,99	40	3	4	2	0,25	85,98	185,3	1,516
12,5	10	5,99	40	3	4	2	0,2	80,32	179,5	1,461
12,5	10	5,99	60	3	4	2	0,25	84,26	183,5	1,825
12,5	10	5,99	60	3	4	2	0,2	80,22	178,8	1,459

Excel program was used for Experimental Design Method. Before transferring the data to the Excel program, the simplification method was used because the units of the data changed in the analysis were not the same. Then with this method Std. Deviation, Sni Calculation and Sni value calculation were made.

Table 3.5: Experiment Design Chart

Experiment Design Chart

EXP NO	X1	X2	X3	X4	X5	X6	X7	X8	Y1	Y2	Y3
1	1	1	1	1	1	1	1	1	88,96	179,9	1,534
2	1	2	2	2	2	1	1	1	87,81	180,5	1,534
3	2	1	2	2	2	1	2	1	88,32	178,6	1,528
4	2	1	2	2	2	1	3	1	85,98	185,3	1,516
5	2	1	2	2	2	1	3	2	80,32	179,5	1,461
6	2	1	2	3	2	1	3	1	84,26	185,3	1,825
7	2	1	2	3	2	1	3	2	80,22	178,8	1,459

Experiment Design Chart

EXP NO	Avarage	Std. Deviation	Sni Calculation			Sni
1	68,70575	84,49446	4720,48	7139,313	0,661195	-1,7967
2	68,56925	84,66775	4701,742	7168,628	0,655878	-1,83177
3	68,16775	83,92692	4646,842	7043,727	0,659714	-1,80645
4	69,09725	86,88333	4774,43	7548,713	0,632483	-1,98951
5	66,322	83,85565	4398,608	7031,771	0,625533	-2,03749
6	68,76625	86,67726	4728,797	7512,948	0,62942	-2,0106
7	66,09125	83,56575	4368,053	6983,235	0,625506	-2,03769

According to these calculations, critical importance ranking was made for eight inputs with seven experimental data.

Table 3.6: Criteria Importance Ranking Chart

Criteria Importance Ranking Chart								
	X1	X2	X3	X4	X5	X6	X7	X8
Level 1	-1,20949	-1,94641	-1,7967	-1,7967	-1,7967	-1,93003	-1,81424	-1,88701
Level 2	-1,97635	-1,83177	-1,95225	-1,91631	-1,91	-1,75175	-1,80645	-1,89999
Level 3	0	0	0	-2,02414	0	0	-2,01882	0
DELTA	1,976348	1,946407	1,952252	0,227439	1,910001	1,930031	0,212377	1,899987
Ranking	1	3	2	7	5	4	8	6

According to this study, it was determined that the variable that most affected the analysis results was X1 (melt material inlet). Regression analysis was performed by adding the inputs and outputs to the table. Regression analysis is a statistical tool used to model the relationship between a dependent variable and one or more independent variables. Regression analysis describes how the typical value of the dependent variables changes when one of the independent variables increases or decreases, while the other independent variables are kept constant [29]. The confidence ratio (R^2) to be achieved in the regression analysis is between 80% and 95%. Since the rate was below the confidence index in the regression analysis performed with the first data, nine more analyzes were performed according to the critical importance ranking chart in order to increase the confidence rate.

Table 3.7: Multiple Regression Analysis Table

SUMMARY OUTPUT

Regression Statistics	
Multiple R	0,937432165
R Square	0,878779065
Adjusted R Square	0,740240853
Standart Error	0,015385905
Observations	16

ANOVA

	df	*SS*	*MS*	*F*	*Significance F*
Regression	8	0,012012855	0,001501607	6,343225115	0,01237585
Residual	7	0,001657083	0,000236726		
Total	15	0,013669938			

	Coefficients	*Standard Error*	*t Stat*	*P-value*	*Lower %95*	*Upper %95*	*Lower 95,0%*	*Upper 95,0%*
Intercept	1,494025053	0,114080094	13,09628171	3,52914E-06	1,224268495	1,76378161	1,224268495	1,76378161
X1	0,005167341	0,003843643	1,344386304	0,220756453	-0,00392143	0,014256112	-0,00392143	0,014256112
X2	-0,00771007	0,005297729	-1,4553539	0,188903927	-0,02023721	0,004817068	-0,02023721	0,004817068
X3	-0,06755037	0,037149915	-1,81831832	0,111843775	-0,15539596	0,020295219	-0,15539596	0,020295219
X4	-0,0014213	0,000666869	-2,13130315	0,070534338	-0,00299819	0,000155595	-0,00299819	0,000155595
X5	0,079489447	0,054805184	1,450400129	0,190233076	-0,05010422	0,209083115	-0,05010422	0,209083115
X6	0,019007691	0,0108088	1,758538556	0,122062451	-0,00655106	0,044566441	-0,00655106	0,044566441
X7	-0,01277633	0,008302616	-1,53883182	0,167736379	-0,0324089	0,006856238	-0,0324089	0,006856238
X8	0,812545379	0,229865439	3,534874064	0,009535221	0,268999987	1,356090771	0,268999987	1,356090771

After listing all the data, the equation obtained with 87% confidence rate is as follows.

Y3=1,4940250525207+0,005167340849*x1-0,00771006998345*x2-0,06755037129947*x3-0,0014212993156*x4+0,07948944653*x5+0,01900769088483*x6-0,01277633035749*x7 +0,812545378971442*x8

Y3 formula was obtained by Analysis-of-Variance (ANOVA) approach. Thus, a formula describing the effect of die geometry parameters on the load bearing capability of the hydraulic bearing element was obtained.

With the input data changed according to this equation, the model closest to the target output was determined and the analysis was repeated. Considering the different runner sizes and fiber ratios, according to the analysis results of the models, the model number 18, in which we improved the load carrying ability by 20% (see table 3.8).

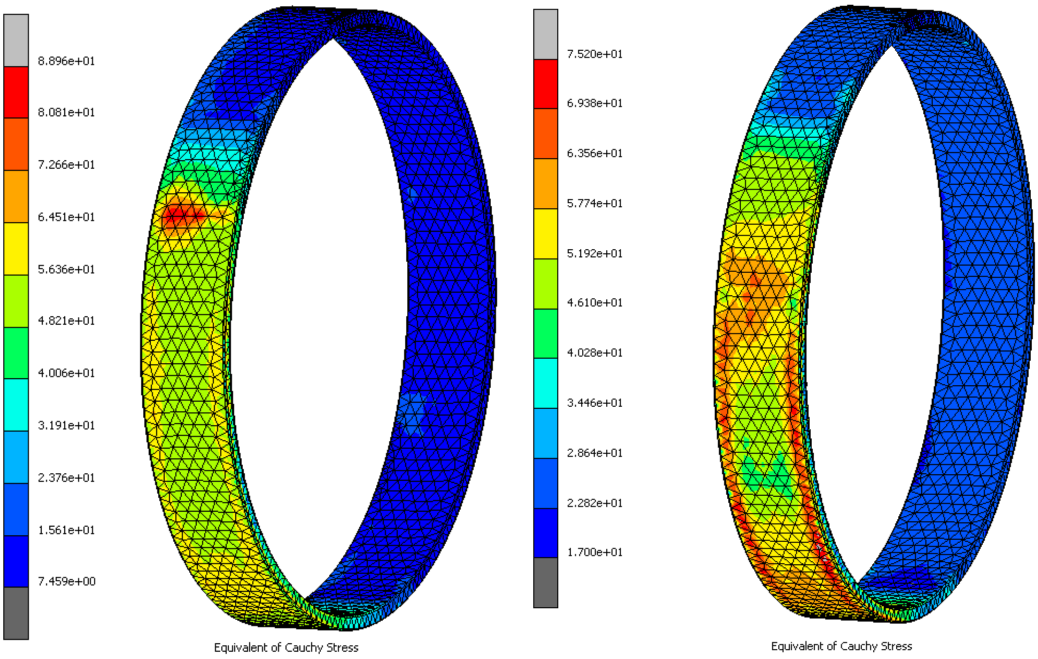
Table 3.8: All Input and Output Data Analysis

NO	X1	X2	X3	X4	X5	X6	X7	X8	Y1	Y2	Y3
1	10	10	4,99	51,34	2,5	4	1	0,25	89	179,9	1,53
2	10	7,5	5,99	40	3	4	1	0,25	87,8	180,5	1,53
3	12,5	10	5,99	40	3	4	0,5	0,25	88,3	178,6	1,53
4	12,5	10	5,99	40	3	4	2	0,25	86	185,3	1,52
5	12,5	10	5,99	40	3	4	2	0,2	80,3	179,5	1,46
6	12,5	10	5,99	60	3	4	2	0,25	80,2	178,8	1,83
7	12,5	10	5,99	60	3	4	2	0,2	80,2	178,8	1,46
8	10	12,5	3,99	60	2	3	1	0,2	87,4	179	1,46
9	5	10	5	60	3	4	2	0,2	90,2	175,8	1,46
10	5	10	5	60	3	4	2	0,25	85,2	183,4	1,52
11	7,5	10	4,99	40	2,5	4	2	0,25	88,9	188,2	1,52
12	7,5	10	8	0	4	6	3	0,25	88,1	179,2	1,52
13	5	10	6	0	3	3	1,5	0,25	98,4	185,2	1,53
14	5	10	6	0	3	4	2	0,25	85,2	184,9	1,52
15	10	10	8	0	4	4	1	0,25	86,2	186	1,52
16	3	10	6	0	3	4	1	0,25	86,2	182,6	1,52
17	2	8	6	0	3	4	1	0,25	84,6	182,6	1,52
18	10	10	3	0	3	4	1	0,25	75,2	213,5	1,89

When the input and output data are examined, when X3 and X5 values are 3 mm and X1, X2 data are 10 mm, an increase of 20% in load carrying capability is observed. With the data obtained from this study, a hydraulic guiding element injection mold with mold dimensions stated in No 18 was produced. The name of the product to be manufactured from this new mold will be indicated as Revised K68/Revised Model in the continuation of the study.

3.3.1 FEA Results of Reference K68 and Revised K68

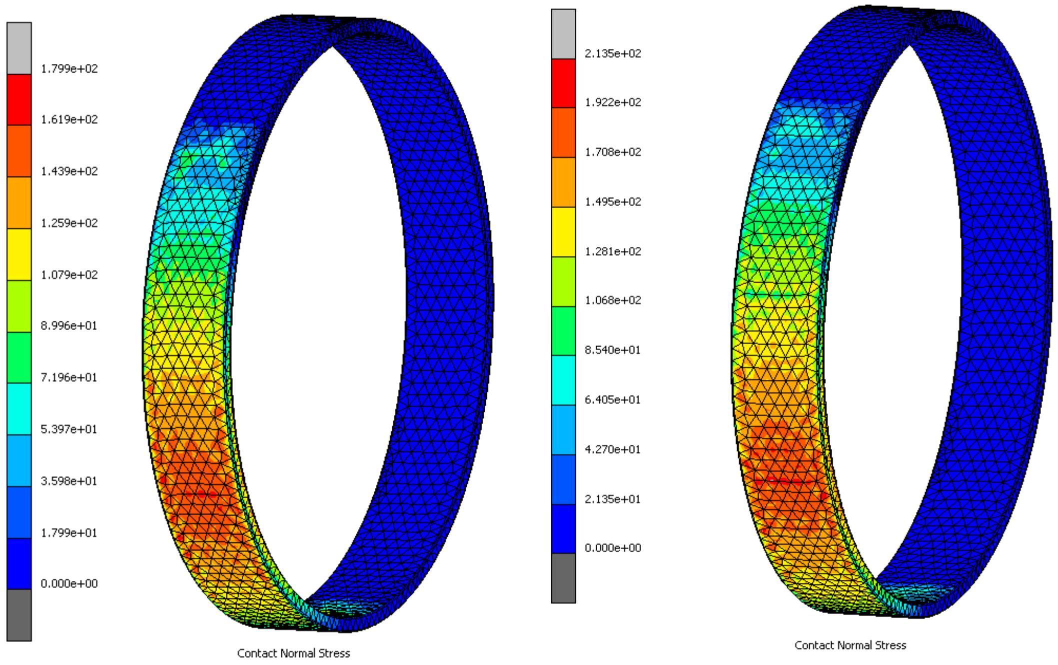
The analyzes were finalized with the integration of different programs and carried out in MSC Marc environment, and the results of the current product and the optimized product are shared as follows.



(a) (b)

Figure 3.3: Analysis Result- Equivalent Cauchy Stress (MPa), (a) Reference Model, (b) Revised Model

Equivalent cauchy stress expresses the internal stress that occurs in the material as a result of deformation. Internal stress in the revised K68 at 0,2 mm deformation is reduced by 15 percent compared to the reference K68.



(a) (b)

Figure 3.4: Analysis Result- Contact Normal Stress (MPa), (a) Reference Model, (b) Revised Model

Contact normal stress is used to examine the stress on the guiding element surface. Contact normal stress in the revised K68 at 0,2 mm deformation is reduced by 16 percent compared to the reference K68. Maximum Equivalent Cauchy Stress and Contact Stress values of K68 guiding element are given in Table 3.9.

Table 3.9: Analysis Results - Stress

	Reference Model	Revised Model
Equivalent Cauchy Stress	88,96 MPa	75,20 MPa
Contact Normal Stress	179,9 MPa	213,5 MPa

It is critical that the products be exposed to less internal stress and more force despite contact stress in the analysis results performed for the bearing element with superior load carrying capability. For this reason, multiple regression analysis was performed to increase the force value by 20% in the analyzes and the values to be increased were determined from the obtained equation. The analysis of the improved model as a result of the determined values and the analysis result of the existing product are given in Table 3.10.

Table 3.10: Analysis Results - Force

	Reference Model	Revised Model
Force (N)	1,534 x 10^5	1,889 x 10^5

3.4 Discussion

Within the scope of this study, glass fiber reinforced POM material was efficiently modeled and improved K68 hydraulic guiding element was obtained by using multilevel approach for short fiber reinforced composite materials with Moldex3D, Digimat and Marc programs as simulation software.

Data used in finite element analysis software were provided by using Moldex3D and Digimat software for the analysis of 25% glass fiber reinforced POM composite material similar to Anna K, Slawomir P and Stanislaw K's studies [16]. Anisotropic behaviour of short fiber reinforced POM polymer successfully modeled by importing fiber orientation data from Moldex3D and applying Mori-Tanaka mean field homogenization method in Digimat to elastic-plastic material model as previous study carried out by Jand AL, Miranda L and Kurkin et al. [27, 12, 30]. With Moldex3D, Digimat and Marc software, the effects of parameters such as mold geometry, sprue size, runner dimension on the load carrying capability of the manufactured product were analyzed in virtual environments and used for the purposes of prevention and optimization of possible problems, and the results were verified through the test device. As a result of this comparison, 92% consistency was observed on the load carrying capacity of the reference guiding element.

With DOE and regression analysis studies, the necessary data for the revised K68 hydraulic guiding element injection mold were obtained and analyzes were carried out in simulation software. According to the results of FEA, a revised hydraulic guiding element with a load carrying capacity of 21% and 18% higher than the test equipment test results was obtained. When the test results were compared with the finite element analysis results for revised guiding element, 88% consistency was observed for the developed product. As stated in other articles and studies, when the glass fiber ratio in the POM material is increased from 20% to 25%, an increase in the load carrying ability of the product has been observed [1, 2, 4-7].

Chapter 4

Conclusions

In this study, the load carrying capacity of hydraulic short fiber reinforced POM hydraulic guiding elements, which will make the greatest contribution to the literature, was calculated and optimized by finite element analysis and validated with tests performed on the test device. In this context, MSc Marc finite element software was used to perform non-linear analysis, Moldex3D to simulate plastic injection simulation and fiber orientation, and Digimat programs to apply anisotropic micromaterial modeling to macromaterial model isotropically. The application of the analysis processes in coupled specific to hydraulic guiding elements is another important contribution to the literature.

Thanks to the studies carried out, the increase in load carrying capability of hydraulic short fiber reinforced POM guiding element was deemed sufficient and it is planned to work with carbon fiber reinforced POM material in future studies.

With this study, the strength and load carrying capacity of the guiding elements have been increased. The goal of prolonging the service life of the sealing elements, hydraulic cylinders, and therefore the complete machine, which has a hydraulic cylinder, has been achieved with the guiding element with superior load-carrying capability.

References

[1] Lingesh, BV, Ravikumar BN, Rudresh BM. Mechanical Characterization of Hybrid Thermoplastic Composites of Short Carbon Fibers and PA66/PP. Indian J. Adv. Chem. Sci. 2016; 4, 425–434.

[2] Yan-fang Y. A geometric approach for polymer applications and processing. 2010 2nd International Conference on Signal Processing Systems (ICSPS) 2010; 1.

[3] Singh JSK, Ching YC, Liu DS, Ching KY, Razali S, Gan SN. Effects of PTFE Micro-Particles on the Fiber-Matrix Interface of Polyoxymethylene/Glass fiber/Polytetrafluoroethylene composites. Materials 2018; 11, 2164.

[4] Singh JSK, Ching YC, Abdullah LC, Ching KY, Razali S, Gan SN. Optimization of mechanical properties for poly(oxymethylene)/glass fiber/polytetrafluoroethylene composites using response surface methodology. Polymers 2018; 10, 338

[5] He M, Zhang D, Guo J, Qin S, Ming X. Mechanical, thermal and dynamic mechanical properties of long glass fiber-reinforced thermoplastic polyurethane/poly(oxymethylene) composites. Polym. Compos. 2014; 35, 2067–2073.

[6] Kumar, S, Panneerselvam K, Optimalization of friction and wear of Nylon 6 and glass fiber reinforced (GFR) Nylon 6 composites against 30 wt. % GFR Nylon 6 Disc. J. Adv. Res. Mat. Sci. 2016, 19, 14–32.

[7] Kawaguchi K, Masuda E, Tajima Y. Tensile behawior of glass-fiber-filled Poly(oxymethylene): Influence of the functional groups of polymer matrices. J. Appl. Polym. Sci. 2008; 107, 667–673.

[8] Li ZH. Addition of CF on tribological properties of POM composite. Mater. Technol. 2012; 27, 230–232.

[9] Lv M, Zheng F, Wang Q, Wang T, Liang Y. Friction and wear behaviors of carbon and aramid fibers reinforced polyimide composites in simulated space environment. Tribol. Int. 2015; 92, 246–254.

[10] Luo W, Ding Q, Li Y, Zhou S, Zou H, Liang M. Effect of shape morphology on mechanical, rheological and tribological properties of polyoxymethylene/aramid composites. Polym. Sci. A 2015, 57, 209–220

[11] Tian YQ, Huo JL. The mechanical and tribological properties of carbon fiber reinforced POM composites. Appl. Mech. Mater. 2012; 182, 135–138.

[12] Kurkin EI, Sadykova VO. Application of short fiber reinforced composite materials multilevel model for design of ultra-light aerospace structures. Procedia Engineering 185, 2017; 182-189

[13] Kastaş Sızdırmazlık Teknolojileri [Internet], Retrieved May 9, 2022 from https://www.kastas.com.tr/document/download/4/31/Kastas_Hidrolik_Katalog_TR_10122021164018416917341ieiD.pdf

[14] Semiz S. New Grade of Thermoplastic Polyurethane with High Thermal Conductivity and Low Coefficient of Friction (master's thesis). İzmir Katip Çelebi University; 2017

[15] Lüftl S, Visakh PM, Chandran S. Polyoxymethylene Handbook: Structure, Properties, Applications and their Nanocomposites; Scrivener Publishing LLC: Hoboken, NJ, USA, 2014

[16] Thakre P, Chauhan AS, Satyanarayana A, Raj Kumar E, Pradyumna, R. Estimation of Shrinkage & Distortion in WaxInjection using Moldex3D Simulation. Materials Today: Proceedings, 2018; 5(9), 19410–19417. doi:10.1016/j.matpr.2018.06.301

[17] Anna K, Slawomir P, Stanislaw K. Basalt/Glass Fiber Polypropylene Hybrid Composites: Mechanical Properties at Different Temperatures and under Cyclic Loading and Micromechanical Modelling. Materials 2021; 14, 5574. https://doi.org/10.3390/ma14195574

[18] Bias, Digimat [Internet], Retrieved May 9, 2022 from https://bias.com.tr/urun/yazilim/yapisal/digimat

[19] Moldex3D [Internet], Retrieved May 9, 2022 from https://www.moldex3d.it/upload/content/702014728102711775.pdf

[20] Msc Software Marc [Internet], Retrieved May 9, 2022 from https://www.mscsoftware.com/product/marc

[21] Hwaseop L, Kwangyeol R, Youngju C. A framework of a smart injection molding system based on real-time data Procedia Manufacturing 11. 2017; 1004 – 1011. doi: 10.1016/j.promfg.2017.07.206

[22] Peter T. Modelling and monitoring in injection molding. Technical University of Denmark ;2001

[23] Moldex3D [Internet], Comprehensive Mold Opening Simulation for Accurate Cooling and Warpage Analysis Retrieved May 9,2022 from https://www.moldex3d.com/blog/tips-and-tricks/comprehensive-mold-opening-simulation-for-accurate-cooling-and-warpage-analysis/

[24] Injectionmoulding [Internet], Retrieved May 9, 2022 from http://www.injectionmoulding.org/how-to-design-injection-mould-gating-system.html

[25] Msc Marc, Marc Volume B – Element Library

[26] e-Xstream engineering: Digimat documentation 5.1.2. Mont-Saint-Guibert; 2014

[27] Jan AL, Miranda L. Fatigue Analysis of Anisotropic Short Fibre Reinforced Polymers - by Use of Digimat and nCode DesignLife; 2015

[28] Ramakrishnan R, Mao DK. Minimization of Shrinkage in Injection Molding Process of Acetal Polymer Gear Using Taguchi DOE Optimization and ANOVA Method. IJMT 2016; 4, 9.

[29] Tseng M, Fu C, Lu L, Shieh J. Emotional and behavioral problems in preschool children with autism: Relationship with sensory processing dysfunction. Research in Autism Spectrum Disorders. 2011; 5(4), 1441–1450.

[30] Andreas W, Kurt H. Influence of Fiber Orientation and Multiaxiality on the Fatigue Strength of Unnotched Specimens – Lifetime Estimation. 2015

Curriculum Vitae

Name Surname : Cem Tanyeri

Education:

2019–2022 İzmir Kâtip Çelebi University, Dept. of Material Science and Eng. Msc

2010–2012 Ege University, Master of Business Administration.

2004–2009 Süleyman Demirel University, Dept. of Mechnical Eng. Bachelor's Degree

2004 Konak Anatolian Highschool

Work Experience:

2019 – 2022 Senior Executive Simulation Technologies-Kastaş Sızdırmazlık Teknolojileri A.Ş

2017 – 2019 Research and Development Exetuive-Kastaş Sızdırmazlık Teknolojileri A.Ş

2013 – 2017 Research and Development Engineer-Kastaş Sızdırmazlık Teknolojileri A.Ş

2012 – 2013 Purchasing Engineer-Emas Makina A.Ş

2011 – 2012 Production/Project Engineer-Klas Isıtma Soğutma Ltd. Şti.