The people's Democratic Republic of Algeria
الجمهورية الجزائرية الدمقراطية الشعبية
Ministry of Higher Education and Science Research
وزارة التعليم العالي والبحث العلمي

UNIVERSITY M'HAMED BOUGARA
BOUMERDES

FACULTY OF TECHNOLOGY

DEPARTEMENT OF MECHANICAL
ENGINEERING

Study, design and simulation of an injection mold for a wheelbarrow's wheel spacer

A Thesis Presented
By

Marwa DAKICHE

In partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

Major in

MECHANICEL CONSTRUCTION

Supervisor
Abdelkrim MERAH

Co-supervisor
Smail BENIDIR

Jury

Mr	Haroun RAGUEB	Presedent
Mr	Belkacem MANSER	Examiner

Academic Year: 2021/2022

Abstract

In recent years, domestic companies are in a competitive situation, this leads them to produce goods and services at the lowest possible cost and in less time. As such, every business is looking for more realistic and appropriate evaluation methods.

The company SOFICLEF decided to enroll in this process. The service and development department of the company took the initiative to manufacture certain parts instead of purchasing them.

This task is considered a contribution in the sense that we will study and design an injection mold for a wheelbarrow's wheel spacer, a part which was purchased before and was carried out by the plastic injection system.

Our main tasks will be to focus firstly on the theoretical part where we will provide the important characteristics of plastics, more precisely on PP (polypropylene), which is the material used for our part production, and the various processes involved for the implementation of parts from these materials. On the practical side we will run a plastic injection simulation using SOLIDWORKS Plastics, design a mold with the software SolidWorks, check its resistance to various conditions both theoretically and by running a resistance analysis using the software ABAQUS and finally we will preform a computer-aided manufacturing process with CAMWORKS.

Key Words: CAD, CAM, Injection molding, Mold, Polypropylene, SOLDWORKS, CamWorks Wheelbarrow, Wheel, Injection, ABAQUS.

Résumé

Ces dernières années, les entreprises nationales sont en situation de concurrence, ce qui les amène à produire des biens et des services au plus bas coût possible et en moins de temps. De ce fait, chaque entreprise est à la recherche de méthodes d'évaluation plus réalistes et plus appropriées.

La société SOFICLEF a décidé de s'inscrire dans cette démarche. Le département service et développement de l'entreprise a pris l'initiative de fabriquer certaines pièces au lieu de les acheter.

Cette tâche est considérée comme une contribution dans le sens où nous allons étudier et concevoir un moule d'injection pour une entretoise de roue de brouette, une pièce qui a été achetée auparavant et qui a été réalisée par le système d'injection plastique.

Nos principales tâches seront de se concentrer tout d'abord sur la partie théorique où nous fournirons les caractéristiques importantes des plastiques, plus précisément sur le PP (polypropylène), qui est le matériau utilisé pour la production de notre pièce, et les différents processus impliqués pour la mise en œuvre de pièces à partir de ces matériaux. Sur le plan pratique, nous effectuerons une simulation d'injection plastique à l'aide de SOLIDWORKS Plastics, nous concevrons un moule à l'aide du logiciel SolidWorks, nous vérifierons sa résistance à diverses conditions, tant sur le plan théorique qu'en effectuant une analyse de résistance à l'aide du logiciel ABAQUS et, enfin, nous réaliserons un processus de fabrication assistée par ordinateur avec CAMWORKS.

Mots clés: CAO, FAO, Moulage par injection, Moule, Polypropylène, SOLIDWORKS, CamWorks Brouette, Roue, Injection, ABAQUS.

ملخص

في السنوات الأخيرة، كانت الشركات المحلية في وضع تنافسي، مما أدى بها إلى إنتاج السلع والخدمات بأقل تكلفة ممكنة وفي أقصر وقت ممكن. نتيجة لذلك، تبحث كل شركة عن طرق تقييم أكثر واقعية وملاءمة.

قررت SOFICLEF اتباع هذا النهج. بادر قسم الخدمة والتطوير بالشركة إلى تصنيع أجزاء معينة بدلاً من شرائها.

تعتبر هذه المهمة مساهمة بمعنى أننا سوف ندرس ونصمم قالب حقن لمباعد عجلة عربة اليد، قطعة تم شراؤها من قبل وصُنعت بواسطة نظام حقن البلاستيك.

ستكون مهامنا الرئيسية هي التركيز أولاً على الجزء النظري حيث سنوفر الخصائص المهمة للبلاستيك، وتحديداً على PP (البولي بروبلين)، وهي المادة المستخدمة لإنتاج الجزء الخاص بنا، والعمليات المختلفة التي ينطوي عليها التنفيذ. من أجزاء من هذه المواد. على المستوى العملي، سنجري محاكاة حقن البلاستيك باستخدام Plastics SOLIDWORKS، وسوف نصمم قالبًا باستخدام برنامج SolidWorks، وسوف نتحقق من مقاومته للظروف المختلفة، من الناحية النظرية ومن خلال إجراء تحليل المقاومة باستخدام برنامج ABAQUS، وأخيراً، سنقوم بتنفيذ عملية تصنيع بمساعدة الكمبيوتر باستخدام CAMWORKS.

الكلمات الرئيسية: CAD، CAM، حقن صب، قالب، بولي بروبيلين، SOLIDWORKS، CamWorks عربة يدوية، عجلة، حقن، ABAQUS.

The only reason for time is so that everything doesn't happen at once.

— Albert Einstein

acknowledgments

> The heart has its reasons of which
> reason knows nothing.
>
> – *Blaise Pascal*

I would first like to thank my thesis advisor Mr Abdelkrim MERAH of the Faculty of technology at the University of Boumerdes. He consistently allowed this thesis to be my own work, but steered me in the right the direction whenever he thought I needed it.

I would like to express my deep gratitude to my co-supervisor, Mr. Smail BENIDIR, for his relevant observations and for his help and guidance throughout this work. The door to Mr BENIDIR's office was always open whenever I run into a trouble spot or had a question about my research or writing.

Most importantly, I must express my very profound gratitude to my parents and siblings for providing me with unfailing support and continuous encouragement throughout my years of study and through the process of researching and writing this thesis. This accomplishment would not have been possible without them. Thank you.

"Last but not least, I wanna thank me, I wanna thank me for believing in me, I wanna thank me for doing all this hard work, I wanna thank me for having no days off and I wanna thank me for never quitting."

Contents

List of Figures

List of Tables

List of symbols

PP : Polypropylene

PE: Polyethylene

PS: Polystyrene

PVC: Polyvinyl chloride

ABS: Acrylonitrile Butadiene Styrene

F: Force

P: Pressure

K: Factor of safety

C: Mass of recovered material per hour

e: average thickness

D: thermal diffusivity

Ti : injection temperature

Te: ejection temperature

Tm: mold temperature

λ : Thermal conductivity

ρ : Mass density

c : Specific heat

M: mass

N: number of cooling cycles per hour

Hi : specific heat capacity at injection

He: specific heat capacity at ejection

Ch : hourly liquid consumption

Qh : extracted heat quantity

Cf : heat capacity of the coolant

Te: water entry temperature

Ts: water exit temperature

Lc: length of the cooling channel

h: heat transfer coefficient.

d: diameter

Tc: channel wall temperature

Tf : fluid temperature at the center of the channel

Vf : fluid speed

Re: reynold’s number

ν: kinematic viscosity

Tm: average fluid temperature along cooling channel

Tc: channel wall temperature

Pr: prandtl number

Y: water conductivity

Rpc: Practical resistance to compression

Re: Material’s Elastic resistance to compression

s: Safety coefficient

Rpg: Practical resistance to shear stress

Reg: Material’s elastic resistance to shear stress

n: number of sections under shear stress.

INTRODUCTION

The design and production of plastic parts is currently of interest to a large number of industrial companies in this sector, given the diversity of uses in the electronic, electrical, packaging and household appliance sectors, etc. This interest stems from the many technical and economic advantages of plastics and their characteristics, which meet the requirements for the use of products made from these materials.

Everyone knows that polymers have been the most brilliant success of industrial chemistry over the last twenty years: the production of polymers has now reached 25 million tonnes. This success is not, of course, due to chance; for while some polymers, such as thermoplastics, which have indisputably been one of the major research themes in macromolecular chemistry, have 'unique' properties, plastics have often won out over previous materials thanks to a satisfactory set of properties, combined with greater ease of shaping. Each plastic part is only produced after a complete study of the different tasks, the main ones being design and manufacturing.

The design office of the company SOFICLEF entrusted us, within the framework of the realization of our end-of-studies project, with the design of a plastic injection mold for a wheelbarrow's wheel spacer by the plastic injection process using the material PP (polypropylene).
This study is divided into five main chapters:

In the first chapter, the general aspect of plastics is presented. Then, in the second chapter, the processing techniques of these materials will be presented.
The design of the study mold and the corresponding verification calculations will be dealt with in the third and fourth chapters..
The fifth and last chapter consists of three different analyses and simulations of injection, resistance and manufacturing.
The technical file containing the technical drawings of the different parts as well as the assembly drawings constitutes the last part of this work.

ABOUT THE COMPANY

Activity

SOFICLEF is an industrial and commercial company, our major asset is the total mastery of the operations of large-scale distribution:

1-Production and manufacturing: lock body, door handle, license plate and wheelbarrow.
2- Mounting and assembly: door lock.
3- Marketing: power tools and equipment, hand tools and accessories, house doors and key blanks.[15]

Figure 1: SOFICLEF Company.

Logistics base

SOFICLEF is at the heart of the development of logistics to meet the requirements of organizational modernization SOFICLEF has one of the largest distribution networks in Algeria.
Also, SOFICLEF has a storage and warehousing space of 15.000m^2, with a racking system with high height pallets, in order to ensure the flow of the goods in full safety.

Delivery fleet

A fleet of vehicles of all types as well as a large distribution network judiciously distributed throughout the national territory allows it to guarantee the delivery of the goods in the best time thanks to a preliminary planning for the optimization of the loads.

Authorized economic operator

The mastery of customs management procedures and compliance with all aspects has led to the granting of a specialized customs line for our customs transactions.
The testimony and the recognition of the Algerian state was translated by the attribution of the status of approved economic operator.
Thanks to its installations and its organization, SOFICLEF was able to obtain the authorization of its private bonded warehouse.

ISO 9001:2015 certification

Soficlef Sarl has illustrated its willingness to maintain quality among its primary concerns by joining a continuous evaluation process of the organization of all its structures. On September 18, 2017, VINÇOTTE international Algeria SPA awarded, after a thorough evaluation, the ISO 9001:2015 quality standard for all of Soficlef's activities.

History of the company

1994 It all started on June 21, 1994, in a 7 m^2 room located at 39 Rue Ahmed Boumaazouza El Madania 'Ex Salembier' Algiers. Where the first activity was the the making of key blanks.

1995
Beginning of the distribution of the key drafts on the big Algiers.

1996
Extension of the distribution of key blanks on the national territory.

1997
1ère importation des ébauches de clé de l'Espagne de la marque JMA.

1998 (Creation of Sarl Soficlef)
1ère génération du logo SOFICLEF.
Création de la Sarl SOFICLEF en date du 18 février 1998.

1999
Exclusive distributor of license plates of the French brand FAAB.

2000
Implementation of the customer care service.

2001
Exclusive distributor of KALE KILIT Turkish locks.

2002
Exclusive distributor of AHRAM Egyptian locks under the IZO brand.

2003
Setting up the after-sales service.

2004
SOFICLEF transfers its headquarters to its own building of 900m^2 located in Tixeraine (Algiers).

2005
Introduction of padlocks and hinges in the hardware product line.

2006
Installation of a local mounting unit and assembly of the handle.

2007
Acquisition of a license plate and handle production line.

2008 (10th anniversary)
2nd generation of the logo.

SOFICLEF moves to a new site of 5000m^2 located in Baba Ali (Algiers).

2009
Introduction of the range of power tools under the SOFICLEF brand and the opening of a showroom of 1800m^2 in Chéraga (Algiers) dedicated to the sale of doors.

2010
SOFICLEF moves to a new site of 35000m^2 located in Si Mustapha (Boumerdes) and obtaining an authorization for the operation of a bonded warehouse at the site.

2011
Increase of the production capacity of the handle and the espagnolette.

2012
1st export of the license plate to Morocco.

2013
Installation of an assembly line and assembly of the lock.

2014
Reorganization of all company structures.
SOFICLEF reaches 290 employees.

2016
1st export of license plate to France.

2017
ISO 9001:2015 management system certification.

2018 (20th anniversary)
Official distributor of the American brands DeWalt, Stanley and Black and Decker.

2019
3rd generation of the logo.
Beginning of the local production of the wheelbarrow.

2022
Establishment of exclusive distributors in the regions of the country.

INTRODUCTION TO PLASTICS

0.1 Introduction

Plastics have become an important part of our daily lives. The clothes we wear, the toothbrushes we use to brush our teeth, the storage containers we use to transport and heat our food, the cars we drive, the electronic devices we use to communicate, the credit cards we use to make payments, and a variety of other products have all become indispensable in our daily lives. These items are usually made of various types of plastic materials.

The words plastic and plastics come from the Ancient Greek words plastikos, which means “suited for molding,” and from the Latin word plasticus, which means “of molding.” Both terms refer to the process of shaping or molding shapes using heat. As a result, practically all plastic production processes begin with the molding of plastics using heat and pressure. Heat is used to soften plastic materials, which is typical of various manufacturing and technological operations. After that, the softened plastic is shaped to match the application. After achieving the desired shape, the material is cooled to allow it to harden and preserve the desired shape.

Because of their reduced cost and lighter weight, plastic products are widely accepted. The advantages of adapting to plastics include ease of processing, resilience, recyclability, and adaptability. Plastics growth and acceptance throughout the previous century has improved people’s comfort and standard of living. Today’s clean water, food delivery, healthcare, clothes, cars, aircraft, agriculture, and consumer goods would be impossible without plastic items.

After WWII, plastics began to be used on a big scale, and they swiftly revolutionized daily life by replacing expensive and seldom available metals counter parts. Millions of plastic goods with distinct functions and benefits are utilized on a daily basis. Plastic’s end uses range from low-cost commodity merchandise to high-cost complicated goods used in aerospace, vehicles, medicinal medication delivery systems, prosthetics, grafts, and other applications [16].

0.2 Concept of Macromolecules and Polymers

Every particle in the cosmos is made up of atoms, which are the fundamental building components. A molecule is made up of two or more atoms. A molecule of water, for example, is made up of two hydrogen atoms and one oxygen atom. Giant molecules, also known as macromolecules or polymers, are generated when a vast number of these molecules are joined. The term polymer is defined as a chemical substance made up of small molecules (monomers) that are arranged in a simple repeating structure to form a big molecule or a chain.

To make a polymer, hundreds of thousands of monomers are chemically bound together by covalent bonds. The bonding of monomers to form a polymer is depicted in Figure 1.1. Monomers are represented by circles in this diagram, while bonds are represented by a straight line. The monomers are joined together to form a polymer during the bonding step.

Most polymers are organic materials consisting of carbon, hydrogen, oxygen, nitrogen, and sulfur [16].

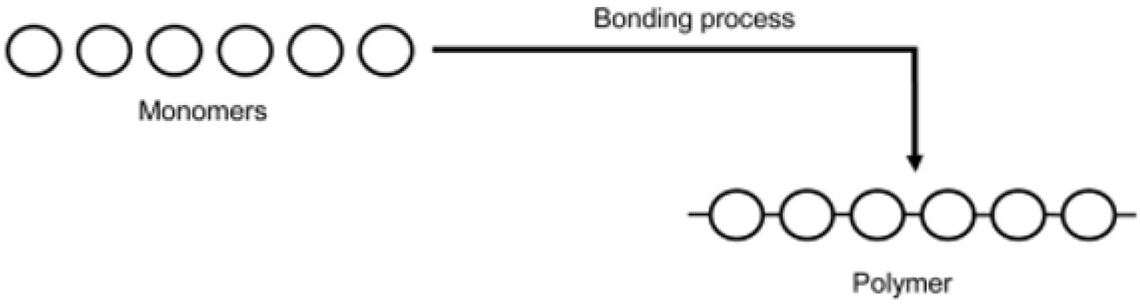
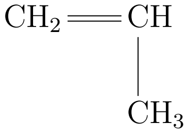
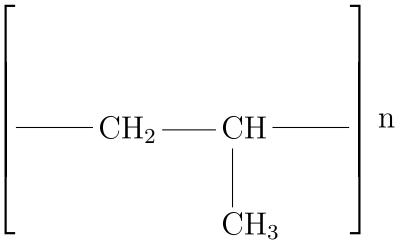
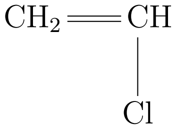
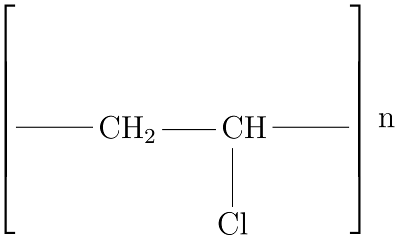
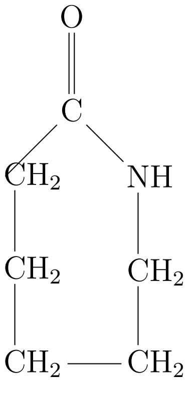
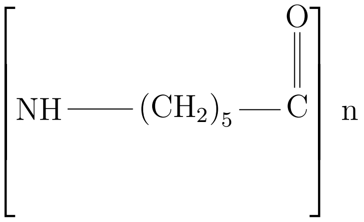


Figure 2: Bonding of monomers to form a polymer

Although the terms polymers and plastics are sometimes used as synonyms, there is a distinction to be made. The polymer is the pure material that arises from the polymerization process, and it is commonly used to refer to materials with long chain-like molecules. This description includes a variety of naturally occurring polymers, including collagen, cellulose, keratin, and rubber, all of which are still frequently utilized today. Newer synthetic variants of rubber are progressively replacing natural rubber, and all types of rubber can be categorized as elastomers because of their highly elastic mechanical properties. Pure polymers are rarely used on their own, and the name “plastic” is only applied when additives are present [14].

Table 1: Examples of some common plastics and their monomers

Monomers		Polymer	
Ethylene	$CH_2=CH_2$	Polythylene	$[-CH_2-CH_2-]n$
Propylene	 $CH_2=\underset{\underset{\textstyle CH_3}{\vert}}{CH}$	Polypropylene	 $\left[-CH_2-\underset{\underset{\textstyle CH_3}{\vert}}{CH}-\right]n$
Vinyl chloride	 $CH_2=\underset{\underset{\textstyle Cl}{\vert}}{CH}$	Polyvinyl chloride	 $\left[-CH_2-\underset{\underset{\textstyle Cl}{\vert}}{CH}-\right]n$
Caprolactame		Poly(E-Caprolactame) (PA6)	 $\left[NH-(CH_2)_5-\overset{\overset{\textstyle O}{\Vert}}{C}\right]n$
Tetrafloure-thylene	$CF_2=CF_2$	PolyTetrafloure-thylene (PTFE)	$[-CF_2-CF_2-)]n$

0.3 Source of polymers

Polymeric materials are used to make all plastics. These polymeric compounds can be found naturally in plants and animals or created artificially in laboratories. Based on their source of origin, plastic materials are classified as natural, semisynthetic, or synthetic[16].

0.3.1 Natural

Natural polymers are materials that are normally found in nature or that are taken from plants or animals. Natural polymers are crucial to daily life since they are the foundation of our human forms. Natural polymers include proteins and nucleic acids found in the human body, cellulose, natural rubber, silk, and wool. Natural rubber is a polymer formed from the latex of a rubber tree, and starch is a natural polymer made up of hundreds of glucose molecules. Honey is an example of a naturally occurring polymer that is widely used in daily life. Natural polymers from plants (latex from rubber trees) and animals are depicted in Figure 1.2. (honey from bees).

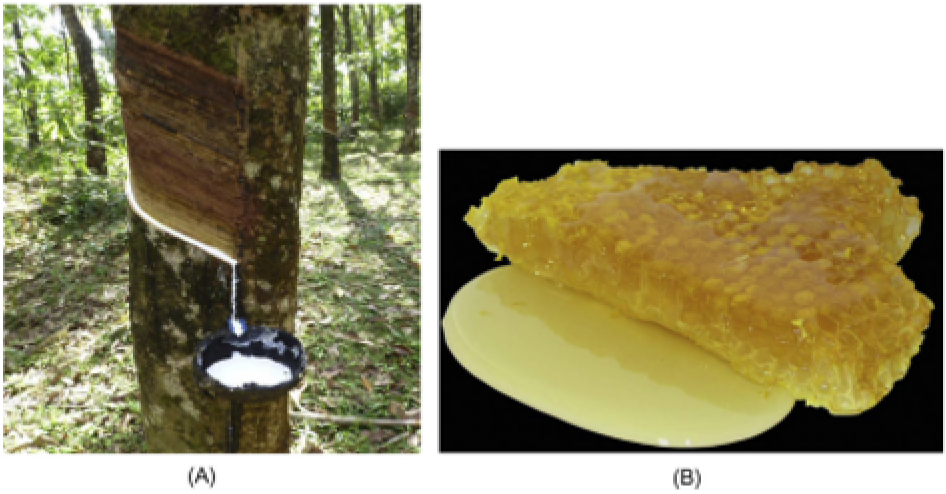


Figure 3: A) latex from the rubber tree. B) Honey from bees

0.3.2 Synthetic

Synthetic polymers are defined as polymers that are artificially produced in laboratories. These are also known as man-made polymers. Some of the examples of synthetic polymers are polyethylene (PE), polystyrene (PS), polyamides (nylon), poly (vinyl chloride) (PVC), synthetic rubber, Teflon, epoxy, and several others.

Table 2: Comparison of Natural and Synthetic Polymers

Natural Polymers	Synthetic Polymers
Occurs naturally	Artificially produced
Have been in used since millions of years	Have been made significant since the last 125 years
Natural reaction controls the properties	Highly engineered properties could be determined by controlling the reaction
Usually biodegradable	Some synthetic polymers are biodegradable
Similar chain lengths of molecules	Chain lengths could be significantly varied based on the reaction conditions
Backbone could be of carbon, oxygen, and nitrogen	Backbone is mostly carbon
Environmentally friendly	Environmental friendliness is of concern
Limited recyclability	Some of the synthetic polymers could recycled multiple times

Synthetic polymers are made up of carbon-carbon bonds and are typically generated from petroleum oil in a controlled environment. The chemical bonds that hold monomers together are altered by heat and pressure in the presence of a catalyst, leading them to connect with one another. A catalyst is a substance that is used to initiate or speed up chemical reactions between monomers.

Synthetic polymers are used in millions of everyday applications. Thermoplastics, thermosets, elastomers, and synthetic fibers are all used in these applications. Some of the qualities and attributes of natural and synthetic polymers are compared in Table 1.2.

0.3.3 Semisynthetic or Regenerated

Semisynthetic or regenerated polymers are defined as polymers created by chemical modification of naturally existing polymers. Vulcanized rubber, cellulose acetate, and rayon are just a few examples. Rayon is manufactured by processing cellulose chemically to create long fibers. Semisynthetic polymers are becoming increasingly popular in textile and medicinal applications. Chitosan, a linear polymer derived chemically from shrimp and other crustacean shells, has been found to have new applications in biomedicine.

0.4 Types of plastics

Plastic materials are classified into different classes based on their macromolecular structure and temperature-dependent physical characteristics. Figure 1.3 depicts a general overview of plastic classification with some typical examples.
Thermoplastics are elastomers with a hard or tough elasticity that can be melted with heat (mechanical, thermal or radiation energy). Elastomers have a gentle elasticity and cannot be melted in most cases. Thermosets are in the hard elasticity application range and cannot be melted[17].

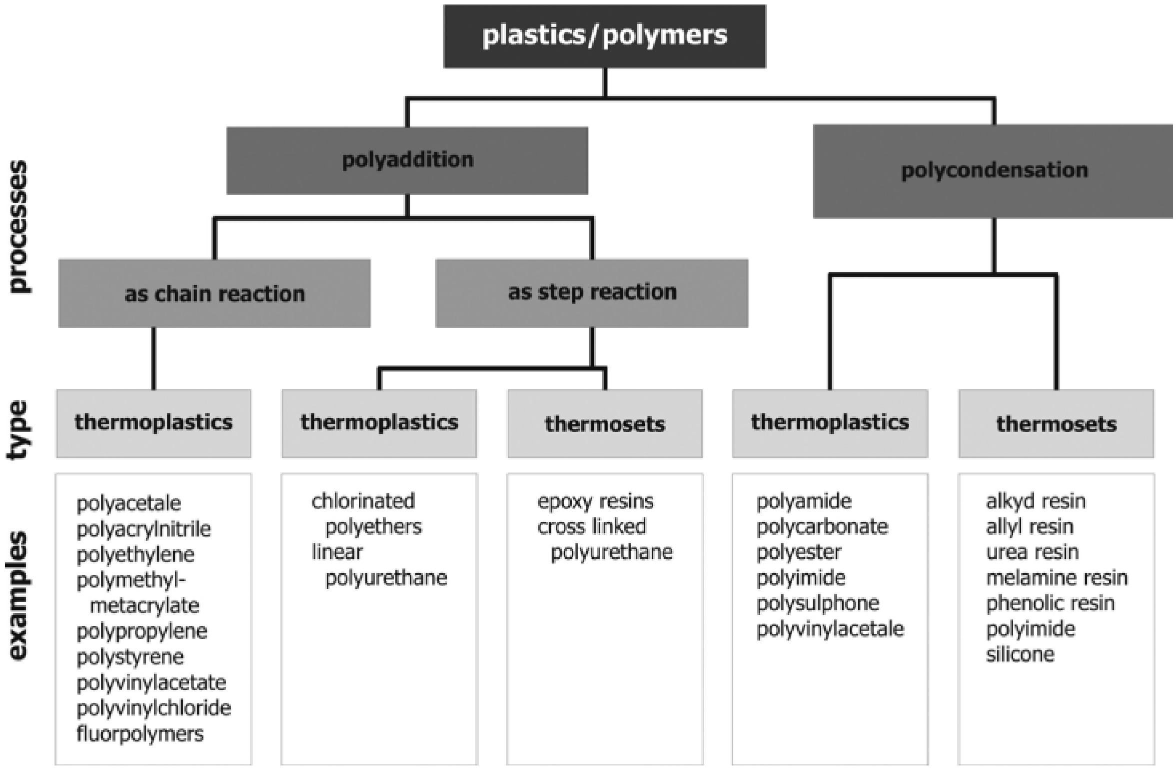


Figure 4: Processes for generating plastics and examples

0.4.1 Thermoplastic materials

The very long chain-like molecules in a thermoplastic material are kept together by relatively weak van der Waals forces. A mass of randomly dispersed long strands of sticky wool is a useful image of the structure. When a thermoplastic material is heated, the intermolecular tensions weaken, causing it to become soft and flexible, and eventually a viscous melt at high temperatures. When the substance cools, it solidifies once more. This cycle of heat softening and cold solidifying may be repeated indefinitely, which is a big advantage because it is the foundation of most processing procedures for these materials. It does, however, have disadvantages because it means that thermoplastic properties are heat sensitive. These materials, like candle wax, may be repeatedly softened by heat and will solidify when cooled, which is a good analogy that is commonly used to describe them[14].

The lengths of individual polymer chains and whether they have side branches have a significant impact on the characteristics of thermoplastics. The molecular weight and molecular weight distribution of the polymer are usually used to calculate chain lengths. Steps can be made during the polymerization process to modify the lengths of the developing chains and to induce the formation of side branches. This allows thermoplastic materials with the same chemical formula to exist in a variety of molecular configurations, each with its own set of properties. This is referred to as isomerism. Polyethylene, the most common thermoplastic, is an excellent illustration of this. When it exists in the form of a largely linear chain, as shown in Figure 1.4(a), the chains can be packed tightly together and the material is known as high density polyethylene (HDPE). When longer side branches are encouraged to grow as shown in Figure 1.4(b), the chains can no longer pack together so tightly and the material is known as low density polyethylene (LDPE). Other variations include linear low-density polyethylene (LLDPE) and ultra-high molecular weight polyethylene (UHMWPE). Other examples of common commercial thermoplastics are polyvinyl chloride (PVC), polystyrene, nylon, cellulose acetate, polycarbonate, polyethylene terephthalate (PET), polymethyl methacrylate, polylactic acid (PLA) and polypropylene.

The presence of a crystalline (ordered) or amorphous (random) structure is an essential distinction within the thermoplastic category of materials. Due to the intricate physical nature of the molecular chains, it is impossible for a molded plastic to have a perfectly crystalline structure in practice. Although some polymers, such as polyethylene and nylon, can acquire high levels of crystallinity, they are more correctly classified as partially crystalline or semi-crystalline. Acrylic and polystyrene, for example, are always amorphous. The presence of crystallinity in crystallizing plastics is highly reliant on their thermal history and, as a result, on the processing conditions utilized to create the molded object. In turn the mechanical properties of the molding are very sensitive to whether or not the plastic possesses crystallinity.

In general, plastics have a higher density when they crystallize due to the closer packing of the molecules. Typical characteristics of crystalline and amorphous plastics are shown below:

Table 3: Typical characteristics of crystalline and amorphous plastics

Amorphous	Crystalline
Broad softening range Thermal agitation of the molecules breaks down the weak secondary bonds. The rate at which this occurs throughout the formless structure varies producing a broad temperature range for softening.	**Sharp melting point** The regular close-packed structure results in most of the secondary bonds being broken down at the same time. This results in a sharp crystalline melting point.
Usually transparent The looser structure transmits light so the material appears transparent.	**Usually opaque** The difference in refractive indices between the two phases (amorphous and crystalline) causes interference to light so the material appears translucent or opaque.
Low shrinkage All thermoplastics are processed in the amorphous state. On solidification, the random arrangement of molecules produces little volume change and hence low shrinkage.	**High shrinkage** As the material solidifies from the amorphous state the polymers take up a closely packed, highly aligned structure. This produces a significant volume change manifested as high shrinkage.
Low chemical resistance The more open random structure enables chemicals to penetrate deep into the material and to destroy many of the secondary bonds.	**High chemical resistance** The tightly packed structure prevents chemical attack deep within the material.
Poor fatigue and wear resistance The random structure contributes little to fatigue or wear properties.	**Good fatigue and wear resistance** The uniform structure is responsible for good fatigue and wear properties.

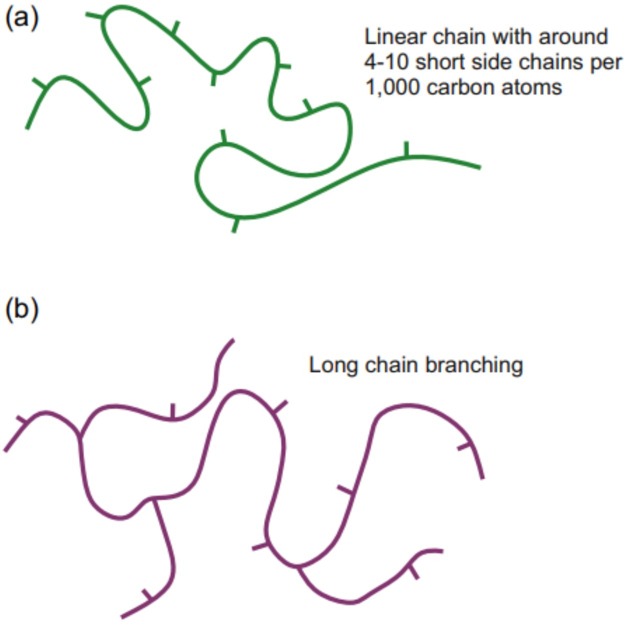


Figure 5: Chain structures of Polyethylene

0.4.2 Thermosetting plastics

A thermosetting plastic is made through a two-stage chemical reaction. The first stage produces long chain-like molecules comparable to those found in thermoplastics, but with the ability to react further.
The second stage of the reaction (chain cross-linking) occurs during the molding process, usually under the influence of heat, pressure, or, in rare cases, UV radiation. When cooled, the resulting moulding will be stiff, and a close network structure will have formed within the material.
The lengthy molecular chains were interconnected by strong bonds in the second stage, preventing the material from being softened again by heat. Figure 1.5 depicts the crosslinked structure of a typical thermoset. These materials will scorch and disintegrate if exposed to too much heat. Boiling an egg is a good analogy for this type of behavior.
Once the egg has cooled and is hard, it cannot be softened again by the application of heat.

Because molecules are cross-linked by strong chemical bonds, thermosetting polymers are often highly rigid and their mechanical properties are not affected by heat. Phenol formaldehyde, melamine formaldehyde, urea formaldehyde, epoxy, silicone, and certain polyesters are examples of thermosets.

Table 4: Examples of amorphous and crystalline thermoplastics

Amorphous	Crystalline
Polyvinyl chloride (PVC)	Polyethylene (PE)
Polystyrene (PS)	Polypropylene (PP)
Polycarbonate (PC)	Polyamide (PA)
Acrylic (PMMA)	Acetal (POM)
Acrylonitrile-butadiene-styrene (ABS)	Polyesters (PET, PBT)
Polyphenylene (PPO)	Fluoropolymers (PTFE, PFA, FEP, ETFE)
	Polylactic acid (PLA)

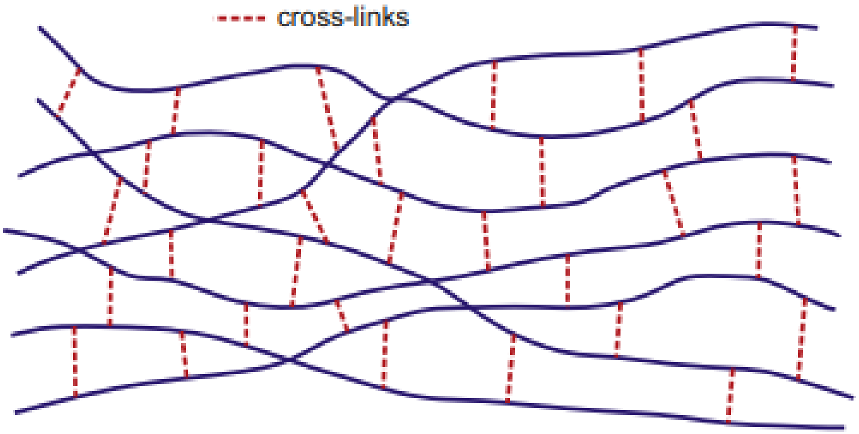


Figure 6: Cross-linked structure of a typical thermoset.

0.4.3 Elastomers

Plastics with a wide netlike crosslinking between the molecules are known as elastomers. They usually can't be melted without the molecular structure degrading. Elastomers are soft elastic above the glass temperature Tg, as the state of application (Figure 1.6). They are hard stretchy and brittle below Tg. With an increase in the number of crosslinks, the glass temperature rises. Butadiene resin (BR), styrene butadiene resin (SBR), and polyurethane resin (PUR) are examples of elastomers .

Raising the temperature causes an increase in elasticity, which is produced by the crosslink's stiffening effects being reduced and the mobility of the molecular chains being increased. The atom connection within and between the molecule chains is disrupted when the decomposition temperature Td is exceeded, and the substance is chemically destroyed[17].

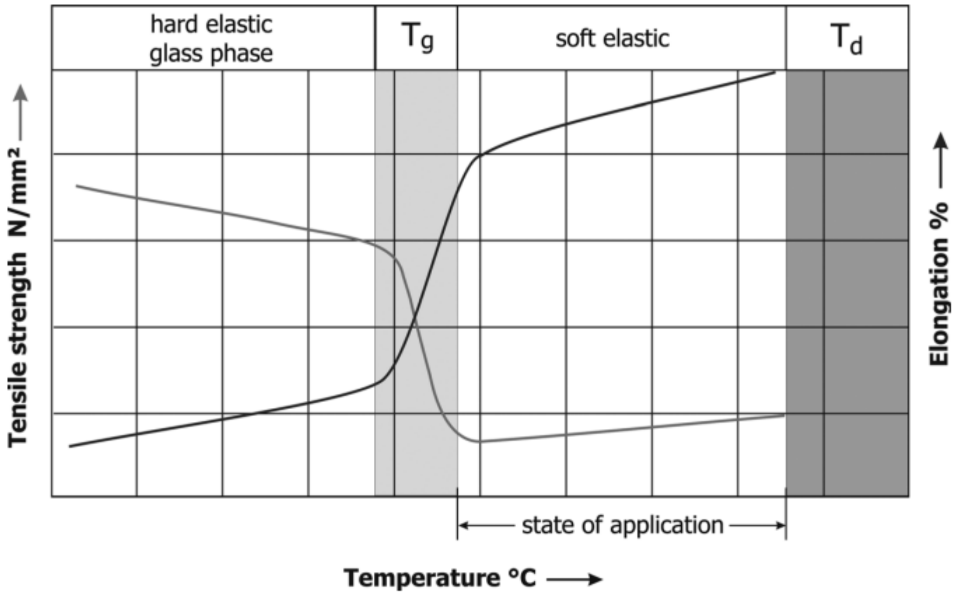


Figure 7: Temperature behavior of mechanical properties of elastomers (schematically).

0.5 Plastic materials properties

- **Density**

Plastics have a low density, ranging from $0.9 g/cm^3$for polypropylene, up to 2.3 for polytetrafluoroethylene, with the majority being between 0.9 and 1.5.
Certain techniques and additives make it possible to obtain, expanded thermoplastics of density ranging from 0.4 to 0.9[18].

Table 5: Typical density, tensile and shear moduli for a range of polymers.(^{a}modulus at 20 ∘C for small strains (<0.2%)).[14]

Materia	Density (kg/m^3)	Tensile modulus $(E)^a$. (GPa)	Shear modulus (G) (GPa)	Poisson's ratio (v)
Polystyrene (PS)	1050	2.65	0.99	0.33
Polymethyl methacrylate (PMMA)	1180	3.10	1.16	0.33
Polyvinyl chloride (PVC) (unplasticised)	1480	3.15	1.13	0.39
Nylon 66 (at 65% RH)	1140	0.99	0.34	0.44
Acetal homopolymer (POM)	1410	3.24	1.15	0.41
Acetal copolymer (POM)	1410	2.52	0.93	0.39
Polyethylene e High density (HDPE)	955	1.05	0.39	0.34
Polyethylene e Low density (LDPE)	920	0.32	0.11	0.45
Polypropylene homopolymer (PP)	910	1.51	0.55	0.36
Polypropylene copolymer (PP)	902	1.13	0.40	0.40
Polylactic acid (PLA)	1252	3.50	1.29	0.36
Polycaprolactam	1146	4.00	1.43	0.40
Polycarbonate (PC)	1194	2.77	0.99	0.40
Polyethersulphone	1390	2.76	0.98	0.41
Polyphenylene oxide (PPO)	1073	2.94	1.07	0.37
Polytetrafluoroethylene (PTFE)	2180	3.06	1.16	0.32
Phenolic	1220	5.79	2.13	0.36
Epoxy	1184	5.05	1.83	0.38

- **Optical characteristics**

Some plastics can be transparent or translucent. Among the transparent ones we can mention Poly (methyl methacrylate), polystyrene, styrene-acrylonitrile, methacrylate-butadiene-styrene, polycarbonate, flexible or rigid crystal vinyl compound, polyethylene terephthalate [18].

- **Coloring**

The coloring of the mass, from crystal or natural type resins, adds an attractive note to the presentation of the plastic profile, which can also be made perfectly opaque.

- **Mechanical characteristics**

The mechanical characteristics present a great variety of values, according to the nature of the material. Thus, the modulus of elasticity can reach 4 200 MPa for certain thermoplastics not filled. Of course, thermoplastics can also be filled, and their modulus of elasticity can then reach 17,000 MPa. However, the other characteristics can evolve simultaneously in various directions depending on the composition of the material.
The low coefficients of friction of some plastics (polytetrafluoroethylene, polyethylene, POM, polyamides, etc.) should also be noted.

- **Electrical characteristics**

Plastics, generally insulating, have excellent dielectric properties. They are widely used in the construction of electrical equipment and cabling.

- **Chemical characteristics**

Plastics are variously resistant to chemical actions. The range of current materials allows to solve most of the problems posed.

- **Thermal characteristics**

The thermal conductivity of plastics is relatively low (even very low in the case of lightweight plastics), which is of particular interest in insulation problems, in the building industry, household appliances, etc. In general, the current deformation temperature of thermoplastics is around 80°C. For some of them, it can rise to 150°C and even more: fluorinated and some technical polymers can reach temperatures of 250 to 300°C in continuous service.
The value of the coefficient of linear thermal expansion depends on the materials and their composition.

Table 6: Typical thermal properties of materials.[14]

Material	Density(kg/m^3)	Specific heat (kJ/kg/K)	Thermal conductivity (W/m/K)	Coeff. of therm exp (μm/m/∘C)	Thermal diffusivity (m^2/s) x 10^{-7}	Glass transition Temp, T$_8$ (∘C)	Max. operating, Temp (∘CC)
ABS	1040	1.3	0.25	90	1.7	115	70
Acetal(homopolymer)	1420	1.5	0.2	80	0.7	-85	85
Acetal(copolymer)	1410	1.5	0.2	95	0.72	-85	90
Acrylic	1180	1.5	0.2	70	1.09	105	50
Cellulose acetate	1280	1.6	0.15	100	1.04	-	6
CAB	1190	1.6	0.14	100	1.27	-	60
Epoxy	1200	0.8	0.23	70	-	-	130
Modified PPO	1060	-	0.22	60	-	-	120
Nylon 66	1140	1.7	0.24	90	1.01	56	90
Nylon 66 (33%glass)	1380	1.6	0.52	30	1.33	e-	100
PEEK	1300	-	-	48	-	143	204
PEEK (30%carbon)	1400	-	-	14	-	-	255
PET	1360	1.0	0.2	90	-	75	110
PET (36% glass)	1630	-	-	40	-	-	150
PHB	1230	1.5	0.22	100	1.19	15	80
Phenolic (glass filled)	1700	-	0.5	18	-	-	185
PLA	1250	1.2	0.13	70	0.87	55	80
Polyamide-imide	1400	-	0.25	36	-	260	210
Polycarbonate	1150	1.2	0.2	65	1.47	149	125
Polyester	1200	1.2	0.2	100	-	-	
Polyetherimide	1270	-	0.22	56	-	200	1

Table 7: Typical thermal properties of materials.cont.[14]

Polyethersulphone	1370	-	1.18	55	-	230	180
Polyimide	1420	-	-	45	-	400	260
Polyphenylene sulphide	1340	-	-	49	-	85	150
Polypropylene	905	2.0	0.20	100	0.65	-10	10
Polysulphone	1240	1.3	-	56	-	180	170
Polystyrene	1050	1.3	0.15	80	0.6	100	50
Polythene (LD)	920	2.2	0.24	200	1.17	-120	50
Polyethylene (HD)	950	2.2	0.25	120	1.57	-120	55
PTFE	2100	1.0	0.25	140	0.7	-113	250
PVC (rigid)	1400	0.9	0.16	70	1.16	80	50
PVC (flexible)	1300	1.5	0.14	140	0.7	80	50
SAN	1080	1.3	0.17	70	0.81	115	6
DMC (polyester)	1800	-	0.2	20	-	-	130
SMC (polyester)	1800	-	0.2	20	-	-	130
Polystyrene foam	32	-	0.032	-	-	-	-
PU foam	32	-	0.032	-	-	-	-
Stainless steel	7855	0.49	90	10	-	-	800
Nickel chrome alloy	7950	-	12	14	-	-	90
Zinc	7135	0.39	111	39	-	-	-
Copper	8940	0.39	400	16		-	

- **Fire behavior**

The fire behavior of the profiles can vary depending on the material used, the geometry of the profile, the additives, the sector targeted and the conditions of use. Some materials are naturally self-extinguishing [15].

- **Behavior to climate**

Some thermoplastics have a good resistance to climatic agents (UV, infrared, salt spray, etc.).

- **Food and Medical Compatibility**

Some plastics have food and medical applications.

0.6 The main additives used in plastics

Antioxidants. In polymers, antioxidants are employed to prevent oxidative deterioration and heat oxidation. During processing at high temperatures, common polymers like polyethylene and polypropylene are susceptible to oxidative deterioration.

Antistatic agents. Because most polymers are poor conductors of current, they accumulate a charge of static electricity, which can have negative consequences. Interference with electrical equipment, dirt attraction, and surface adhering are all issues. Antistatic chemicals draw moisture from the air to the plastic surface, enhancing surface conductivity and lowering the risk of a spark or discharge.

Blowing agents. Blowing agents are added to plastics to create foam or cellular structures in the final product. They may either take the form of physical blowing agents (where a gas is injected at high pressure directly into the plastic melt during processing), or chemical blowing agents (where a chemical agent is added to the plastic, which then decomposes during processing to evolve gas).

Coupling agents. To increase the connection of the plastic to inorganic filler elements like glass fibers, coupling agents are applied. They play a crucial role in the majority of polymer-based composite materials. For this, inorganic compounds such as silane, titanate, and zirconia are utilized.

Fillers. Fillers are solid particles of generally inorganic materials that are used to improve the properties and prices of polymers. Short fibers or flakes, for example, improve a plastic's mechanical characteristics. Others, known as extenders, allow a huge volume of plastic to be made with a small amount of genuine polymer resin. Extenders made of low-cost materials like calcium carbonate, silica, and clay are widely employed.

Flame retardants. Most polymers, because they are organic materials, are flammable. Additives that contain chlorine, nitrogen, bromine, phosphorous or metallic salts reduce the likelihood that combustion will occur or spread.

Heat stabilizers. Heat stabilizers protect polymers during thermal processing and avoid product deterioration in short and long-term use at elevated temperatures. They are an essential additive for PVC, which is susceptible to rapid thermal degradation.

Lubricants. Lubricants such as wax or calcium stearate reduce the viscosity and stickiness of the molten plastic and improve its forming characteristics.

Nucleating agents. Nucleating agents are important for controlling the structure and properties of semi-crystalline plastics. They promote nucleation and crystallization and they ensure that crystal structures are regular with smaller and more numerous spherulites. They include inorganic agents such as talc, mica and calcium carbonate, and organic agents such as mineral oils.

Pigments. Pigments and dyes are used to produce colors in plastics. Many provide secondary benefits, e.g., carbon black and titanium dioxide (white) act as UV stabilizers. Pigments are insoluble and may be organic or inorganic. Dyes are soluble and invariably organic.

Plasticizers. Plasticizers are added to plastics to aid flow and processing. They are low molecular weight materials which alter the properties and forming characteristics of the plastic. They help to develop new, improved properties not present in the original plastic, e.g., the production of flexible grades of PVC by the use of plasticizers.

Reinforcement. The strength, stiffness and toughness of polymers are improved by adding fibers of glass, carbon, aramid etc. Natural fibers, such as hemp, flax and jute, may also be used. Reinforcement with nano-clay particles, carbon nanotubes and graphene are leading to the development of new nanocomposite materials.

Release agents. Release agents can play a vital role in preventing plastics from sticking to processing machinery and in some cases are often more appropriately termed as mold release agents.

UV stabilizers. UV stabilizers are used to prevent or protect plastics from degradation by ultraviolet rays and thereby extend the life of the end products. Discoloration and brittleness are the key problems they address[14].

0.7 Material Selection

The first and most important steps in the design process are to define clearly the purpose and function of the proposed product and to identify the service environment. Then one has to assess the suitability of a range of candidate materials[14].

0.7.1 Get in the right group

The following image puts plastics into groups depending on what is most important to the application[1].

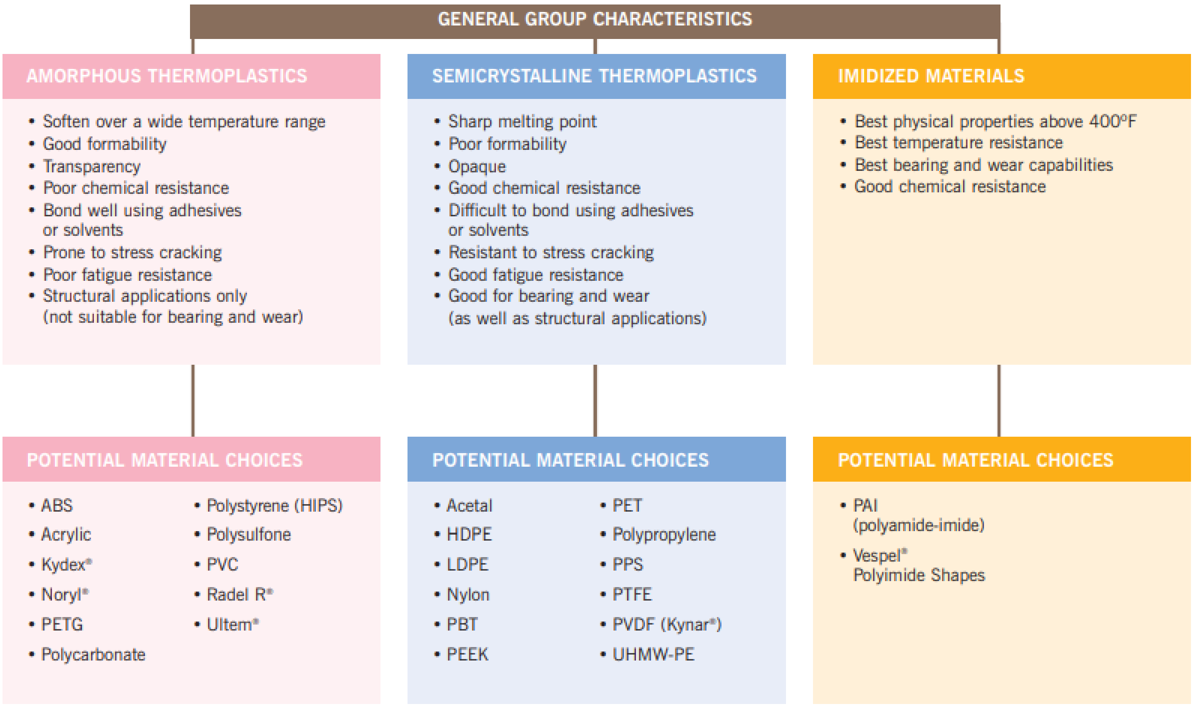


Figure 8: Plastics and their applications[1].

0.7.2 Temperature and cost

The following images compares plastics by temperature resistance and cost.

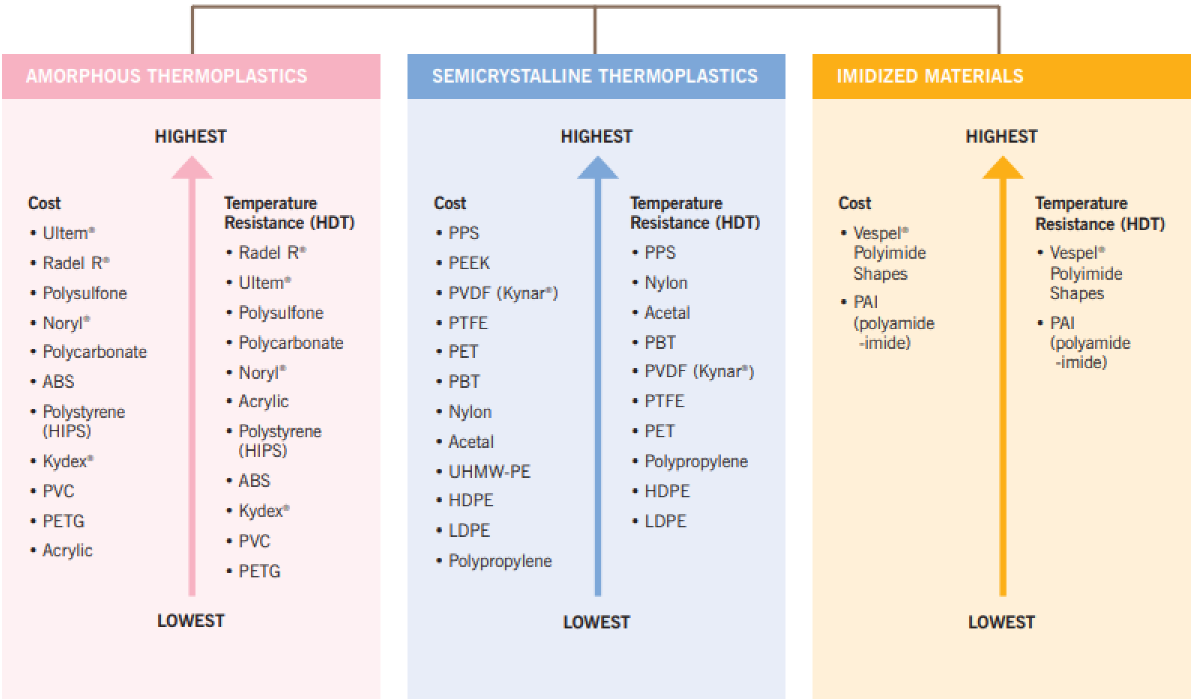
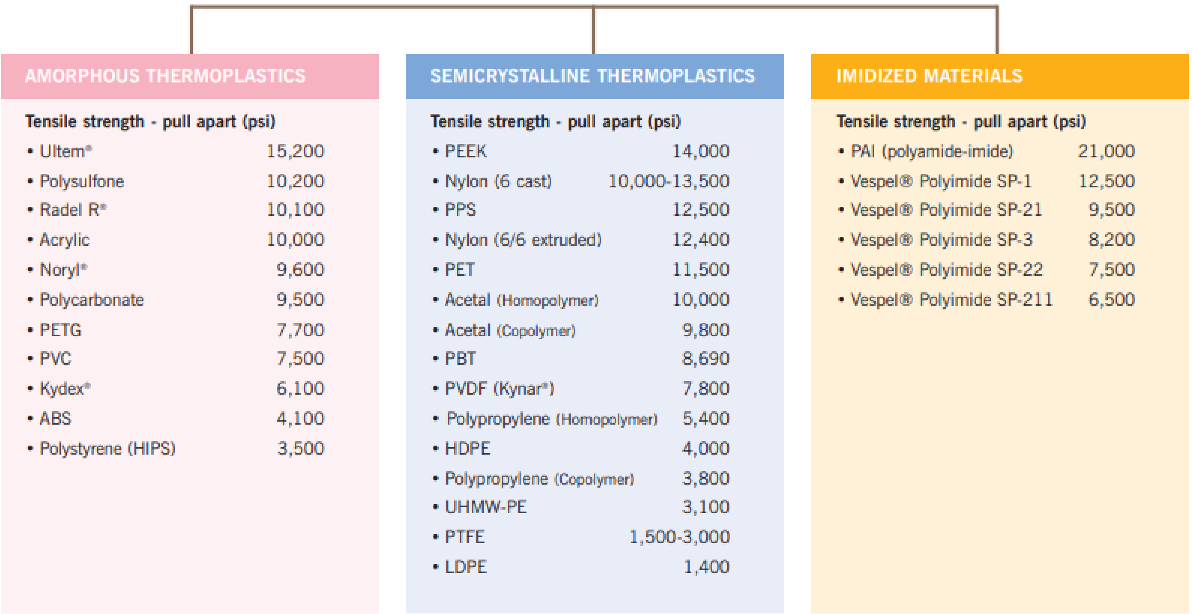


Figure 9: Plastics heat resistance and cost comparision[1].

0.7.3 Mechanical properties

The following image compares plastics by tensile stress (resistance to being pulled apart).



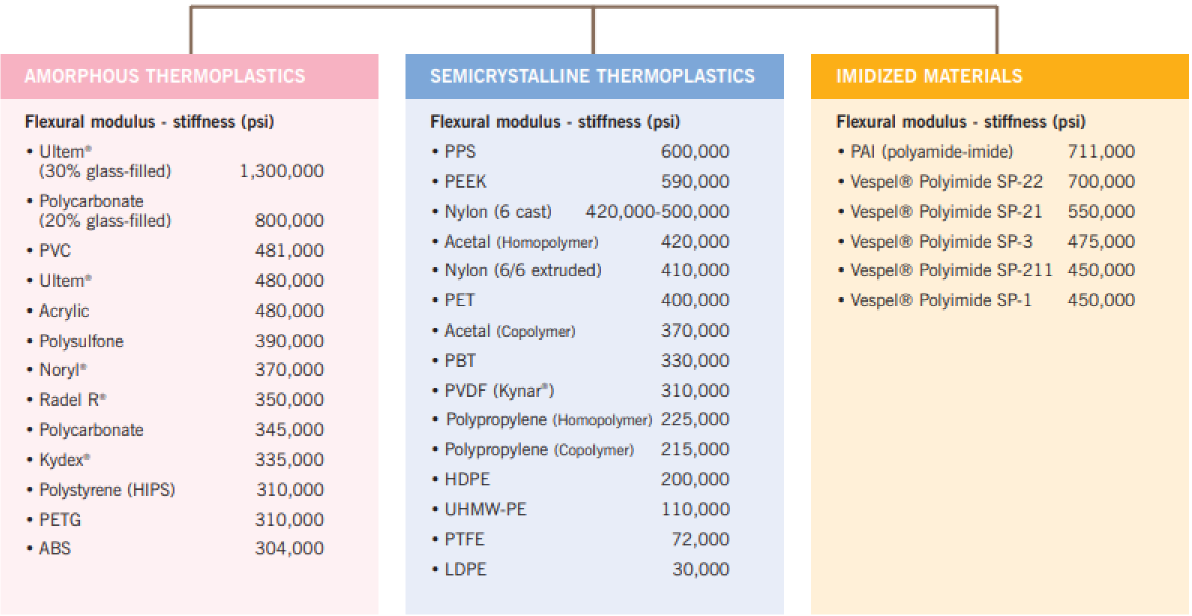
AMORPHOUS THERMOPLASTICS	
Tensile strength - pull apart (psi)	
• Ultem®	15,200
• Polysulfone	10,200
• Radel R®	10,100
• Acrylic	10,000
• Noryl®	9,600
• Polycarbonate	9,500
• PETG	7,700
• PVC	7,500
• Kydex®	6,100
• ABS	4,100
• Polystyrene (HIPS)	3,500

SEMICRYSTALLINE THERMOPLASTICS	
Tensile strength - pull apart (psi)	
• PEEK	14,000
• Nylon (6 cast)	10,000-13,500
• PPS	12,500
• Nylon (6/6 extruded)	12,400
• PET	11,500
• Acetal (Homopolymer)	10,000
• Acetal (Copolymer)	9,800
• PBT	8,690
• PVDF (Kynar®)	7,800
• Polypropylene (Homopolymer)	5,400
• HDPE	4,000
• Polypropylene (Copolymer)	3,800
• UHMW-PE	3,100
• PTFE	1,500-3,000
• LDPE	1,400

IMIDIZED MATERIALS	
Tensile strength - pull apart (psi)	
• PAI (polyamide-imide)	21,000
• Vespel® Polyimide SP-1	12,500
• Vespel® Polyimide SP-21	9,500
• Vespel® Polyimide SP-3	8,200
• Vespel® Polyimide SP-22	7,500
• Vespel® Polyimide SP-211	6,500

Figure 10: Plastics tensile stress comparision[1].

The following image compares plastics by their bending stiffness.



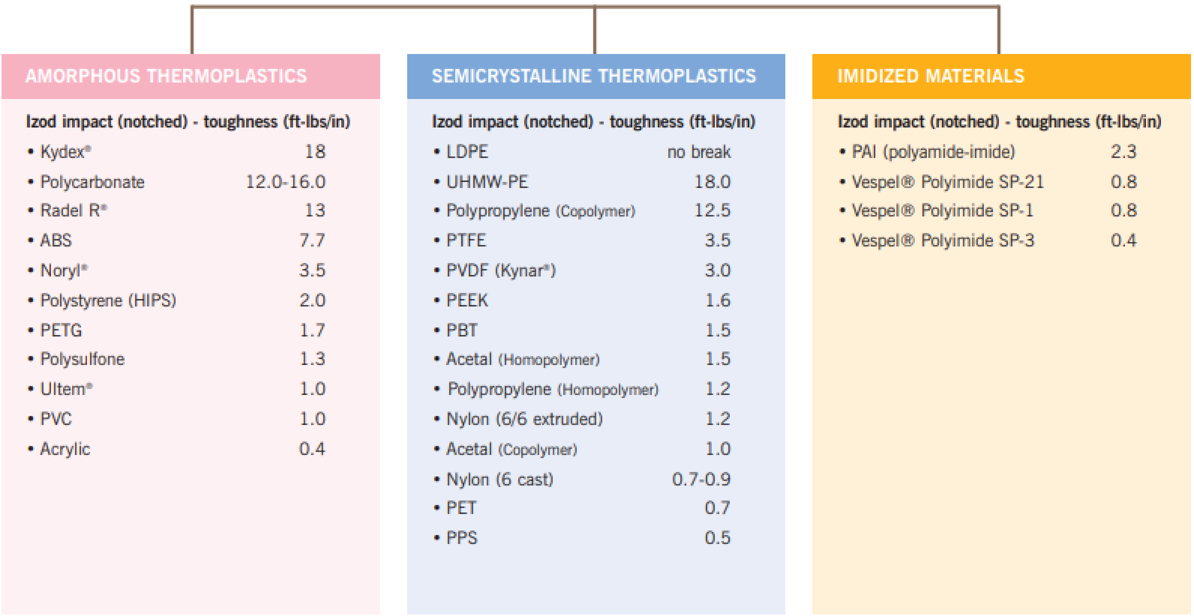
AMORPHOUS THERMOPLASTICS	
Flexural modulus - stiffness (psi)	
• Ultem® (30% glass-filled)	1,300,000
• Polycarbonate (20% glass-filled)	800,000
• PVC	481,000
• Ultem®	480,000
• Acrylic	480,000
• Polysulfone	390,000
• Noryl®	370,000
• Radel R®	350,000
• Polycarbonate	345,000
• Kydex®	335,000
• Polystyrene (HIPS)	310,000
• PETG	310,000
• ABS	304,000

SEMICRYSTALLINE THERMOPLASTICS	
Flexural modulus - stiffness (psi)	
• PPS	600,000
• PEEK	590,000
• Nylon (6 cast)	420,000-500,000
• Acetal (Homopolymer)	420,000
• Nylon (6/6 extruded)	410,000
• PET	400,000
• Acetal (Copolymer)	370,000
• PBT	330,000
• PVDF (Kynar®)	310,000
• Polypropylene (Homopolymer)	225,000
• Polypropylene (Copolymer)	215,000
• HDPE	200,000
• UHMW-PE	110,000
• PTFE	72,000
• LDPE	30,000

IMIDIZED MATERIALS	
Flexural modulus - stiffness (psi)	
• PAI (polyamide-imide)	711,000
• Vespel® Polyimide SP-22	700,000
• Vespel® Polyimide SP-21	550,000
• Vespel® Polyimide SP-3	475,000
• Vespel® Polyimide SP-211	450,000
• Vespel® Polyimide SP-1	450,000

Figure 11: Plastics bending stiffness comparision[1].

The following image compares plastics by their toughness (resistance to impact[1].).



AMORPHOUS THERMOPLASTICS

Izod impact (notched) - toughness (ft-lbs/in)	
Kydex®	18
Polycarbonate	12.0-16.0
Radel R®	13
ABS	7.7
Noryl®	3.5
Polystyrene (HIPS)	2.0
PETG	1.7
Polysulfone	1.3
Ultem®	1.0
PVC	1.0
Acrylic	0.4

SEMICRYSTALLINE THERMOPLASTICS

Izod impact (notched) - toughness (ft-lbs/in)	
LDPE	no break
UHMW-PE	18.0
Polypropylene (Copolymer)	12.5
PTFE	3.5
PVDF (Kynar®)	3.0
PEEK	1.6
PBT	1.5
Acetal (Homopolymer)	1.5
Polypropylene (Homopolymer)	1.2
Nylon (6/6 extruded)	1.2
Acetal (Copolymer)	1.0
Nylon (6 cast)	0.7-0.9
PET	0.7
PPS	0.5

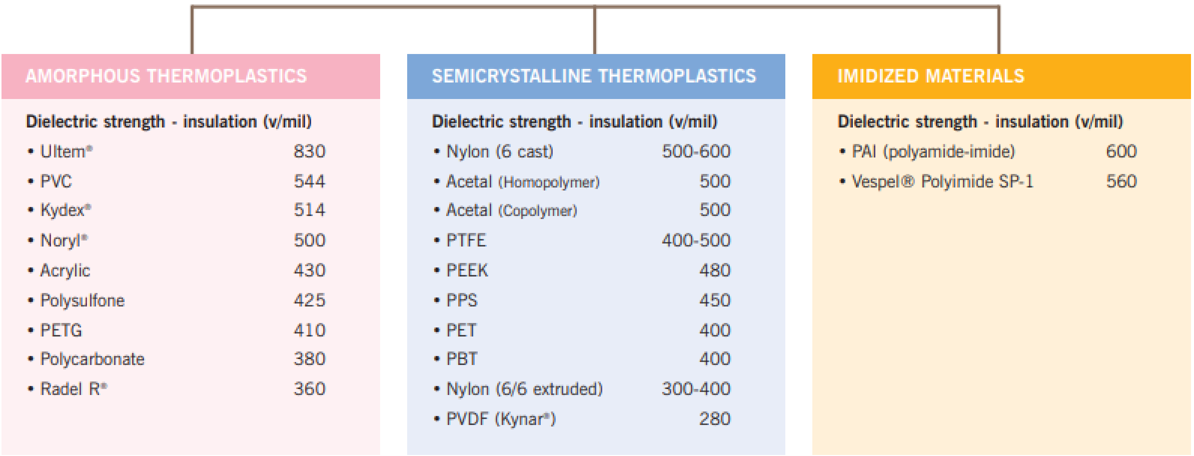
IMIDIZED MATERIALS

Izod impact (notched) - toughness (ft-lbs/in)	
PAI (polyamide-imide)	2.3
Vespel® Polyimide SP-21	0.8
Vespel® Polyimide SP-1	0.8
Vespel® Polyimide SP-3	0.4

Figure 12: Plastics toughness comparision[1].

0.7.4 Electrical insulation

The following image compares plastics by their Dielectric strength.



AMORPHOUS THERMOPLASTICS

Dielectric strength - insulation (v/mil)	
Ultem®	830
PVC	544
Kydex®	514
Noryl®	500
Acrylic	430
Polysulfone	425
PETG	410
Polycarbonate	380
Radel R®	360

SEMICRYSTALLINE THERMOPLASTICS

Dielectric strength - insulation (v/mil)	
Nylon (6 cast)	500-600
Acetal (Homopolymer)	500
Acetal (Copolymer)	500
PTFE	400-500
PEEK	480
PPS	450
PET	400
PBT	400
Nylon (6/6 extruded)	300-400
PVDF (Kynar®)	280

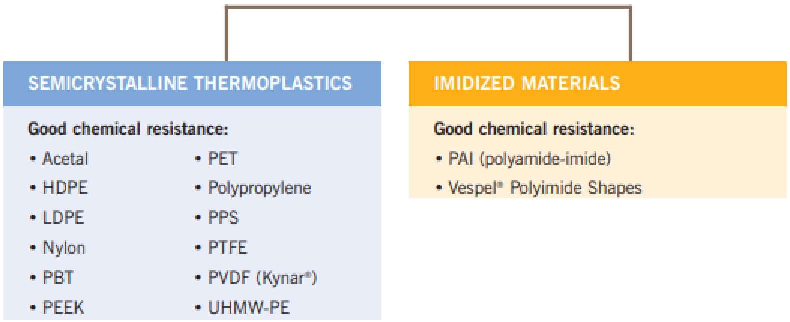
IMIDIZED MATERIALS

Dielectric strength - insulation (v/mil)	
PAI (polyamide-imide)	600
Vespel® Polyimide SP-1	560

Figure 13: Plastics dielectric strength comparision[1].

0.7.5 Chemical resistance

The following image compares plastics by their Chemical resistance.



SEMICRYSTALLINE THERMOPLASTICS

Good chemical resistance:

- Acetal
- HDPE
- LDPE
- Nylon
- PBT
- PEEK
- PET
- Polypropylene
- PPS
- PTFE
- PVDF (Kynar®)
- UHMW-PE

IMIDIZED MATERIALS

Good chemical resistance:

- PAI (polyamide-imide)
- Vespel® Polyimide Shapes

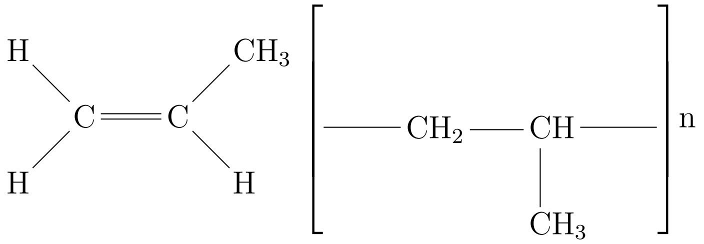
Figure 14: Plastics chemical resistance comparision[1].

0.8 Polypropylene

Polypropylene was chosen as the plastic material for this thesis's product design. The material was chosen based on the polypropylene's qualities and advantages in the application of the part design. Polypropylene, in particular, has remarkable features that are suitable and sought for part design production, such as light weight, strength, high heat resistance, and stiffness. Furthermore, polypropylene can be manufactured utilizing a variety of processes, the most common of which is the injection molding process for this part design [19].

0.8.1 Formation of Polypropylene

Basic constitute regarding the formation of polypropylene is the monomer propene, it has the following chemical structure:



Gas oil, ethane, naphtha, and propane are all used to make propene. Propene becomes polypropylene when it undergoes the necessary chemical reaction of addition polymerization. The Bulk Process and the Gas Phase Process are two industrially used polypropylene manufacturing processes that use a well-known catalyst called Ziegler Natta Catalyst, which is a combination of aluminum chloride and aluminum alkyl[19].

• **The Bulk Process**:
This polymerization process is carried on in the absence of a solvent where liquid propane is polymerized at a temperature of 340-360 K and a pressure of 30-40 atm and after the process is done solid propene polymer particles are separated from the liquid.

• **The Gas Phase Process**:
The gas phase process is carried on at a high temperature of 320-360 K and a relatively low pressure of 8-35 atm. Propene is mixed with a gas and passed over the catalyst during the process and then finally the polymer is separated from the gaseous propene and hydrogen.

0.8.2 Grades of Polypropylene

Polypropylene is available in three main forms of grade as:

• **Homo polymers**: Common purpose grade, most commonly applicable.

• **Block-Copolymer**: Which is specially designed for enhanced impact strength properties and composed with 5-15% ethylene.

• **Random Copolymers**: Random arrangement of the monomers in molecular chain base that results on enhanced flexibility and clarity.

0.8.3 Brands of polypropylene

Polypropylene has different brand names in the market, and the most common are mentioned below:

- Carlona P.
- Herkulon.
- Moplen.
- Napryl, Profax and Propathene[19].

0.8.4 Properties of polypropylene

Most common typical properties of the plastic material polypropylene are revised on the table below:

Table 8: Typical Properties of Polypropylene PP

ISO Test Property	HPP*	HPP-filled	CPP**	CPP-filled
ISO1183 **Specific gravity**	0.90-0.91	0.97-1.27	0.89-0.91	0.98-1.24
ISO62 **Warer absorption** (%)	0.01-0.03	0.01-00.09	0.03	0.01-0.02
ISO527 **Tensile strength**(MPa)	31.03-41.37	24.13-110.32	27.58-37.92	17.24-68.95
ISO527 **Elongation at brak**(%)	100-600	1.5-80	200-500	2.2-50
ISO527 **Tensile modulus**(MPa)	113.7-155.1	258.5-689.5	89.6-124.1	34.4-241.3
ISO178 **Fluxural modulus**(MPa)	117.2-172.3	144.8-689.5	89.6-137.9	144.8-661.9
ISO180 **Notched Izod impact strangth** (J/m)	21-75	32-641	59-747	32-214
ASTM **Hardness** Rockwell R	80-102	75-117	65-96	81-105
ISO8302 **Thermal conductivity** (W/mk)	0.22	0.25-0.51	0.22	0.25-0.51
ISO11359 **Coefficient of thermal conductivity** ($10^3$4m/m$\circ$C)	1.4-1.8	0.27-0.90	1.08-1.80	0.36-1.08
ISO75 **Deflection temperature** ($\circ$C) At 1.80 MPa At 0.45 MPa	49-60 107-121	54-166 104-149	49-60 85-104	47-138 77-152

*Homopolymer polypropylene **Copolymer polypropylene

0.9 Conclusion

In this chapter, it was concluded that plastics are widely used materials in various fields because of their properties (physical, chemical and mechanical) which meet different requirements, such as their light weight, surface finish and cost. In addition, it has enabled us to distinguish the different types of plastics, their production processes and their recycling possibilities.

PLASTIC PROCESSING

0.10 Introduction

Plastic processing can be defined as the process of converting the plastics raw material into semi-finished products. Ex: Buckets, Automobile Parts, Crates, Tanks, Pipes, Bottles, Carry bags, Ropes, Profiles etc [20].

At least 125,000 million pounds of plastic are consumed globally (by weight). About 36 percent is processed by extruders, 32 percent by injection molding, 10 percent by blow molding, 6 percent by calendars, 5 percent in coatings, 3 percent in compression, 2 percent in powder form, and 6 percent using other processes. These percentages do not correspond to the number of machines in use; for example, injection machines outnumber extruders by three to one[2].

0.11 Classification of processing methods

- **Primary Processing Methods** : Injection, Extrusion, Blow, Compression and transfer molding.
- **Secondary Processing Methods** : Roto, Thermoforming, Coating, Casting, Fabrication and Calendaring etc.
- **Tertiary Processing Methods** : Cutting, Drilling, Welding and Bending etc.[20]

0.12 Fundamentals of Processing

Processability is generally the ease or difficulty with which a plastic can be handled during its fabrication into film, molded products, pipe, profile etc.
A plastic with good processability has the qualities needed to make it simple to form it into the required shape. The molecular weight, homogeneity, additive type, content, and plastic feed rates are the primary features or properties that define a plastic's processability.

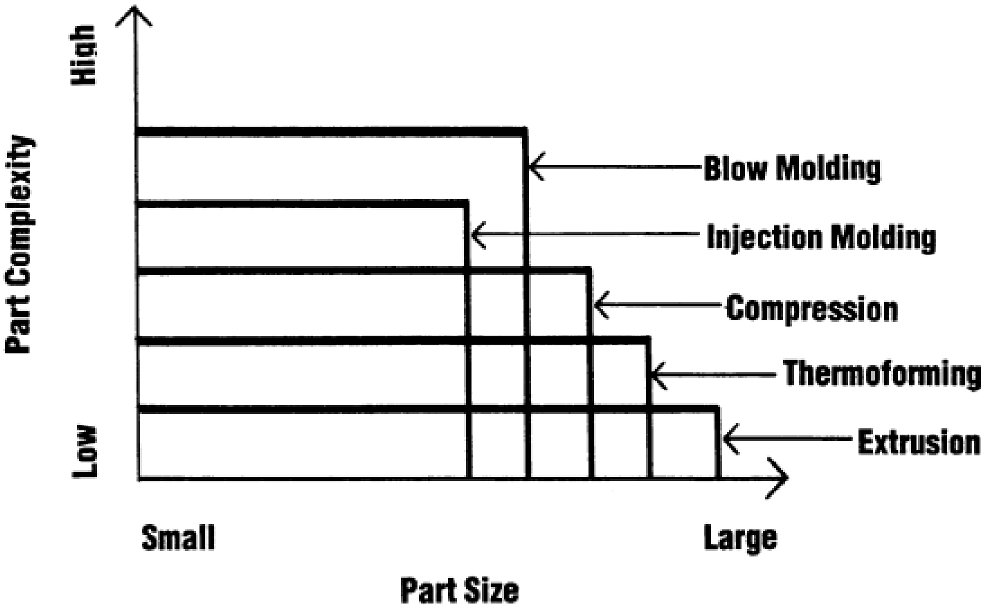


Figure 15: Processing characteristics.[2]

Table 9: Competitive Processes.)

	INJECTION MOLDING	EXTRUSION	BLOW MOLDING	THERMOFORMATION	REACTION INJECTION MOLDING	ROTATIONAL MOLDING	COMPRESSION AND TRANSFER MOLDING	MATCHED MOLD SPRAY UP
Bottles, necked containers, etc.	2, A		1	2, A		2		2
Cups, trays, open containers, etc.	1			1	1		1	2
Tanks, drums, large hollow shapes, etc.			1	2, A		1		2
Caps, covers, closures, etc.	1			2	2		1	
Hoods, housings, auto parts, etc	1		2	2	2		1	1
Complex shapes, thickness changes, etc.	1						1	2
Linear shapes, pipe, profiles, etc.	2, B	1					2, B	
Sheets, panels, laminates, etc.		1, C					2	2

1. Prime process.

2. Secondary process.

A. Combine two or more parts with ultrasonics, adhesives, etc.

B. Short sections can be molded.

C. Also calendering process.

0.13 Primary Processing Technology Types

0.13.1 The injection molding

Injection molding is a method of making a plastic product out of powdered thermoplastics by feeding the material via a machine component called a hopper into a heated chamber to soften it before forcing it into the mold with the help of a screw. In this whole process pressure should be constant till the material is hardened and is ready to be removed from the mold. This is the most popular and preferred method of making any type of plastic product, regardless of complexity or size.
Injection molding permits mass production net shape manufacturing of high precision, three-dimensional of plastic parts [21].

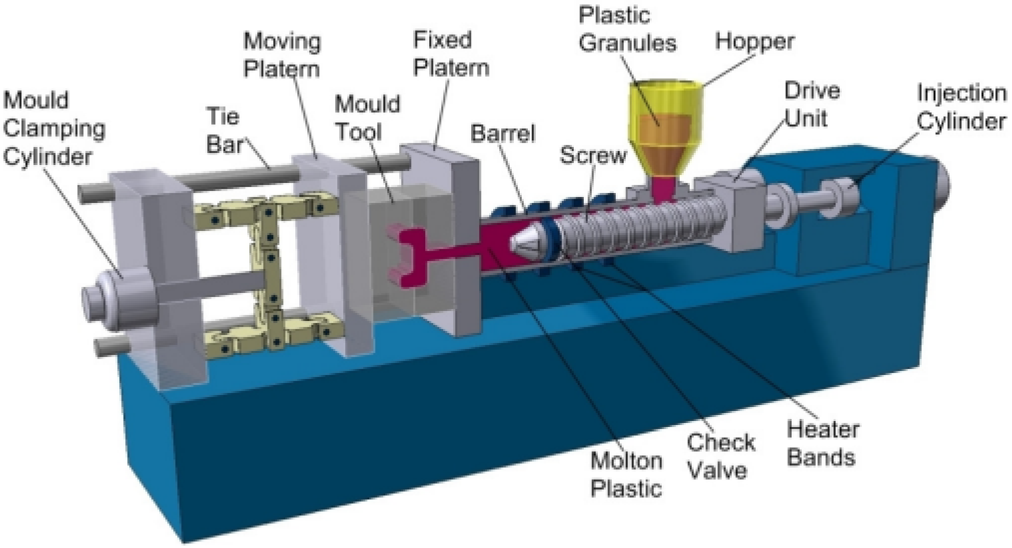


Figure 16: Injection Molding Machine.

0.13.1.1 The injection molding process

The machine used for plastic injection molding consists of two main units: the Injection Unit and the Clamping Unit. There are 6 steps which complete an injection molding process. The first 2 take place in the injection unit and the last 2 steps are done in the clamping unit.

- **Step 1** . Plastic granules are inserted into a hopper, which leads them into a heated cylinder.

- **Step 2** . Here they are melted into a molten state. This molten plastic is then passed into the split-die chamber in either of two ways: using a hydraulic

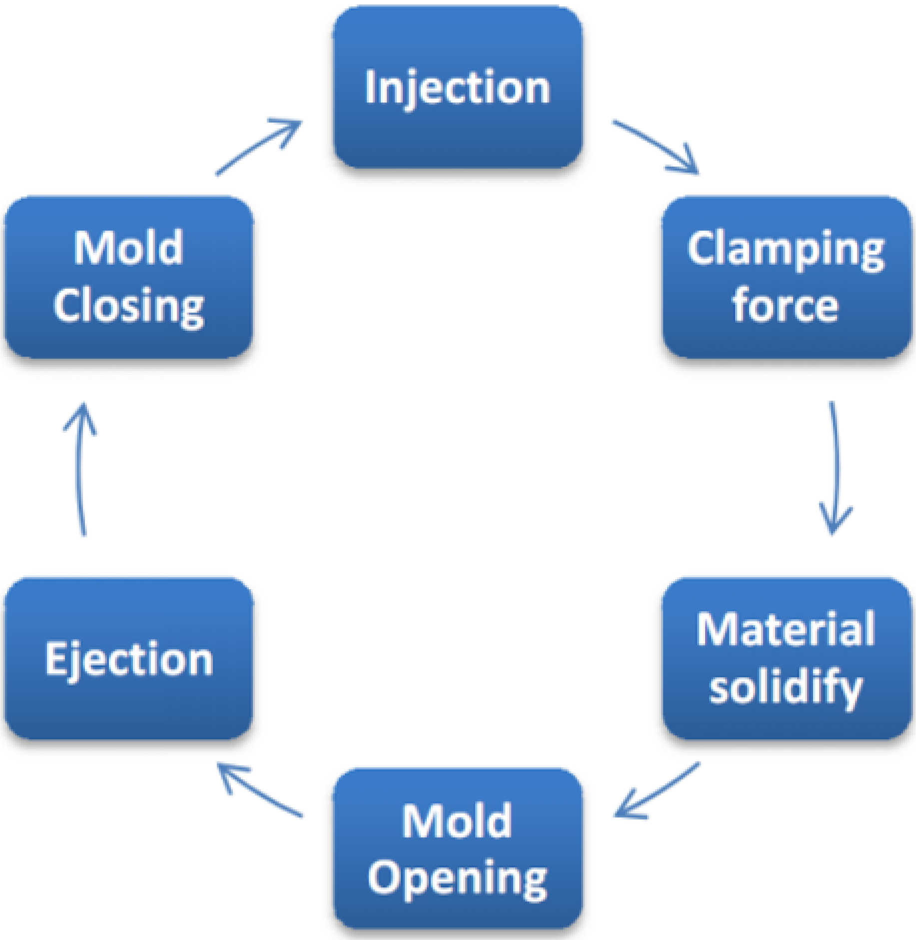


Figure 17: Injection molding cycle.

plunger, or more commonly a reciprocating rotating screw.

- **Step 3** . Material volume control. The rotating screw moves a pre-set distance in the opposite direction due to the pressure that builds up at the mold's entrance. This ensures that the required volume of plastic will be passed into the mold.

- **Step 4** . The screw moves forward, pushing plastic into the mold cavity, hence the term "Plastic Injection Molding". The clamping unit ensures that the two parts of the mold are perfectly aligned.

- **Step 5** . The screw moves back to its original position. After the part sets, cures (thermosets) or cools (thermoplastics) the mold opens up, so that it is removed from the cavity.

- **Step 6** . The mold closes so that the process can be repeated.

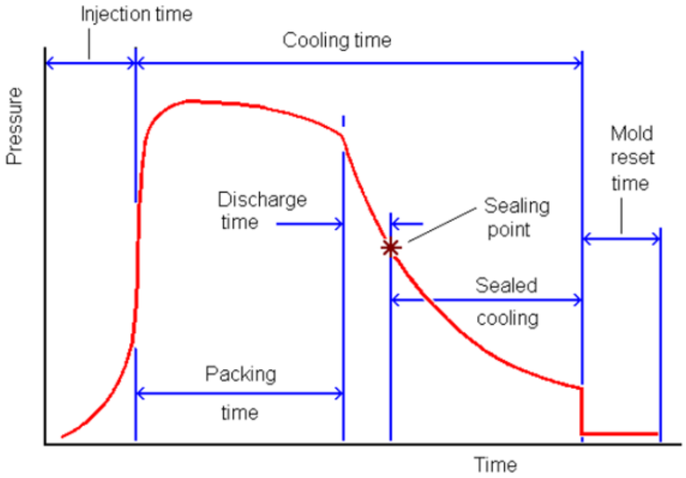


Figure 18: Plastic Injection Molding Cycle time breakdown

0.13.1.2 Injection molding machine

As for the injection molding machine, several types have been developed so far, but presently the in-line screw type injection molding machine as shown in Figure 3.35 has become the main type[22].

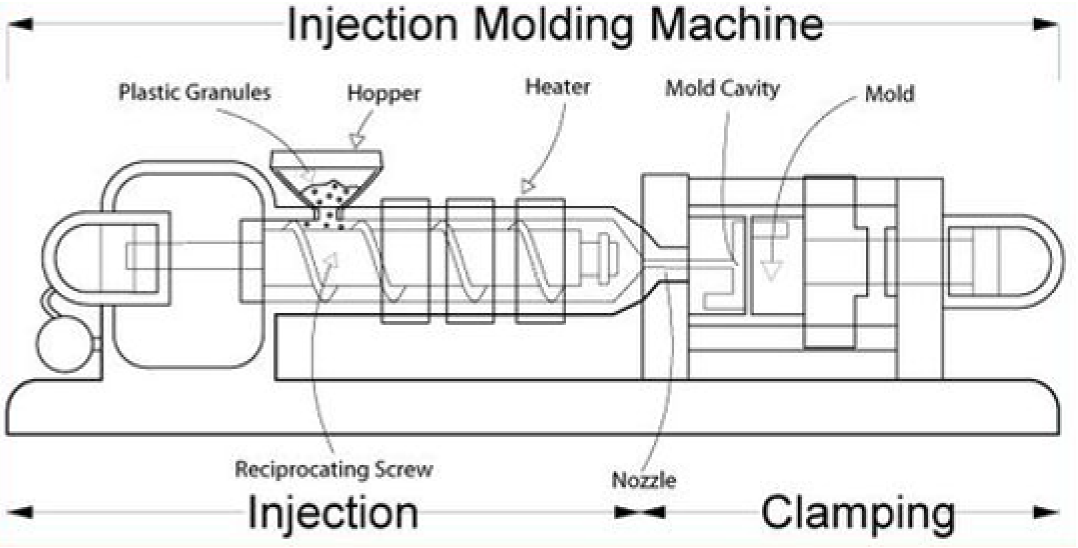


Figure 19: Plastic Injection Molding Machine and its Units

0.13.1.3 Injection molding machines units

The injection unit and the clamping unit make up the injection molding machine, and their features are explained below[22].

0.13.1.3.1 Injection unit .

Injection capacity: The proper injection capacity is found from the relationship of the molding machine capacity for the weight of 1 shot as shown in Figure 3.36. It is necessary to select the molding machine that satisfies the capacity of the shaded area. This figure is the summary of the actual molding results in the past, but basically, it is based on the following idea. Plasticizing time and injection time increase longer on the side where the capacity is small, and it is employed at the molding machine's narrow capacity. That is, a lack of filling is caused by a longer molding cycle and a slower filling rate.

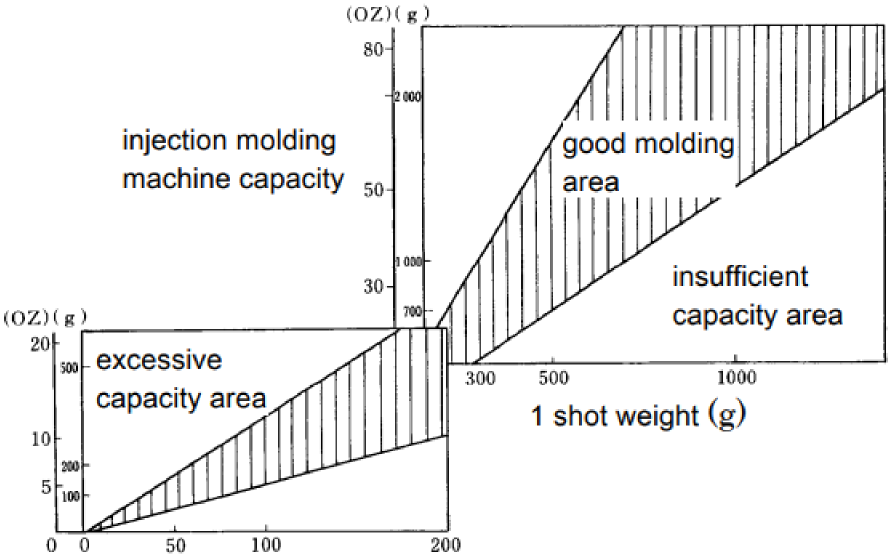


Figure 20: Selection of molding machine from the injection capacity

On the other hand, when the capacity is high, the resin's dwell time inside the cylinder increases, and the resin thermally decomposes. The capacity range in the figure is quite broad, but when working with materials that are easily thermally destroyed and contain a lot of colors and additives, it is best to mold at a shot weight of 70∼80 percent of the injection capacity.

Hopper: Plastic materials are given in the form of tiny pellets during the molding process. The hopper serves as a storage container for these pellets. The pellets are gravity fed into the barrel from the hopper.[21]

Barrel: The barrel's primary function is to provide support for the screw. The heater bands in the barrel serve as a temperature recorder for each segment of the barrel.

Screw: The reciprocating screw, also known as the compression screw, is used to compress, melt, and transfer plastic material. The feeding zone, the transition zone, and the 16-metering zone are the three zones that make up the Screw. The plastic materials will stay pellets in the feeding zone and will be transported to the transition zone, where the pellets will melt and the molten plastics will be sent to the metering zone, where the molten material will be ready for injection.

Nozzle: The nozzle's primary role is to link the barrel to the sprue bushing, which then seals the mold and barrel together. It's critical to match the nozzle temperature to the melt temperature of the materials.

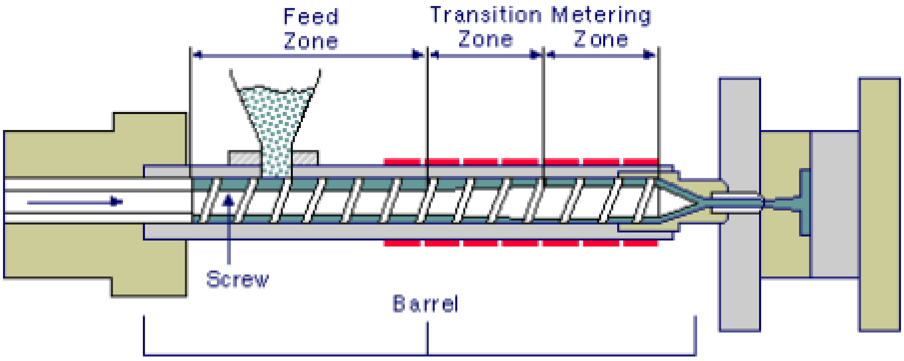


Figure 21: Different Zones of the Screw

0.13.1.3.2 Clamping unit clamping force F can be calculated by the following equation:

$$F(T)=(0.35\sim0.50)xS$$

where S: projected area as indicated in Figure 2.8. However, it is necessary to note that when the arrangement of molding is eccentric from the center of mold (center of die plate), the clamping force, which is higher than the above formula is required[22].

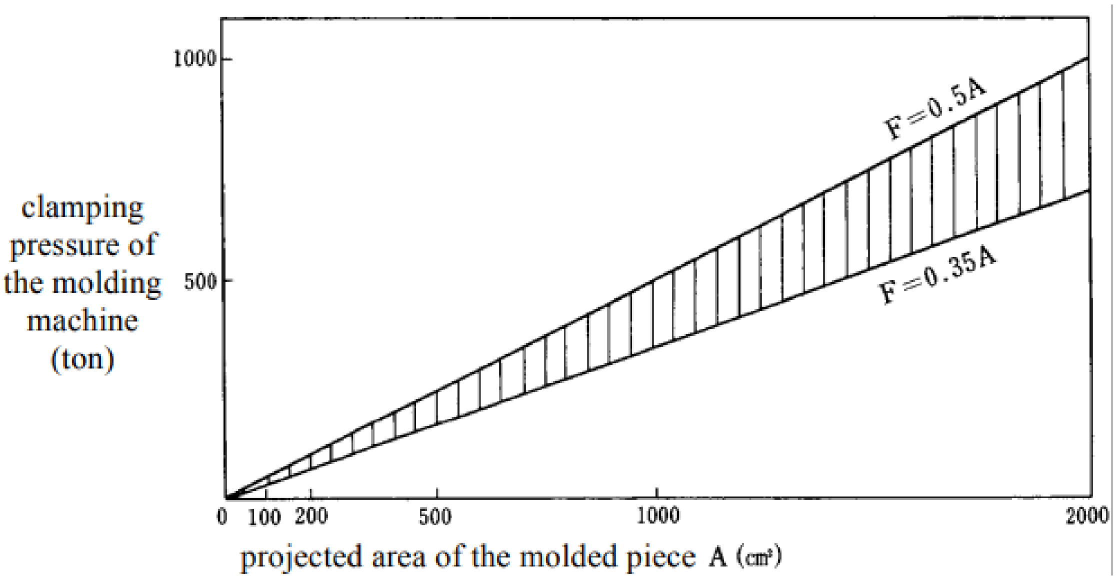
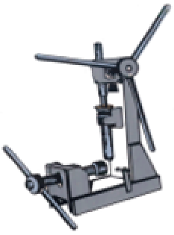


Figure 22: Selection of molding machine from clamping pressure.

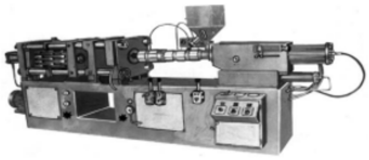
0.13.1.4 Injection molding machine types

- Hand injection molding.
- Semi-auto (Plunger type) Injection molding.
- Fully-auto (Screw type) Injection molding.
- Advanced injection molding[20].

1. Hand Injection molding



2. Semi auto injection molding (Plunger type)



(Horizontal Machine)

(Vertical Machine)

3. Fully auto injection molding: (Screw type)

(Products)

4. Advanced injection molding

(Machine)

0.13.2 Blow Molding

A heated thermoplastic extrusion tube is positioned between the two halves of an open split mold and inflated against the walls of the closed mold with air pressure.

Types of Products: Bottles, Containers, Air ducts, Panels, Portable toilets, Arm rests, tanks, gas tanks.

0.13.2.1 Types of Blow molding machine

- **Extrusion Blow molding.**
- **Injection Blow molding.**
- **Stretch Blow molding**[20].

0.13.2.1.1 Extrusion Blow molding: Plastic is melted and extruded into a hollow tube in extrusion blow molding (EBM) (a parison). This parison is then encased in a cooled metal mold. The parison is then inflated into the shape of the hollow bottle, container, or part by blowing air into it. The mold is opened and the part is evacuated after the plastic has cooled sufficiently.

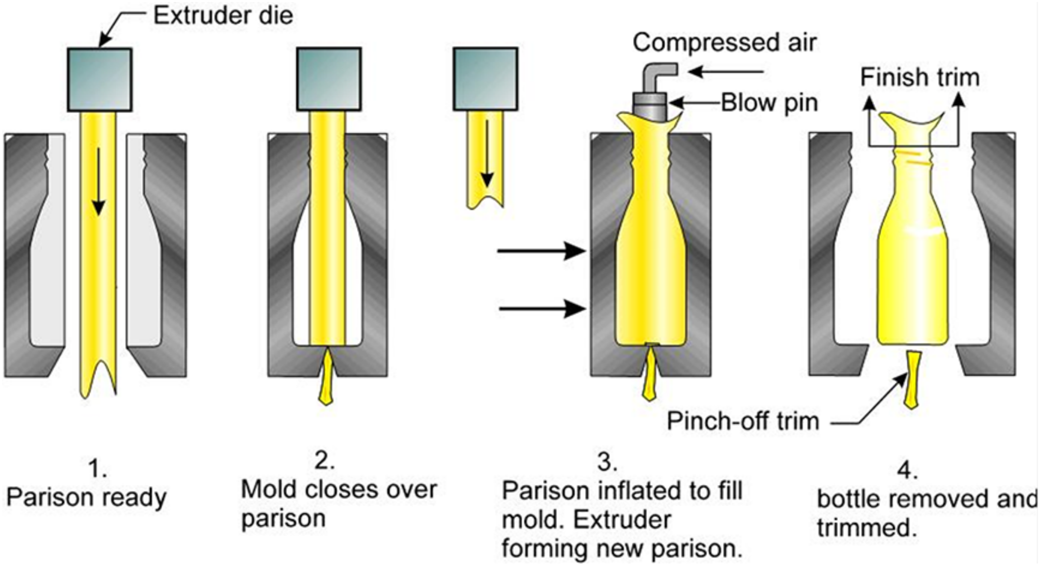


Figure 23: Extrusion Blow molding Process[3].

0.13.2.1.2 Injection Blow molding: Injection blow molding (IBM) is a technology that is used to mass-produce hollow glass and plastic products in huge quantities. Melting the plastic and shaping it into a parison (or injection) is the first step in the blow molding process. The parison is a plastic tube with a hole in one end that allows compressed air to pass through.

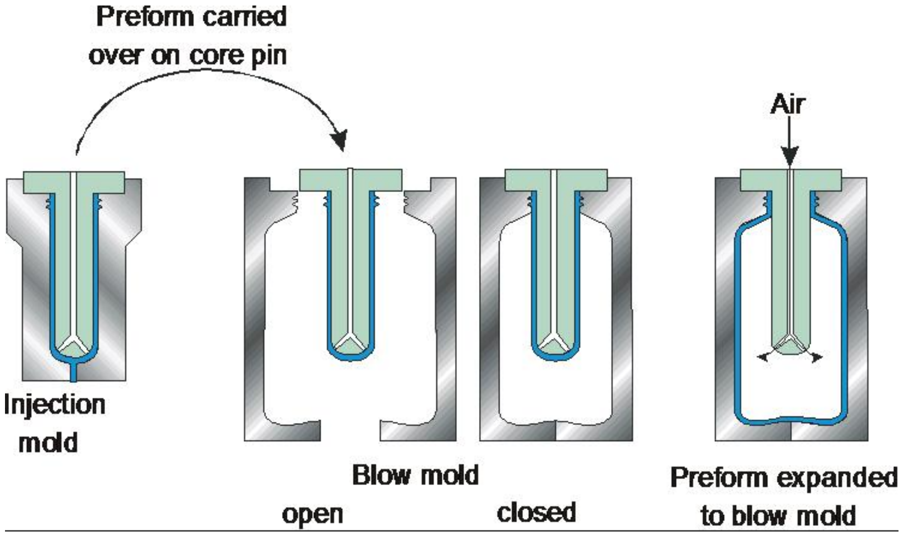


Figure 24: Injection Blow molding Process[4].

0.13.2.1.3 Stretch Blow molding: The plastic is initially formed into a "preform" using the injection molding method in the stretch blow molding (SBM) process. The preforms are heated above their glass transition temperature (usually using infrared heaters), then blown into bottles with high pressure air using metal blow molds. As part of the procedure, the preform is always stretched using a core rod. Preform manufacturing and bottle blowing are both done in the same equipment in the single-stage process.

0.13.3 Extrusion

It is a continuous procedure. A hopper feeds thermoplastic molding compound/material to a screw pump, which plasticizes it before pumping it out through the shaping orifice (die) to create the appropriate cross section.

Types of products: Films, Pipes, Strapping, Sheets, Multilayer films, Profiles etc.

0.13.4 Compression Molding

Thermoset compound is normally prepared and placed in a heated mold cavity; the mold is closed and the material flows and fills the cavity with heat and pressure.

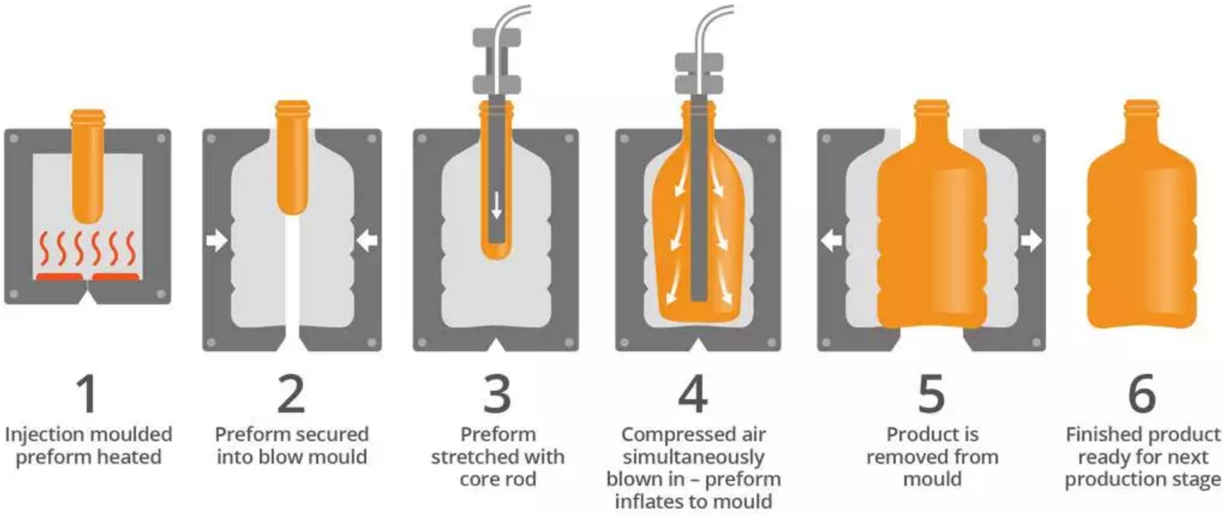


Figure 25: Stretch Blow molding Process[5].

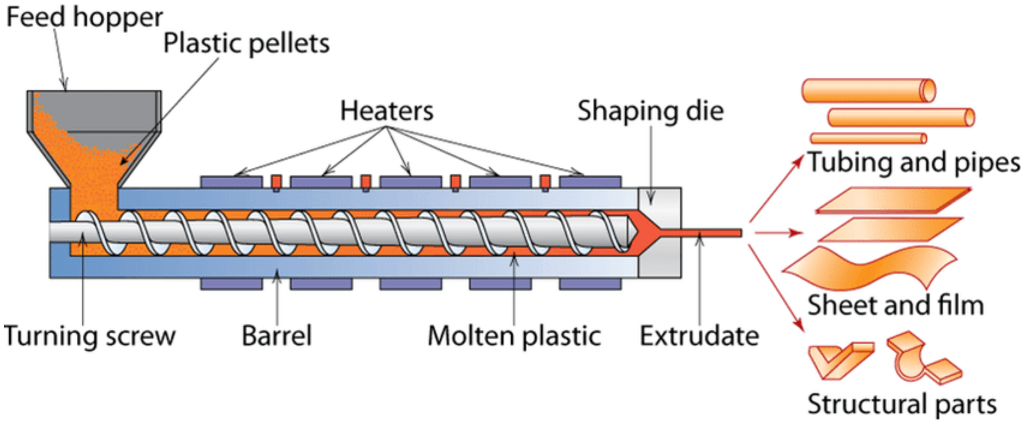


Figure 26: Extrusion Process.[6]

Heat completes the polymerization process and identifies the ejected part.

Types of Products: Plugs, sockets, handles, Engine Casing switches, cistern etc.

0.13.5 Transfer Molding

Thermoset molding compound is fed into transfer chamber where it is then heated to plasticated; it is then fed by a plunge through sprue, runners, and gates into a closed mold where it cures; mold is opened and part ejected.

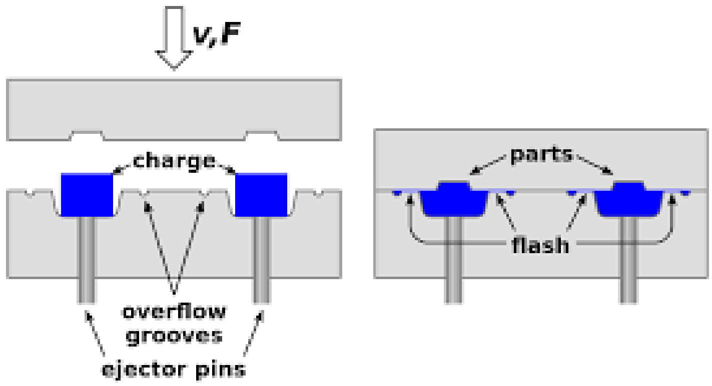


Figure 27: Compression Molding Process[7].

Types of Products: Plugs, Sockets, Handles, Engine Casing Switches, Cistern etc.

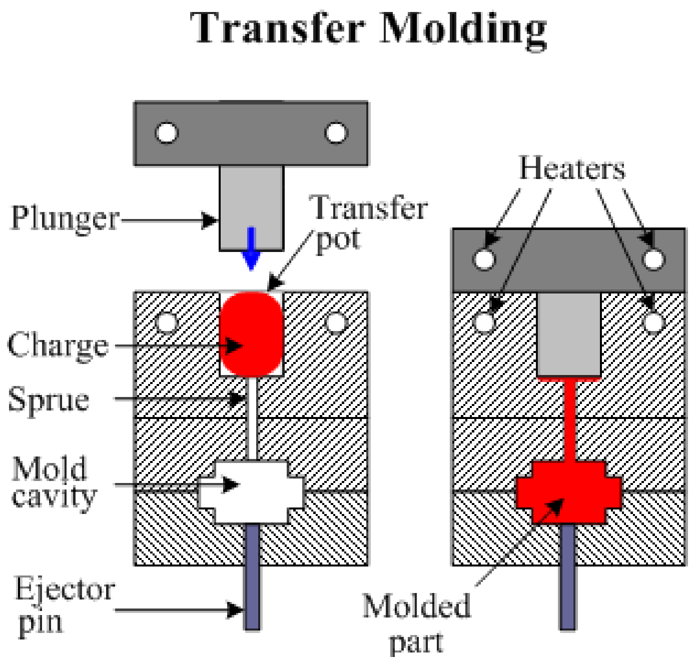


Figure 28: Transfer Molding Process[8].

0.14 Secondary Processing Technology Types

0.14.1 Rotation Molding

A predetermined amount of powdered thermoplastic material is poured into mold; mold is closed, heated, and rotated in the axis of two planes until contents have fused to the inner walls of mold; mold is then opened and part is removed.

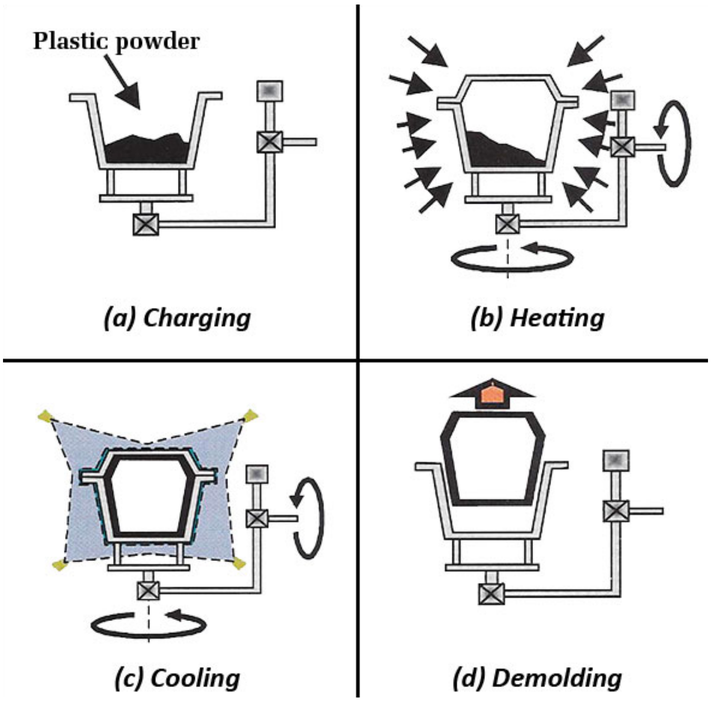


Figure 29: Rotation Molding Process[9].

0.14.2 Thermoforming Molding

Heat-softened thermoplastic sheet is positioned over male or female mold; air is evacuated between sheet and mold, forcing sheet to conform to contour of mold.

Types of Products: House wares, Ducts, Toys, Refrigerator panels, Boat windshields etc.

0.14.3 Calendaring

Dough-consistent thermoplastic mass is formed into a sheet of uniform thickness by passing it through and over a series of heated or cooled rolls. Calendars are also utilized to apply plastic covering to the backs of other materials.

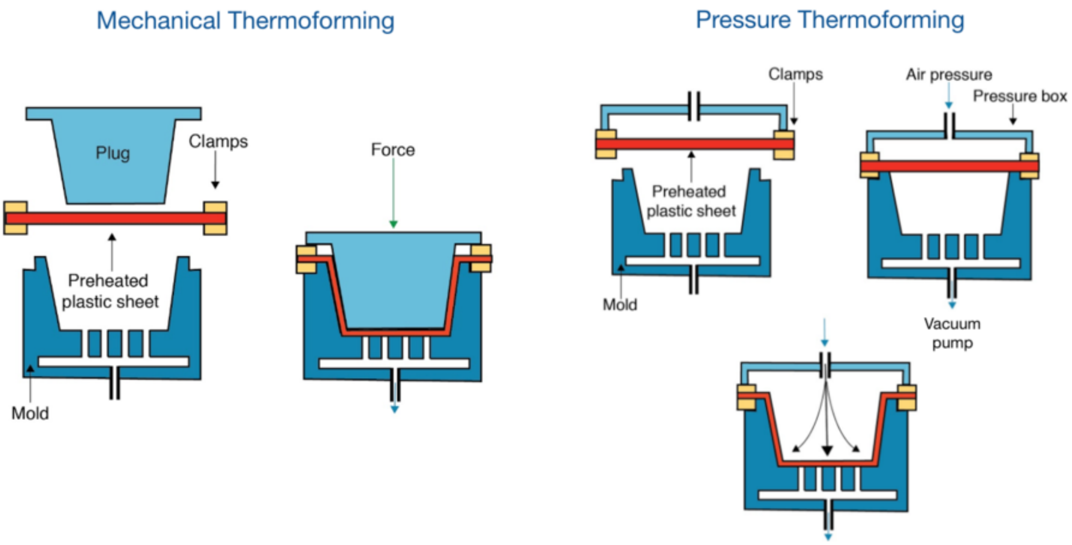


Figure 30: Thermoforming Molding Process[10].

Types of Products: Luggage, Rain wear, Tank lining, Credit cards, Trays, Helmet liner etc.

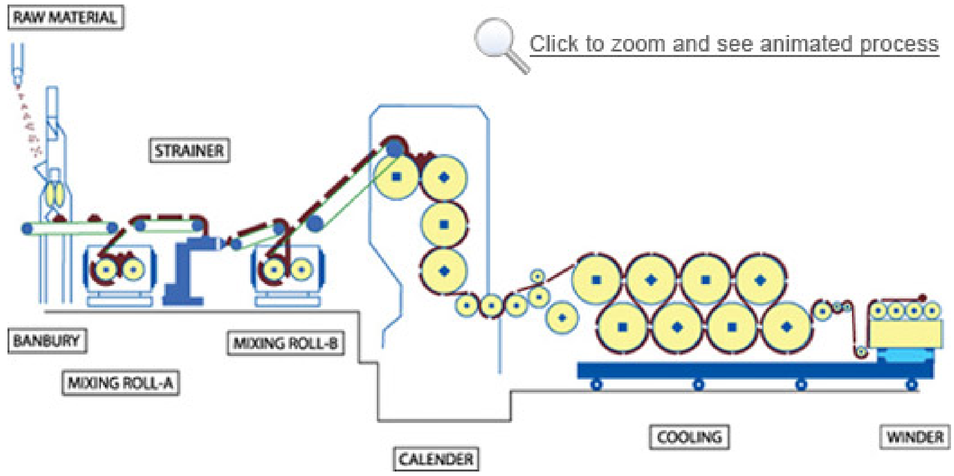


Figure 31: Calendaring Process[11].

0.14.4 Coating

Process methods both thermoplastics and thermosets are widely used in coating of numerous materials, roller coating is similar to calendaring process. Spread coating employs blade is front of roller to position resin on material.

Types of Products: Polyethylene coating, Outdoor fencing, Chemical tanks, Plastics racks, Dish washers etc.

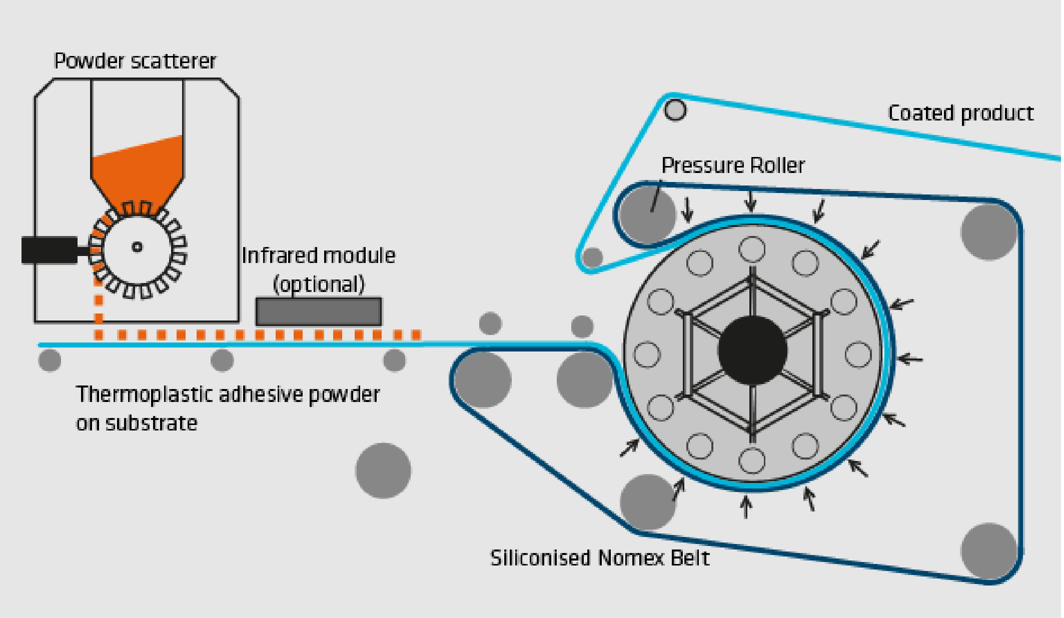


Figure 32: Coating Process[12].

0.14.5 Casting

Liquid resin which is generally thermoset except for acrylics is poured into a heated mold without pressure, cured, and taken from the mold. Cast thermoplastic films are produced via building up the material against a highly polished supporting surface.

Types of products: Rain boots, Shoes, Hollow toys, Balls, large pipes and tubes etc.

0.14.6 Fabrication

Plastic fabrication methods that are able to strengthen plastics. Compounding or Blending is the process of combining two or more types of plastic materials by melting, molding, and cooling them into different shapes.

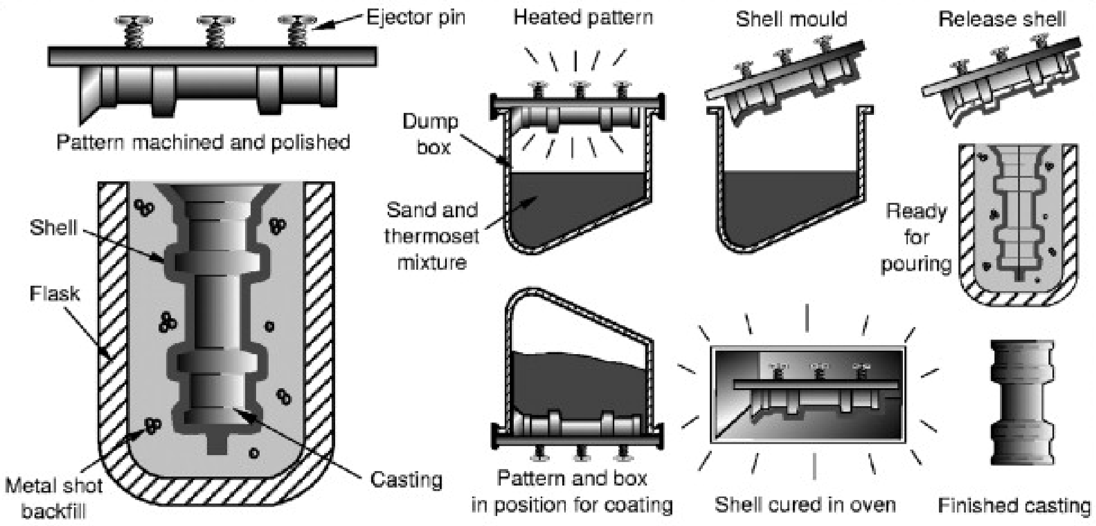


Figure 33: Casting Process[13].

0.15 Conclusion

The processes used to produce plastic parts, sometimes even aluminium alloys, with complex shapes and usually in a single operation. Plastic injection moulding is the most widely used process in this field.

In a competitive environment, it is necessary to constantly update designs. This can be achieved by changing the mold.

MOLD DESIGN

0.16 Introduction

Any molding job's success is largely determined by the skills used in the mold's design and manufacture. An injection mold is a precision device that must resist hundreds of thousands of high-pressure molding cycles while remaining durable. The added expense for a well-engineered and constructed mold can be repaid many times over in molding efficiency, reduced down time and scrap, and improved part quality[23].

There are some essential facts that must be acquired before a mold design can begin. These include figuring out how many cavities to make, what material to use to make the mold, and other information that we'll go over in this part. To collect all of the essential knowledge, it is likely that more than one area of expertise will be required. This may necessitate enlisting the help of specialists such as material engineers and financial experts. However, depending on whatever knowledge and information is available at the time, we can make some basic assumptions[24].

0.17 Mold Basics

Molds are made up of two major components: the cavity and the core. The part's major internal surfaces are formed by the core. The cavity is responsible for the majority of the external surfaces. As the mold opens, the core and cavity usually separate, allowing the item to be removed. This mold separation occurs along the interface known as the parting line.The parting line might be stepped or inclined to accommodate irregular part features, or it can be in one plane matching to a major geometric feature such as the part top, bottom, or center-line.

Undercuts that cannot be avoided via reasonable adjustments in the parting line require mechanisms in the mold to disengage the undercut prior to ejection[23].

0.18 Types of Molds

0.18.1 Two-Plate Mold

The most common mold configuration is a two-plate mold, which consists of two mold parts that open along a single parting line (see figure 3.1). Material can enter the mold cavity directly via a sprue gate, or indirectly through a runner system that delivers the material to the desired locations along the parting line. The movable mold half usually contains a part-ejection mechanism linked to a hydraulic cylinder operated from the main press controller.

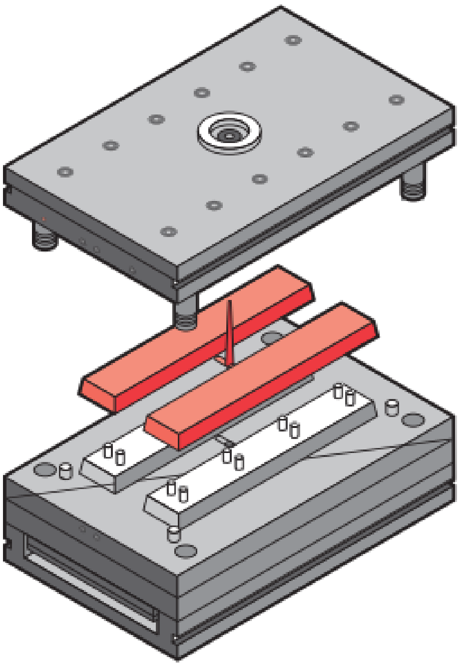


Figure 34: A conventional two-plate mold with two cavities.

0.18.2 Three-Plate Mold

Instead of one major opening, the three-plate mold arrangement has two significant openings. The mold-opening sequence for a typical three-plate mold is shown in Figures 3.2 and 3.3. The mold-opening sequence is usually controlled by a connection system between the three principal mold plates. Breaking the pinpoint gates and separating the components from the cavity side of the mold, the mold first opens at the major parting line. The mold then separates at the runner plate, making it easier to remove the runner system. Finally, the runner is separated from the retaining pins by a plate, and the components and runner are ejected from the mold.

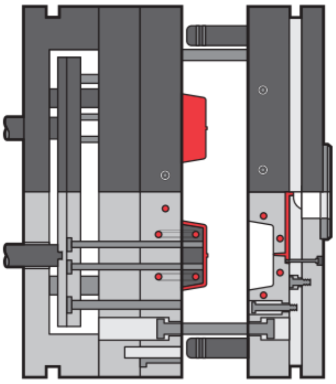


Figure 35: Schematic of a two-cavity, three-plate mold with cutaway view showing first stage of opening.

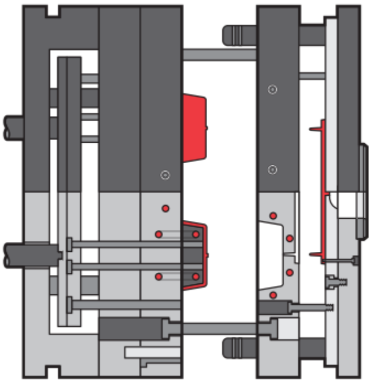


Figure 36: Schematic of a two-cavity, three-plate mold with cutaway view showing second stage of opening.

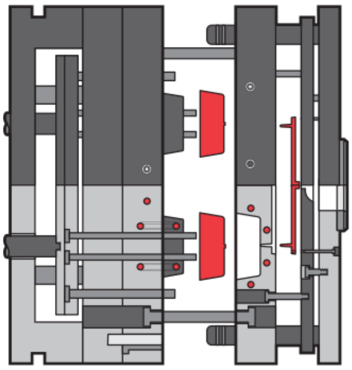


Figure 37: Schematic of a two-cavity, three-plate mold with cutaway view showing final opening phase and stripper plate in forward position.

Three-plate molds, unlike traditional two-plate molds, may gate directly into interior surface areas away from the outer edge of parts, which is advantageous for center-gated items like cups or big parts with several gates over a surface. Additional mold complexity and large runners that can generate excessive regrind are disadvantages. Furthermore, the small pinpoint gates needed for clean automatic de-gating can cause severe shear, resulting in material degradation, gate blemish, and packing issues. Three plate molds are not advised for shear-sensitive materials such as Cadon SMA or materials with shear-sensitive colorants or flame retardants due to the high shear rates created in the tapered runner drops and pinpoint gates.

The stack mold, another arrangement, decreases the clamp force required by multi-cavity molds. Multiple cavities are typically aligned on a single parting line, and the required clamp force is the sum of each cavity's clamp plus the runner system. In stack molds, cavities are formed by stacking two or more separating lines. Because the injection forces acting on the plate separating parting lines cancel out, the clamp force is the same as if only one parting line were present. Stack molds produce more parts per cycle than would be possible in a molding press of the same size.

0.19 Mold Bases and Cavities

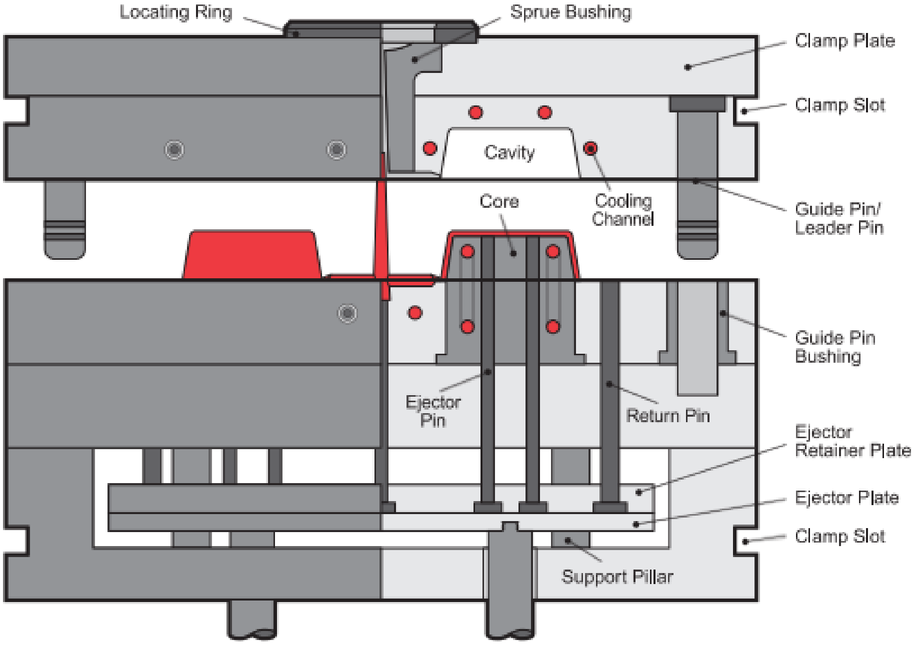


Figure 38: Components of a standard two-plate mold base with two cavities.

The majority of an injection mold's mass is made up of the mold foundation. Most molding needs can be met with standard off-the-shelf mold bases. A locating ring (see figure 3.5) and provisions for a sprue bushing in the stationary half of the mold and an ejector assembly in the moving half are typical mold base features. Clamp slots are included in both sections to secure the mold in the press. Holes in the movable half accommodate bars that connect the press ejection mechanism to the mold's ejector plate.

The mold halves are aligned by leader pins emerging from the corners of the stationary portion. When the ejection mechanism is in the forward (eject) position, return pins connected to the ejector plate corners protrude from the mold face. The return pins retract the ejector plate (if not already retracted) as the mold shuts, preparing for the next cycle.

Mold cavities, or core and cavity sets, can be cut directly into the mold plates, placed in parts into the mold base, or inserted as full cavity units in the mold. For big components and/or parts with basic geometries, cutting cavities directly into the mold base can be the most cost-effective option. It's critical to choose the mold base steel carefully when doing so. Standard mold base steels may not provide the physical qualities required for heavy-wear areas or important steel-to-steel contact points. In certain places, it's critical to employ inserts made of appropriate materials.

When the cavity is assembled in the mold base, different metals can be used for the various cavity components, improving the mold's durability and performance. It also makes it easier and faster to replace old or damaged cavity components. Assembling the cavities from pieces can also make component fabrication easier. Mold-base cavity assemblies have a number of disadvantages, including a high initial mold cost, inefficient mold cooling, and potential tolerance buildup issues with the cavity components.

Many of the benefits of mold-base cavity assemblies are also available in cavity units. Wear and damage cavities are simply replaced since many cavity components are face-mounted in the mold base for rapid removal. Some mold bases are made to accept conventional cavity-insert units so that parts can be changed quickly while the mold is still in the molding press. These cavity units usually have their own cooling circuits and ejector mechanisms, which connect to the mold-base ejector system automatically.

0.20 Molding Undercuts

Undercuts, which prevent straight ejection at the separating line, tend to add to mold complexity and raise mold construction and maintenance costs. Avoid undercuts whenever possible by redesigning the part. Problematic undercuts in the mold can typically be eliminated with minor part design adjustments. Through holes, for example, can provide access to the underside of features that would otherwise be undercuts (see figure 3.6). Similarly, rather than using a side-action mechanism, simple adjustments allow the mold to produce a hole in the sidewall with bypass steel.

Undercut elements that cannot be avoided through redesign necessitate ejection devices in the mold. Side-action slides, lifter rails, jiggler pins, collapsible cores, and unscrewing mechanisms are examples of these devices. The rest of this section delves into these possibilities.

Cam pins or hydraulic (or pneumatic) cylinders are used in side-action slides to retract parts of the mold before the ejection. As the mold opens, cam-pin-driven slides retract (see figure 3.7). The cam pins return the slides to their original position as the mold shuts, ready for the next injection cycle. Slides powered by hydraulic or pneumatic cylinders can actuate at any point during the molding cycle, which is useful in situations where the slides must actuate before the mold opens or closes.

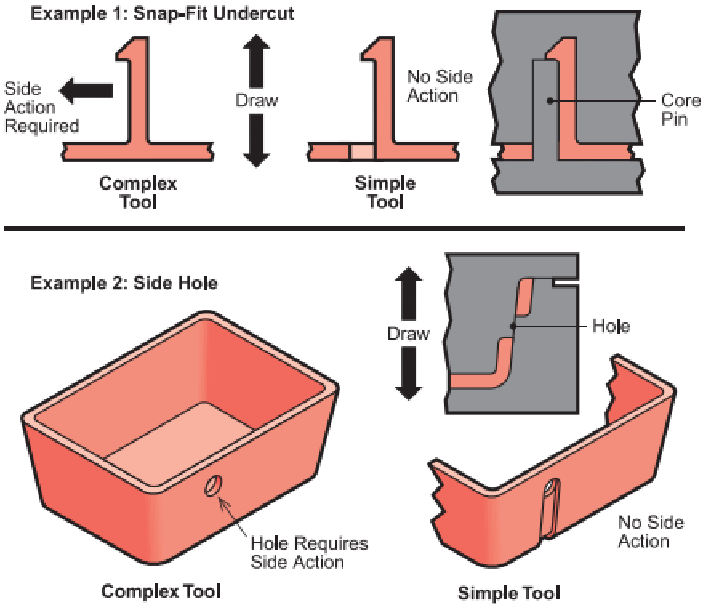


Figure 39: Undercut Alternatives (Simple/complex part design for undercuts).

Spring-loaded lifters (see figure 3.8) or lifter rails attached to the ejector system can commonly generate shallow undercuts. During mold opening or ejection, these lifters travel with the component at an angle until the lifter clears the undercut in the part. The "jiggler" pin (see figure 3.9) is a variation on this concept, with angled surfaces that direct the pin away from the undercut during

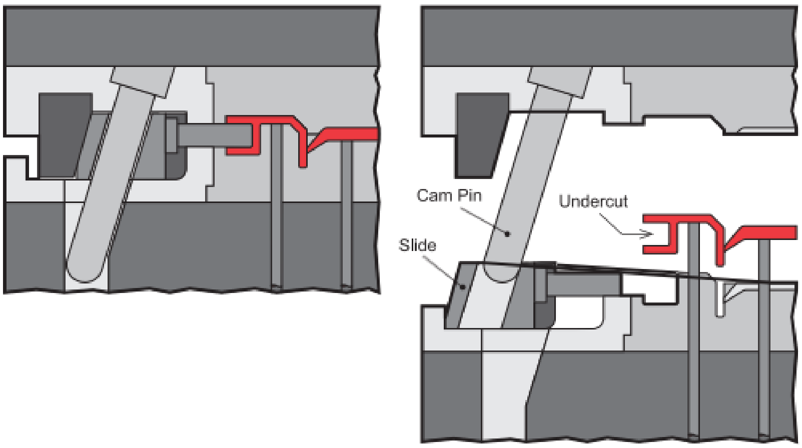


Figure 40: Side-Action Slide (The cam pin retracts the slide during mold opening).

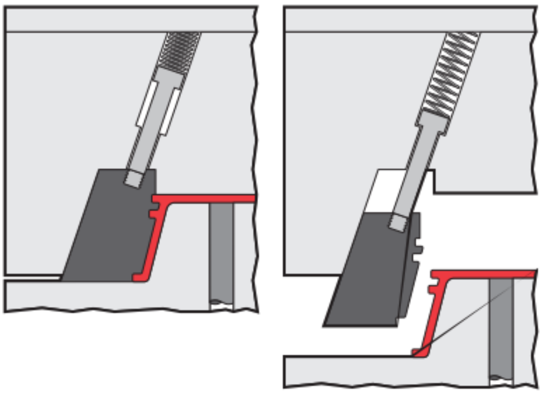


Figure 41: Lifter (Typical spring-loaded lifter mechanism).

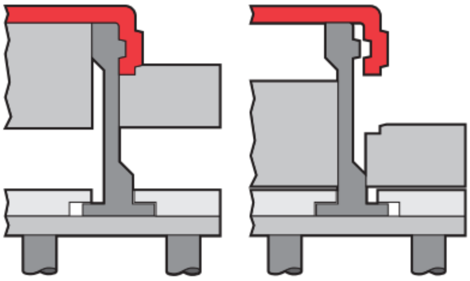


Figure 42: Jiggler Pin (Angled surfaces slide the jiggler pin to clear the undercut during ejection).

ejection and then return it to the molding position when the ejector system retracts.

Internal threads, dimples, slots, and grooves on the inside of holes or caps may need the use of collapsible cores. These complex cores are constructed in pieces that collapse toward the center once the mold is opened (see figure 3.10). These specialized cores are often adjusted to generate the appropriate undercut shape and are available in a number of standard sizes from various mold component manufacturers. The minimal size of a collapsible core is limited by the quantity and complexity of individual core components.
Internal threads are frequently created using unscrewing devices. Rackand-pinion devices activated by mold opening, motors, or hydraulic cylinders, as well as motor-driven gear and chain mechanisms, can all be used to rotate the threaded cores.

Slides, cams, collapsible cores, and unscrewing mechanisms all add to the mold's cost and complexity, as well as the cost of mold maintenance. Using clever part design, we can typically get rid of those pesky undercuts. Some undercuts are most cost-effectively generated as secondary processes, especially if they can be automated or completed during the press cycle.

0.21 Part Ejection

Ejector systems are usually built inside the movable part of molds. These systems are activated by the molding press's ejection unit. The press controller may control the timing, speed, and length of the ejection stroke using rods connecting the press-ejector mechanism to an ejector plate in the mold. Reverse

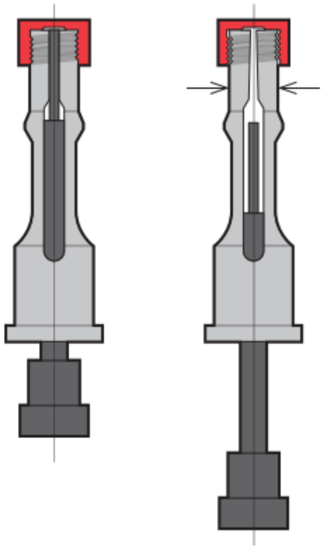


Figure 43: Collapsible Core (Standard-style collapsible core pin in expanded and contracted position).

injection molds use independent ejection mechanisms controlled by springs or hydraulic cylinders to eject pieces from the stationary side of the mold. Direct injection onto the inner or back surface of cosmetic parts is made easier with this design. Reverse-injection molds are more expensive because of their added complexity.

Specialized ejection components, such as knockout (KO) pins, KO sleeves, or stripper plates, project from the mold ejector plate to the part surface where they push the part out of the mold (see figures 5.5 through 3.13). These topics are discussed in this section.

The common round knockout pin is a simple and cost-effective way to eject parts. These low-cost, off-the-shelf items resist wear and fracture thanks to their high surface hardness and strong core. From a wide range of common sizes, the mold maker chooses the desired diameter and shank length and machines it to fit. To avoid flash, the fit of the ejector pin into the round ejector hole must be very tight. Ejector blades, which are KO pins with a rectangular cross section, work similarly to round pins but are more difficult to fit and maintain. They're typically employed on the borders of ribs or walls that are too thin for traditional round pins.

KO pins are frequently extended to the mold face's parallel surfaces. Consider adding grooves to the part design to avoid pin deflection if KO pins are pushed against angled surfaces (see figure 3.14). KO pins that extend to thin walls and

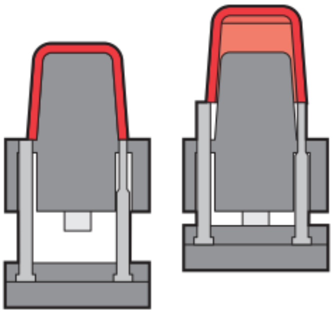


Figure 44: Ejector Pins and Blades.

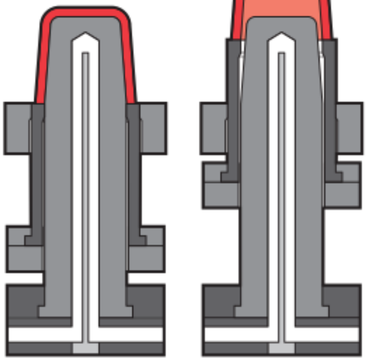


Figure 45: Ejector Sleeves.

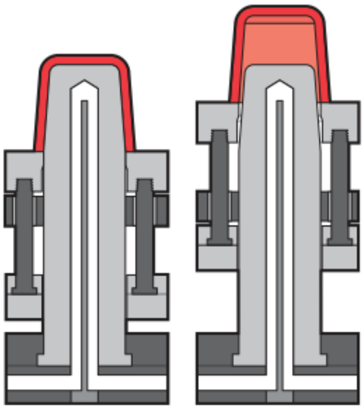


Figure 46: Stripper Plate.

edges can be stepped or positioned such that just a section of the pin contacts the molded part. This eliminates the need for small-diameter KO pins, which are more difficult to maintain and can deflect or bend.

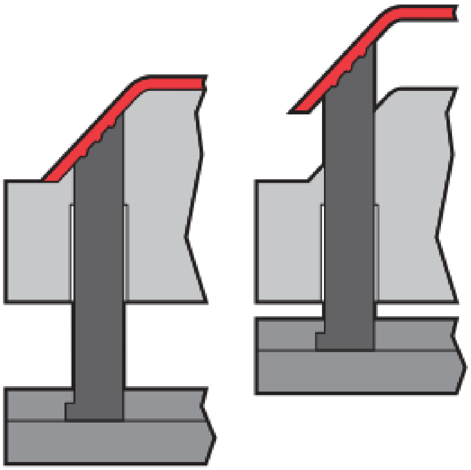


Figure 47: Angled Ejector Pin.

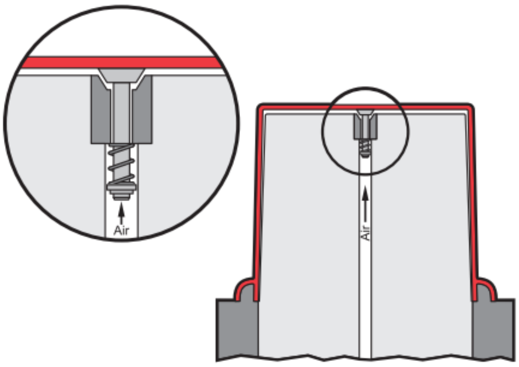


Figure 48: Air-Poppet Valve.

Where the pin hits the part, KO pins create witness marks, little indentations or circles that may be undesirable on cosmetic surfaces. Additionally, if the part is difficult to eject or the ejector area is too small, they can read-through to the opposite surface.

The quantity of ejector area required is determined by a number of elements, including part shape, mold finish, material-release properties, and part temperature at the time of ejection. Thin-walled parts require larger ejectors and a larger ejector area than comparable parts with thicker walls to avoid damage during ejection.

Draw polishing the mold steel in the direction of ejection generally helps ejection. Also, adding a generous amount of mold draft helps ejection. Draft refers to the slight angle or taper added to part features to ease part ejection. Most LANXESS materials require at least one degree of draft for easy ejection. Lustran SAN resins require at least two degrees of draft.

Internal mold release materials can minimize the necessary ejection force and solve various ejection issues. While spray mold releases are generally useful as a temporary cure, they can lengthen the molding cycle and cause cosmetic issues.

If a vacuum arises between the part and the mold during ejection, ejection problems can occur. This problem usually arises in parts with a deep core and a closed bottom. Off-the-shelf mold components like air-poppet valves (see figure 3.15) can help solve the issue. During ejection, air-poppet valves relieve the vacuum and deliver pressurized air between the part and the mold surface.

0.22 Runners, Gates, and Venting

0.22.1 Overview

Molten plastic is pushed into a closed mold during the injection molding process. To begin, the trapped air in the closed mold must be released. The plastic must then flow through the mold and into the cavity image, where it will solidify before being ejected. A runner system, a rating system, and correct venting are the three essential components required for this to work successfully. We'll go over each of these components in detail in the next section, as well as some of the variations available and the requirements for each. Let's start with the sprue bushing [24].

0.22.2 Role of The Sprue Bushing

The sprue bushing is a component of the mold that serves as a connection between the injection molding machine's cylinder nozzle and the mold's runner system. The sprue is the plastic that forms within the sprue bushing during the molding process. Because each sprue bushing is particularly special to a particular mold design and product design, certain rules are applied to its design.

Figure 3.16 shows how to properly measure a sprue bushing. On the face of the large head end, the bushing has a spherical radius. This radius must be the same as, or slightly larger than, the radius of the nozzle on the molding machine in order to produce a seal. Flash will arise if the sprue bushing radius is too tiny. Every cycle, the sprue will stick in the bushing instead of pulling free as it should (some mold designs, notably for insert molding, do not require this radius, and are in fact flat). For the same reasons, the lower diameter of the interior tapered hole must be equal to or slightly greater than the machine nozzle's mating hole. The smaller hole should be as small as possible while yet being large enough to suit the specifications of the specific material being molded. If necessary, the nozzle diameter can be adjusted to match the sprue bushing diameter. Although any dimension is feasible, the most typical diameters for this interface are roughly 2, 4, 5.5, and 7 mm.

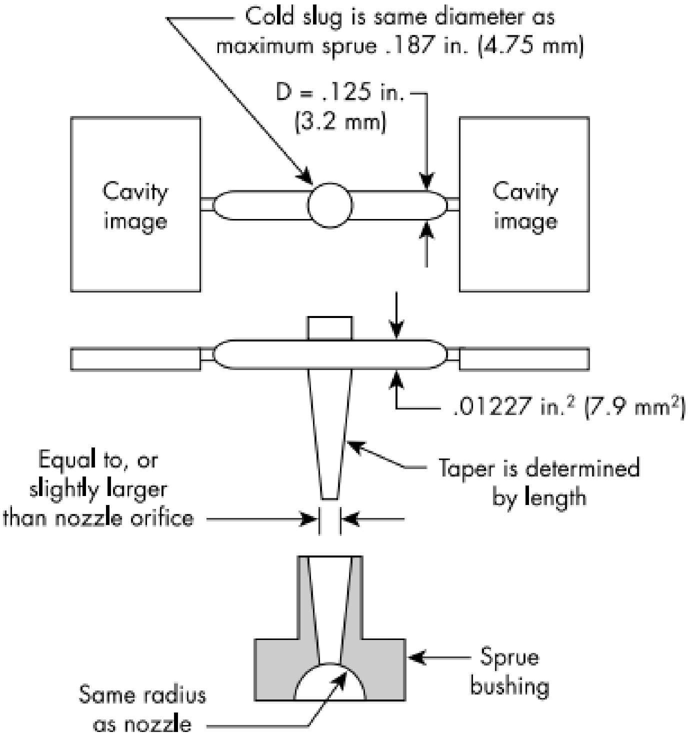


Figure 49: Sprue bushing dimensions.

The overall length required determines the taper of the inside sprue-bushing hole and should be kept at a minimum.

0.22.3 Runner Systems

Runner systems direct molten plastic from the sprue bushing to the cavity images that create the molded object once it enters the mold. The plastic ”runs” along the channels machined into the mold base, hence the name ”runners.” Runners are typically reserved for multi-cavity molds. A single-cavity mold, in theory, does not require a runner because the plastic is injected straight into the single cavity, which is an ideal circumstance. Some single-cavity molds, on the other hand, may be positioned in an off-center point in the mold, necessitating the use of a runner to transport material from the sprue bushing to that spot. This is especially true in “family” or “universal” molds, which may only have one (or a few) cavities flowing at any given time.

0.22.3.1 Surface Runners

Surface runners are usually cut into a block called a runner block inserted into the A and B halves of the mold, as shown in Figure 3.17. This is done to allow adjustments and changes to the shut-off and dimensional functions of the basic runner system, without affecting the entire mold base or cavity layout.

A full-round runner is ideal due to the proper construction of the surface runner. This is because a circular cross section exerts equal pressure on the plastic molecules in all directions, but a noncircular cross section exerts unequal pressure. This is seen in Figure 3.18, which compares a full round runner design against a conventional trapezoid runner shape. When the molten plastic flows through the runner toward the cavity, the runner design on the left side of Figure 3.18 minimizes the amount of molecular distortion caused. The trapezoidal cross section on the right induces molecular deformation, which causes stresses in the material to build up. These stressed molecules are transported into the cavity, where their stressed condition solidifies.

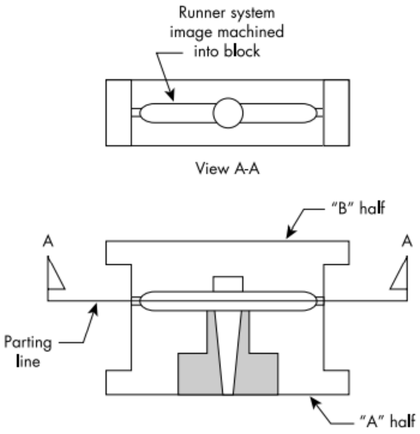


Figure 50: Typical runner block concept.

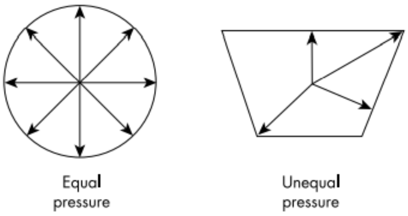


Figure 51: Comparing round runner cross section with trapezoid.

The diameter of the suitable runner cross section is determined by the type of plastic being molded. Larger diameters are required for high-viscosity (stiff) materials than for low-viscosity polymers. Normal runner diameters for various popular materials are shown in Table 3.1. As illustrated, the wider the runner diameter at the start, the longer the flow path the plastic must go along. The required increase in diameter is due to the fact that the plastic cools as it enters the runner system and begins to solidify as it advances near the gate. As a result, the runner must be made larger to allow the molten plastic to move quicker down the course before solidifying. The greater the runner diameter, however, the longer it will take for it to cool down and be ejected from the mold. As a result, when evaluating the entire cycle time for molding the product, we must carefully consider the bigger diameter need.

The runner diameter is typically larger than the thickest section of the part to be molded. As a result of affording the runner enough time to cool down, the cycle time is excessively long. While it does not have to cool entirely to become solid, it must cool long enough to become slightly rigid and expelled effectively. In addition, the beginning diameter must be increased by a factor of 20% every time the runner must make a right-angled turn. This is done to compensate for the pressure decrease that occurs when the material is driven down the runner path, as shown in Figure. 3.19.

The ideal runner path takes material on a straight line from the sprue bushing to the cavity image. However, due to waterline interference, bolt hole locations, and ejector pin layouts, a straight line may not be possible. In those cases, a runner is usually designed to take right-angled turns as shown in Figure 3.19.

Table 10: Runner Diameters for Common Materials.

	Running Length		
	(76.2 mm)	(152.4 mm)	(254 mm)
	Runner Diameter		
Material	(mm)	(mm)	(mm)
ABS	(2.4)	(2.8)	(3.9)
Acetal	(1.6)	(2.4)	(3.1)
Acrylic	(3.1)	(3.9)	(4.7)
Cellulose acetate	(2.4)	(2.8)	(3.9)
Cellulose acetate butyrate	(2.4)	(2.8)	(3.1)
Ionomer	(1.6)	(2.4)	(3.1)
Nylon 6/6	(1.6)	(1.9)	(2.4)
Polycarbonate	(3.1)	(3.9)	(5.1)
Polyethylene	(1.6)	(2.4)	(3.1)
Polypropylene	(1.6)	(2.4)	(3.1)
Polyphenylene oxide	(3.1)	(3.9)	(5.1)
Polyphenylene sulfide	(3.1)	(3.9)	(5.1)
Polysulfone	(3.9)	(4.7)	(5.5)
Polystyrene	(2.4)	(2.8)	(3.1)
Rigid PVC	(3.1)	(4.7)	(6.3)

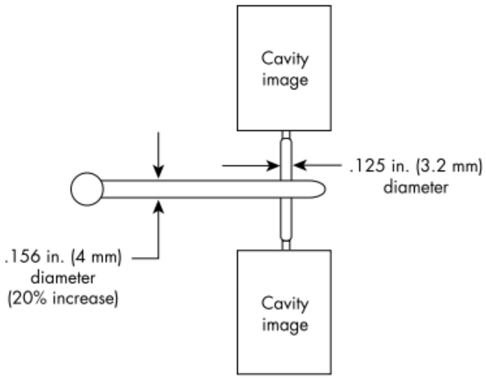


Figure 52: Increasing runner diameter by 20%.

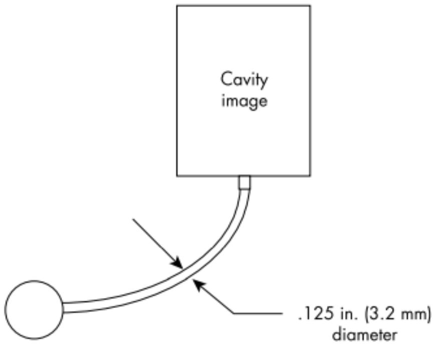


Figure 53: Sweeping radius runner path.

A better way, if possible, is to use a sweeping radius design as shown in Figure 3.20. This approach eliminates the pressure drop caused by sharp turns and the need to increase the runner diameter by 20% for each of those turns. The runner diameter is still determined by measuring its length between the cavity image and the sprue, but there are no other factors to apply. This minimizes the runner size and helps lower the overall molding cycle. Steps should be taken in the early mold design stages to incorporate this concept if possible.

When laying out the cavity images of a multicavity mold in the first place, thought should be given to creating a "wagon-wheel" design, with runners taking the place of the wagon-wheel spokes, as shown in Figure 3.21. This is the best design because it uses the straight-line runner approach and minimizes the travel the plastic flow fronts must make to get to the cavity images. It also keeps the runner diameter at a minimum, thus reducing overall cycle times.

0.22.3.2 Insulated Runners

The long cycle times associated with standard surface runners (due to their thickness) caused molders to try to find ways to overcome the condition. One such effort resulted in what is known as an insulated runner. In this system, the runner is not machined into the surfaces of the A and B plates, but rather between the A plate and an in-between plate known as the X plate, or third plate.

Figure 3.22 shows the insulated runner system concept. The A plate and X plates are bolted together. The runner is machined between those two plates, and drop gates are used to bring the molten plastic to the cavity image that is

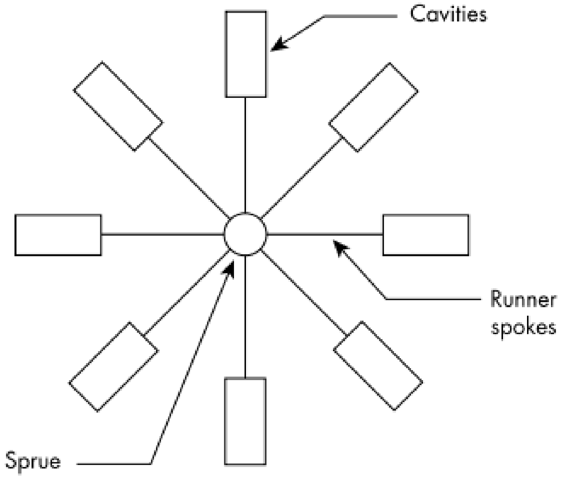


Figure 54: Wagon-wheel runner layout.

located between the B and X plates. When the mold opens, the runner is trapped between the A and X plates. The molded part is ejected and the mold closes for the next cycle. The runner thickness is large and the runner begins to solidify from the outside in, but the center core of the runner stays hot and molten. When the next cycle starts, the plastic is injected through the center core of the runner and the still-molten material fills the mold. The result is that the center core area has been insulated from the cooling effect of the mold steel because of the very thick skin forming on the outside of the runner diameter. This process can continue as long as the center core of the runner stays hot enough to keep the plastic molten. A slight interruption of the cycle may be enough to cause the center core to cool and solidify. When this happens, the mold must be dismantled and the solidified plastic must be machined out of the runner system. Then, the runner must be polished and the mold can be reassembled for another molding attempt.

The major advantage to the insulated runner system is that the runner does not have to be included in the calculation of cycle times. The cooling portion of the molding cycle only applies to the molded part and the overall cycle can be much shorter than if runner diameter thickness were included. A secondary advantage is that there is no runner to dispose of after molding. That eliminates the need to use or inventory regrind (recycled process resin) that would normally be generated by a standard runner system.

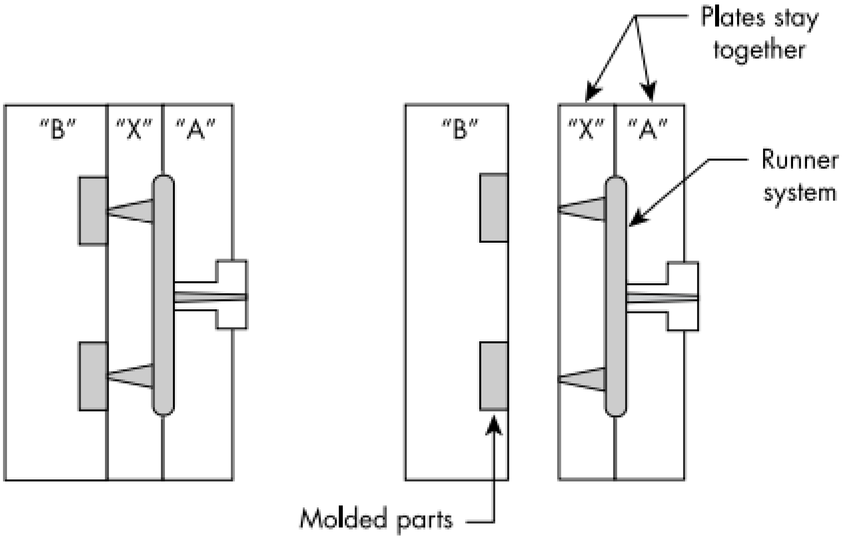


Figure 55: Basic insulated runner concept.

0.22.3.3 Hot Runners

The inefficiencies and uncertainties associated with insulated runner systems led the way to creation of a hot runner system. The purposes for both systems are the same: eliminate the surface runner system and reduce overall cycle times. However, the hot runner system uses individual cartridge heaters to keep the plastic molten in the runner path and does not rely on the insulating properties of the runner skin. In fact, as Figure 3.23 shows, the runner skin never does solidify. However, the entire hot runner system is insulated from the rest of the mold to keep the basic mold temperature comparatively low while the runner temperature can be high.

The hot runner concept has allowed a remarkable thing to happen. In effect, the nozzle of the molding machine has been moved directly to the cavity image, thereby eliminating the stress and control conditions normally found in the use of a standard surface runner system. This, in turn, results in some major advantages. Cycle times are shorter by an average of 25% because there is no runner to include in the calculation, defects are minimized as a result of minimizing stresses, and scrap is reduced because there is no runner to dispose of after molding.

Hot runner systems are commercially available, but must be engineered for each mold and material on a specific basis. They are expensive, with the average cost in the area of 25,000 dollars, but they pay for themselves quickly due to faster cycle times and less scrap. Even heat-sensitive materials, such as polyvinyl chloride (PVC), can be run in modern hot runner systems. If the investment can be justified, hot runner systems should be considered for every mold.

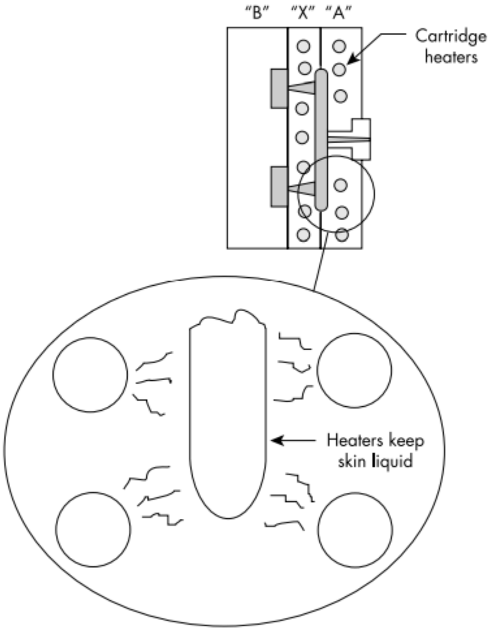


Figure 56: Basic hot runner concept

0.22.3.4 Runners for Multi-cavity Molds

Runners for multi-cavity molds require special attention. Runners for family molds, molds producing different parts of an assembly in the same shot, should be designed so that all parts finish filling at the same time. This reduces overpacking and/or flash formation in the cavities that fill first, leading to less shrinkage variation and fewer part-quality problems. Consider computerized mold-filling analysis to adjust gate locations and/or runner section lengths and diameters to achieve balanced flow to each cavity (see figure 3.24).

The same computer techniques balance flow within multi-gated parts. Molds producing multiples of the same part should also provide balanced flow to the ends of each cavity. Naturally balanced runners provide an equal flow distance from the press nozzle to the gate on each cavity. Spoked-runner designs (see figure 3.25) work well for tight clusters of small cavities. However, they become less efficient as cavity spacing increases because of cavity number or size[23].

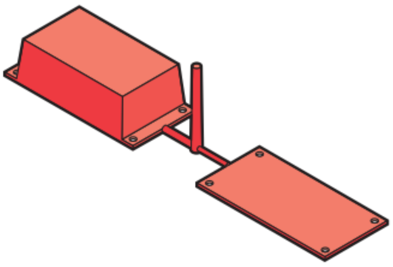


Figure 57: Family Mold.

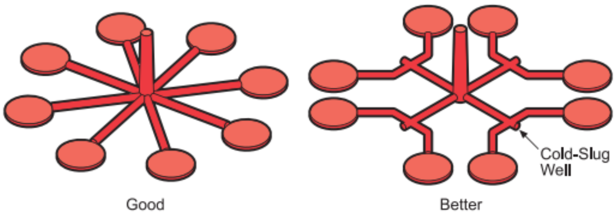


Figure 58: Spoked Runners.

Often, it makes more sense to orient cavities in rows rather than circles. Rows of cavities generally have branched runners consisting of a primary main feed channel and a network of secondary or tertiary runners to feed each cavity. To be naturally balanced, the flow path to each cavity must be of equal length and make the same number and type of turns and splits. This generally limits cavity number to an integer power of two — 2, 4, 8, 16, 32, etc. — as shown in figure 3.26. Generally, the runner diameter decreases after each split in response to the decreased number of cavities sharing that runner segment. Assuming a constant flow rate feeding the mold, the flow-front velocity in the cavity halves after each split. The molding press flow-rate performance may limit the number of cavities that can be simultaneously molded if the press cannot maintain an adequate flow-front velocity.

Artificially balanced runners provide balanced filling and can greatly reduce runner volume. Artificially balanced designs usually adjust runner-segment diameters to compensate for differences in runner flow length. For instance, in ladder runners, the most common artificially balanced runner design, a primary runner feeds two rows of cavities through equal-length secondary runners. The diameters of these secondary runners are made progressively smaller for the cavities with shortest runner flow distance (see figure 3.27). These designs require enough secondary runner length to flow balance using reasonable runner diameters.

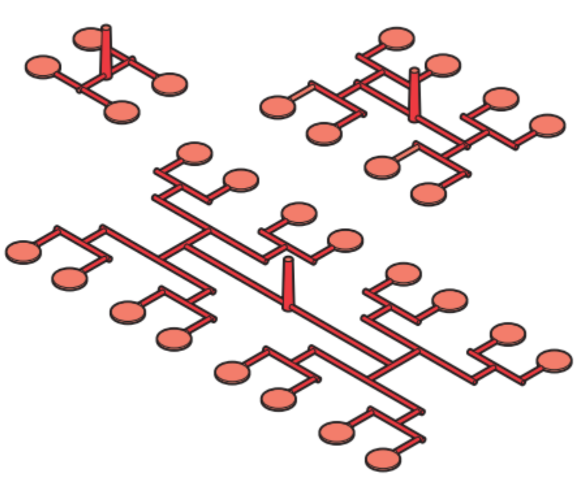


Figure 59: Naturally Balanced Runners.

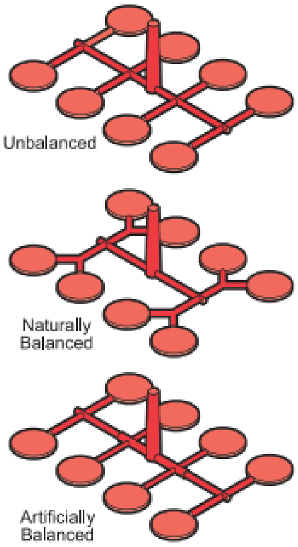


Figure 60: Runner Balancing.

0.22.4 Gating Methods and Designs

After molten material travels through the runner path, there are many ways to get it into the cavity image. The runner itself can continue directly into the cavity image, and, in fact, that was the method used in the very early days of our industry. However, that made it difficult to remove the molded part from the runner system, due to the mass of material at the junction formed by the runner and part. Therefore, a reduction in runner size was created at this junction. This reduction became known as the gate through which material must travel to enter the cavity image. The reduced area allowed easier removal of the molded part from the runner. In addition, this reduced area created a slight amount of friction that caused the plastic going through it to heat up. This extended the flow of the plastic material and made it easier to fill the cavity image. Today, the gate is used for two purposes: to control the flow of molten material entering the cavity image and to ease separation of the molded part from the runner system[24].

Early gate designs were primitive and not scientific in nature. Most gate sizes were estimated and alterations were made after running the mold to see the results. However, as more materials became available for molding (some of which were heat-sensitive), and as molding machines became more controllable, gate design became an important issue. It was determined that specific materials could have wide molding parameters, while others needed very tight control, especially in the area of gate design. A gate that was too "tight" might cause thermal degradation and stress to be molded into a part. However, a gate that was too large might result in excessive cycle times and difficulty in removing molded parts from runners. Gate design, then, became a science in itself and resulted in a variety of shapes and concepts, some of which we will investigate in the following section.

0.22.4.1 Determining Gate Location and Number

First, determine where the part should be gated and how many gates might be needed. A statement can be made that says, “any part can be filled with a single gate.” While this is true, it might be better to add gates to overcome some of the problems associated with just a single gate, depending on product design and final requirements of the molded part. For instance, if absolute flatness is required of a molded part that has varying wall thickness, a single gate may not be able to produce such a part due to internal distortions caused by the flow of material. Other factors may also affect the number and location of the gates.

Most parts are not designed to have uniform wall thickness throughout the part. Because of this, the ideal spot for gating is in the thickest area first. Then, the material can flow from thick to thin. As it begins to cool down and solidify, it has a better chance of filling when it travels from thick to thin. If that were reversed, the material would begin to solidify as it passed through the thin sections and not enough material would be available to finish filling the thicker sections. In addition, the part is much stronger when filling from thick to thin, as shown in Figure 3.28. The ball-shaped molecules traveling in the part gated from thick to thin are steadily compressed and bonded tightly throughout the part. However, in the part gated from thin to thick, the molecules are expanded and not well bonded, and the material has not been allowed to “pack.” This results in a much weaker part. Our first desire then is to locate the gate in the thickest area of the part.

As a general rule-of-thumb, a gate should exist for every (203 to 254 mm) of flow, in any direction. However, due to the differences in shear sensitivity, specific heat rates, and MI, it is better to analyze gating situations using one of the many finite analysis programs available today.

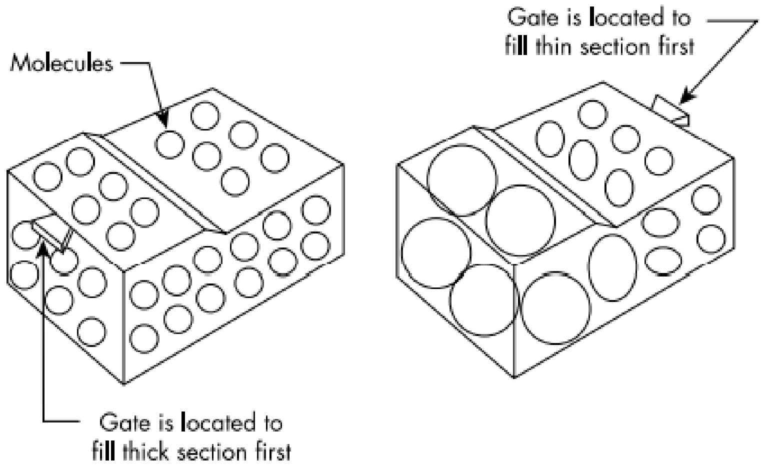


Figure 61: Gating to fill from thick to thin.

0.22.4.2 Sprue Gates

Sprue gating has long been considered the best way to gate a part. It is used primarily in single-cavity molds due to its basic nature, as shown in Figure 3.29. In this situation, the molten plastic is injected directly into the cavity image without the need for runner systems, and the gate is circular in cross section. Because the cavity is located centrally in relation to the sprue bushing, the flow of plastic is central to the cavity. This results in even flow distribution across the entire cavity. Stress is minimized by this condition, and flow lines are greatly reduced. In addition, sink marks are virtually nonexistent because the sprue stays molten all during the injection phase of the process, and holding pressure can be applied until the material in the part solidifies, thus minimizing shrink conditions.

Dimensioning the sprue gate requires that the diameter where the sprue meets the cavity be slightly larger than the wall thickness of the part at that junction. This ensures the sprue bushing will stay molten while the material in the cavity is flowing and solidifying. The shape of the sprue bushing, which is commercially available, forms the sprue.

The biggest disadvantage in using sprue gates is that they must be removed from the molded part. Even the best removal process leaves some visual evidence of the sprue gate, and this may be objectionable on specific part designs. A decorative decal can be used to cover the vestige, or the sprue gate can be moved slightly off center, if necessary, to help disguise the condition.

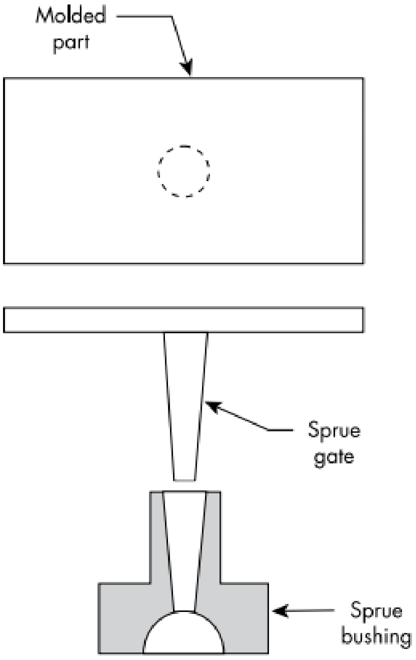


Figure 62: Sprue gate concept.

0.22.4.3 Basic Surface Gate

While a surface gate with a circular cross section is ideal for creating minimum stress, it is difficult to machine into a mold and keep concentric around its centerlines. This is because of the expansion and contraction movement of the steel over time. Any offset that occurs can create undue shear that may result in thermal degradation of the plastic. Therefore, rectangular surface gates have become the norm.

Figure 3.30 shows how a basic surface gate can be dimensioned. The basic surface gate is the most popular gate used. It is an opening in a wall and has three essentials: D (depth), W (width), and L (land). Each requires logical dimensioning that is based on the viscosity of the material being molded. Stiff materials (high viscosity) require larger openings than easier-flowing materials.

We start with the D dimension. We always gate into the thickest section of the part. The depth of our gate (D) will be 40 to 90% of that thickness. The thinner dimension would be used for easy flowing materials, such as nylon (a polyamide), while the thicker dimension would be used for stiffer materials, such as polycarbonate. We want the gate to be as thin as possible because it determines the overall cycle time (thicker gates require longer cooling time). So, we should start with the thinner dimension and only increase it if necessary. A good source of information for desired gate thickness is the material supplier of the specific

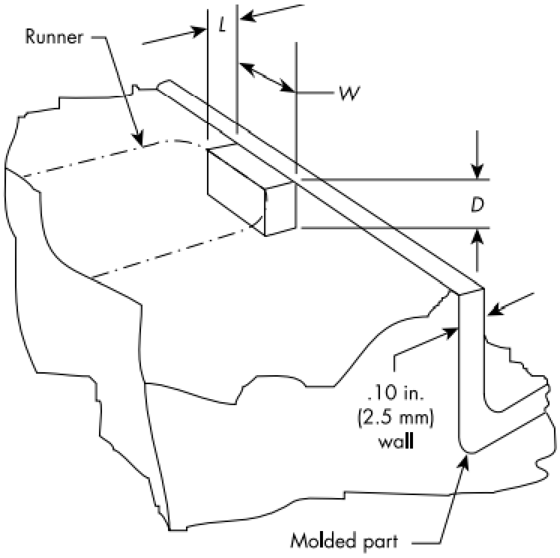


Figure 63: Basic surface gate dimensions.

material being molded.

After the D dimension has been determined, we can develop the W (width) dimension. A rule-of-thumb states that the width of the gate should be at least double the depth, and can be as much as 10 times the depth. We can start with W being twice D. We can increase W later, if needed.

That brings us to the L (land) dimension. The L dimension should be half of the D dimension, but never greater than (1.6 mm). If the L dimension is too great, it will cause the molten material to begin to solidify because the plastic must travel a long distance through a small opening. The surrounding steel pulls heat from the plastic quicker than if the L dimension is short. A typical effect of a land that is too great is "worming," a snakelike appearance on the molded part surface, emanating from the gate area.

0.22.4.4 Edge Gate

The edge gate is a variation of the basic surface gate and is used primarily for molding parts with large surfaces and thin walls, such as flat plates. Figure 3.31 shows a typical edge gate design. Note that the land (L) of this gate design is thin and narrow, and there is a secondary runner (d diameter) feeding from the primary runner. The narrow land acts as a throttle and causes molten material to fill the secondary runner area before it can enter the cavity image. This results in a uniform filling condition as the molten plastic leaves the land area and continues across the cavity image. The result is a part with uniform shrinkage in all directions (even crystalline materials) and parallel molecular orientation across the whole width, which is critical for optical products such as lenses.

Dimensioning an edge gate depends upon the viscosity of the material being molded, but, in general, the following values can be used. The L dimension should be from (0.5 to 1.5 mm) and this land should run across the entire width of the part. The thickness of this land (L) should be 25 to 75% the thickness (t) of the part being molded, depending on how easily the plastic flows, with the stiffer plastic requiring the higher percentage.

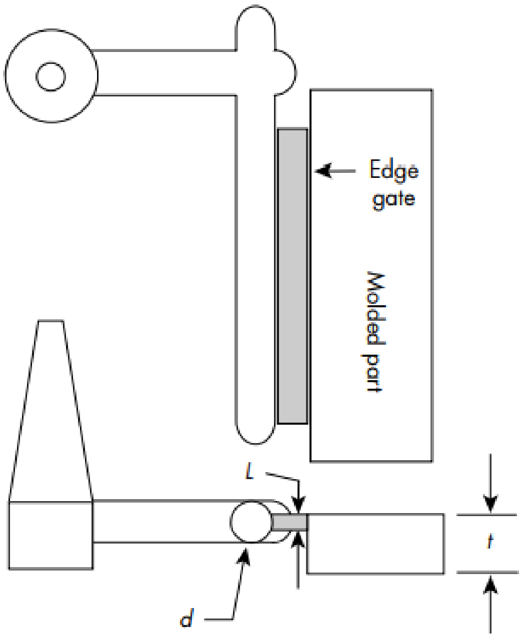


Figure 64: Typical edge gate design.

The diameter of the secondary runner (d) should range in thickness from the same as the thickness (t) of the part being molded, to 30% greater than that thickness (t + 30% of t), depending on how easily the plastic flows. If d must be greater than t, the overall cycle time will be excessive because the d dimension will take longer to solidify than the part itself. Like the land, the secondary runner d should span the entire width of the part.

0.22.4.5 Disc Gate

Use of the disc gate design allows uniform filling when molding a cylindrical, sleeve-shaped part, as shown in Figure 3.32.

A disc gate is actually a diaphragm. It spans the opening at one end of a sleeve-

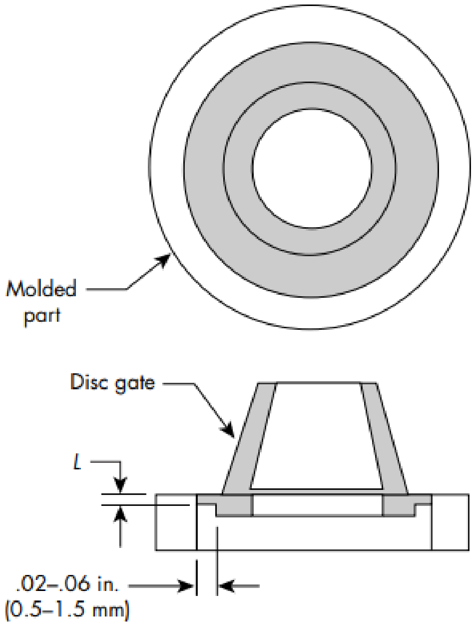


Figure 65: Disc gate design.

shaped part and feeds material directly into the part. The diaphragm is thick in its primary body, equal to or 25% greater than the wall thickness of the molded part. This area, known as the land is directly connected to the molded part. The thickness (L) of this land should be 50% to 75% of the molded part wall thickness. The land should be (0.5 to 1.5 mm) in width around its perimeter.

0.22.4.6 Ring Gate

A ring gate is a variation of the disc gate and is used for molding long, sleeve-shaped, cylindrical parts that need internal cores to be supported at both ends of the part. The ring shape results from the use of two runners, or feed channels. The main runner feeds a secondary runner that encircles (internally or externally) the cavity image, as shown in Figure 3.33.

External ring gates are the most common, and can be used if gate witness lines are visually acceptable. If not, internal ring gates may be used; however, they are more expensive to create and result in additional knit lines (at least two) being formed. The nature of ring gate designs results in knit lines. While these are unavoidable, they can be minimized through processing parameters. Ring gate knit lines are always stronger than knit lines formed by other conditions.

As always, the dimensions of the ring gates depend on the viscosity of the mate-

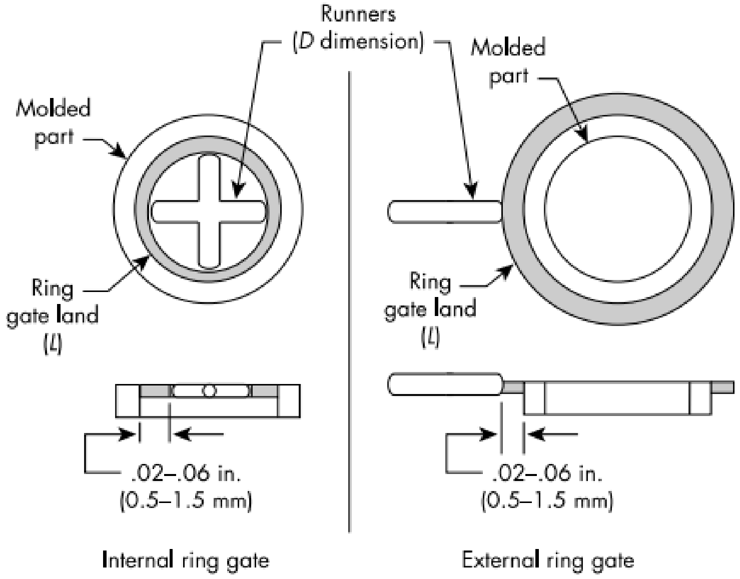


Figure 66: Ring gate designs, internal and external.

rial being molded. The D dimension of the primary and secondary runners should be 25% greater than the wall thickness of the part being molded. The thickness of the L (land) dimension should be 50 to 75% of the wall thickness of the molded part. The width of the land should be (0.5 to 1.5 mm) around the perimeter of the cavity image.

0.22.4.7 Drop Gates (Three-plate Molds)

Sometimes called pin gates, drop gates are useful when a part cannot be gated with conventional surface gates due to aesthetics or mechanical interference of the gate vestige in later assembly. Drop gates allow automatic degating of the part from the runner as the mold opens and the parts are ejected, as shown in Figure 3.34. The drop gate is formed as the result of using a three-plate mold construction. The three-plate mold allows us to locate the runner system between the A plate and the X plate. Then, conical holes are machined from the runner to the face of the X plate. These are the drop gates, called that because they drop from the runner to the cavity image.

The use of drop gates requires the runner to have a larger diameter so that enough material can travel the extra length incurred. In addition, the drop gate major diameters are large to keep the molten plastic from solidifying before the cavity is filled. Therefore, the entire runner system contains much more material than a standard surface runner would contain, and this material only has value as regrind.

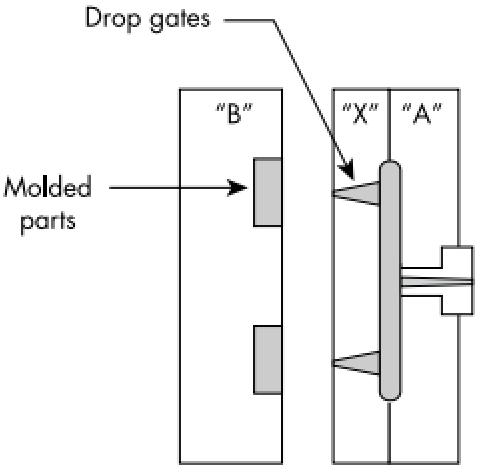


Figure 67: Drop gate (pin gate) concept.

The biggest advantage to a drop gate is the ability to place it practically anywhere on the molded part. It is especially easy if the gate can be located on the surface of the part formed between the X and B plates. With creative manipulation, the gate can be located inside the part, under the part, or on the sidewalls of the part, almost as easily. The advantage of placing the gate anywhere allows much better control of flow into the cavity image, and can even simplify construction of certain molds. As mentioned earlier, drop gates allow automatic degating of the molded part from the runner system during the molds operational phases.

0.22.4.8 Tunnel Gates

Tunnel gates, also known as subgates, submarine gates, and banana gates, are used for both automatic degating of molded parts within the mold, and locating gates in areas not accessible by standard surface gating. They are commonly used for multiple cavity molds producing small parts, but also can be utilized for larger parts and single-cavity molds.

As shown in Figures 3.35 and 3.36, tunnel gates have two basic design options: pointed tunnel or truncated tunnel. The pointed tunnel provides a smaller, rounder orifice than the truncated version and results in faster cooling time for the gate. However, by freezing off (solidifying) earlier, the pointed version does not permit long holding pressure time that might be needed for close-tolerance or high-strength products.

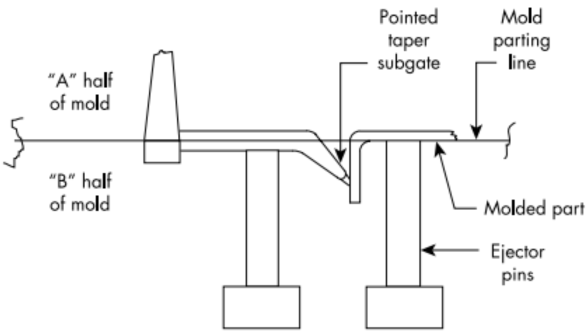


Figure 68: Tunnel gate with pointed tunnel.

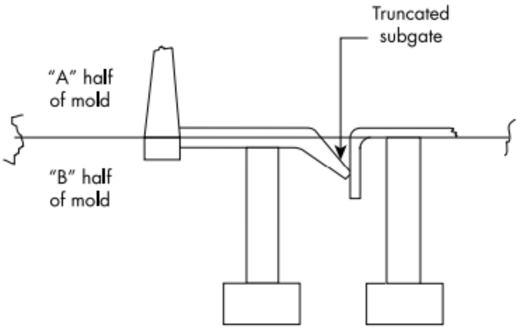


Figure 69: Tunnel gate with truncated tunnel.

Figure 3.37 shows a curved tunnel gate, commonly referred to as a banana gate. Banana gates are not common but can be used for low-depth parts made of softer plastics, or when the sharper corners of standard tunnel gates are undesirable (possibly causing stresses or shear degradation).
Figure 3.38 shows a tunnel gate being used to gate into an ejector pin. This concept allows gating into the inside surface (or B side) of a molded part (similar to the banana gate), which may be required for aesthetic purposes. Remember, if the part is thicker there than anywhere else, gate the thickest section whenever possible.

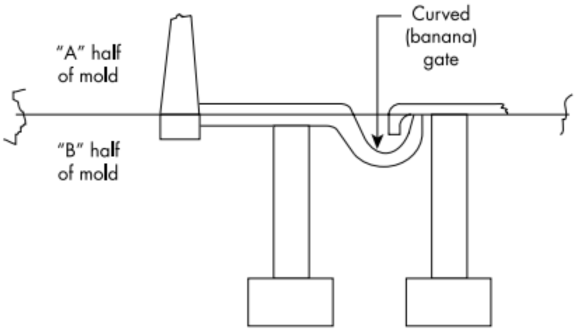


Figure 70: Curved or banana gate design.

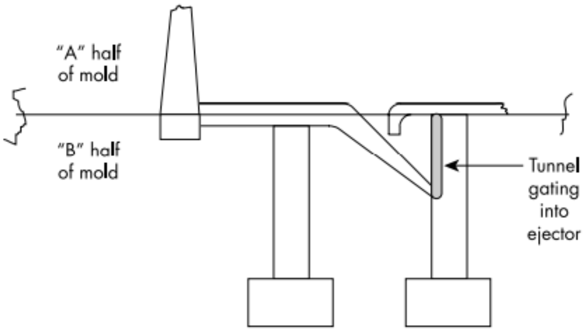


Figure 71: Tunnel gating into an ejector pin.

One note of importance concerning all gates, but especially tunnel-type gates: an ejector pin must be located as close as possible to the junction of the tunnel gate and the molded part. This is to ensure proper separation of the gate from the part, and proper removal of the gate from the mold. Without this ejection, the gate may break away from both the part and the runner and stay in the mold causing successive cycles to produce blanks in that cavity.

0.22.5 Mold Venting

0.22.5.1 Why Vent?

Vents are needed to allow trapped air and processing gases to vacate the mold. When the mold is closed in preparation for injecting molten plastic, air is trapped in every cavity or opening within the closed mold, including the runner system. Without venting, this air will compress under the pressure of incoming material. In fact, it compresses the material so much that it will ignite and burn up the oxygen available to it. This results in charred plastic. It also requires very high injection pressure to overcome the resistance of the trapped air to compression. High injection pressures cause undue stress to be molded into the plastic part.

0.22.5.2 Types, Sizes, and Locations of Vents

Basic venting is a series of paths machined across the shut-off land of the cavity image, as shown Figure 3.39. Dimensioning the vent begins with determining the proper D dimension. This can be taken from Table 3.2 or estimated based on the viscosity of the flowing plastic. Easy flowing plastics, such as nylon, require a thin vent, while thicker vents can be used for high viscosity materials, such as polycarbonate.

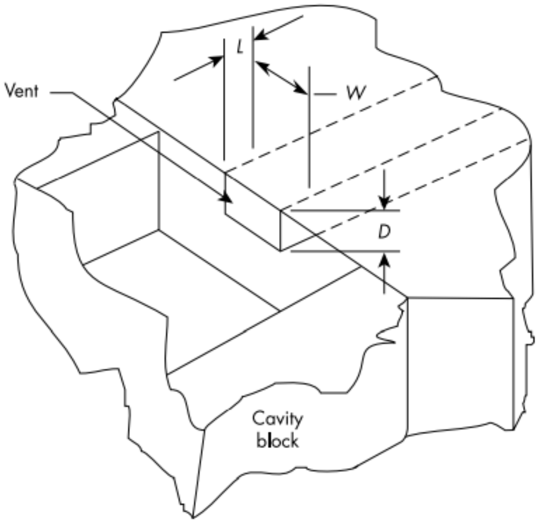


Figure 72: Basic venting concept.

Note that the vent depth for the runner as shown in Table 3.2 is twice the depth of vent for the cavity. This is because we are not too concerned with potential flash on the runner, but flash on the part can cause injury to the end user, and loss of pressure buildup in the cavity. The column designating depth of vent for the cavity

Table 11: Recommended Vent Depths.

Material	Cavity (mm)	Runner (mm)
ABS	(0.05)	(0.10)
Acetal	(0.017)	(0.038)
Acrylic	(0.05)	(0.10)
Cellulose acetate	(0.025)	(0.05)
Cellulose Acetate butyrate	(0.025)	(0.05)
Ionomer	(0.017)	(0.038)
Nylon 6/6	(0.0127)	(0.025)
Polycarbonate	(0.05)	(0.10)
Polyethylene	(0.025)	(0.05)
Polypropylene	(0.025)	(0.05)
Polyphenylene oxide	(0.05)	(0.10)
Polyphenylene sulfide	(0.0127)	(0.025)
Polysulfone	(0.025)	(0.05)
Polystyrene	(0.025)	(0.05)
Rigid PVC	(0.05)	(0.10)

lists dimensions that allow air to escape but keeps plastic from entering the vent.

After the D dimension is determined, we need to determine the W dimension, or width of vent. The minimum dimension for vent width should be (3.2 mm). A more practical and preferred dimension is (6.4 mm). However, the W dimension has no maximum. In theory, the vent width can run all the way around the perimeter of the parting line of the cavity image, without stopping. Of course, it would then eliminate itself. Therefore, we need to be practical in determining the maximum W dimension. We use a rule-of thumb that states that at least 30% of the perimeter of the cavity parting line should be vented. That leaves some strength to the steel surrounding the cavity image, but provides adequate venting.

Removing trapped air from normally inaccessible areas requires other types of vents. For example, such vents can be used at the bottom of a blind hole, at the base of deep walls or corners, or where parting line vents are not effective. Figure 3.40 shows how these locations can be vented.

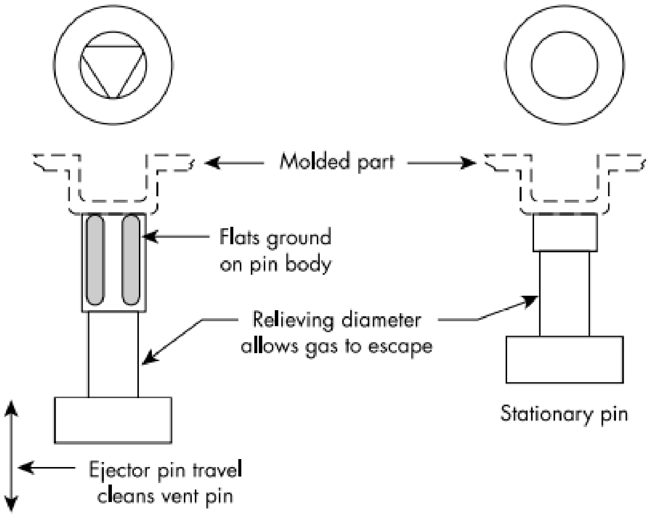


Figure 73: Venting nonparting line areas.

0.23 Controlling Mold Temperatures

0.23.1 Overview

After the molten plastic is injected into the cavity image, it is allowed to stay there, under pressure, until it has cooled down and solidified enough to be removed from the mold. The plastic does not need to be totally cooled, just enough to allow ejection of the finished product without unacceptable distortion occurring to the plastic. This cooling is done by a mold temperature control system that removes heat from the mold and maintains the correct mold temperature. In the following section, we will examine the more common methods used to accomplish this feat[24].

0.23.2 Waterlines

The use of waterlines machined throughout the mold to allow water to flow through the mold is the most common method of controlling mold temperature. This is accomplished by drilling holes as close as possible to the actual molding area of the cavity sets (see Figure 3.41).

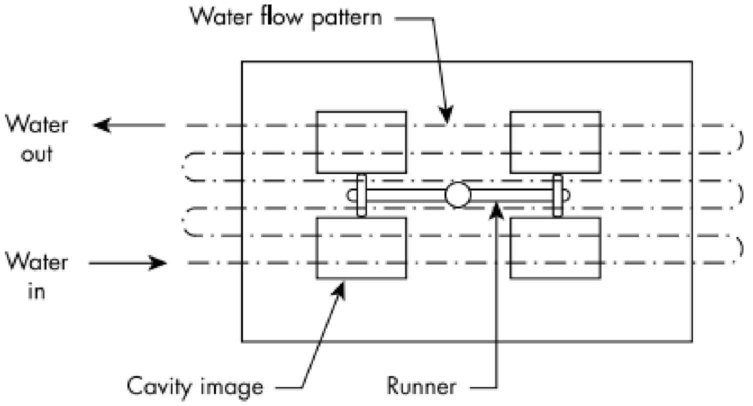


Figure 74: Common waterline layout.

Because of the accuracy required, the drilled holes are usually machined using a gun drill or deep-boring tool. Then, they are fitted with pipe threads for connecting nipples, hoses, and/or quick-disconnect fittings. Hoses are connected to floor-mounted temperature-control units or to machine-mounted water manifolds. In all cases, the temperature difference should not be more than 10°F (5.5°C) between any two points. This includes on the cavity molding surfaces, on the same mold half, or on both mold halves. A difference of more than 10°F (5.5°C) creates excessive stress in the part and will result in unbalanced plastic flow during the injection phase. These areas also shrink more than the cooler sections.

0.23.2.1 Control Units versus Manifolds

There are two primary ways to control mold temperatures: through use of control units and manifolds. Though the methods are different, the results are the same.

0.23.2.1.1 Control Units Figure 3.42 shows a mold temperature control unit. If floor-mounted temperature control units are used, a single unit should be used for each half of the mold. This eliminates the practice of using heated water from the first mold half to “cool” the second mold half, as is done when only one control unit is used.

The unit has a set point that is determined by the user. Water is circulated through the unit and heated until it meets the desired set point. As the water recirculates and the temperature increases, the unit begins to replace the heated water with tap water to maintain the desired temperature. This process continues as the unit functions to maintain the desired temperature. The circulating water temperature is displayed through a thermometer mounted on the unit. Remember that this is only an indication of the water temperature and not the mold temperature. The mold temperature must always be measured by using a surface pyrometer on the cavity surfaces.

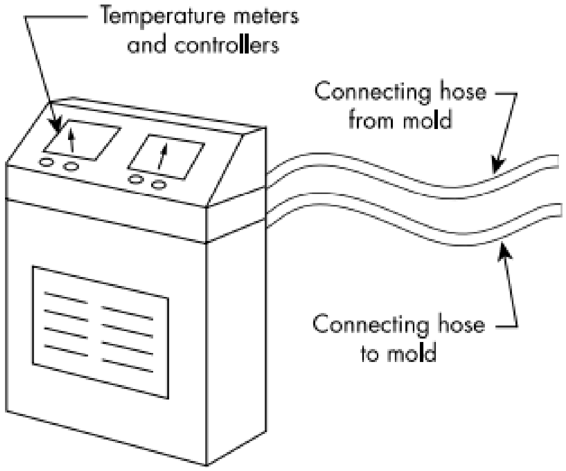


Figure 75: Mold temperature control unit.

0.23.2.1.2 Manifolds If water manifolds are utilized, the water flow is manipulated by throttling shutoff valves at the connection points of the manifold. Each line can be controlled independently. The mold temperature is determined by the flow of the water through these valves, but must be measured by a surface pyrometer on the mold surface itself. In most cases, the manifolds are connected to a source of chilled water, usually at a temperature of approximately 50°F (10°C). The manifold concept is based on a system that can control the mold temperature after it is established, but requires a slow and steady buildup as the metal of the mold absorbs heat from the injected plastic. The manifold system then depends on the continuous heat of incoming plastic to create a buildup of heat in the mold to provide the proper molding temperature. After that temperature is reached, the manifold system can maintain it. However, this process may take an hour or two to level out and the molding parameters may have to be adjusted constantly while this is happening to ensure properly molded parts. This creates a condition of having to "tweak" the manifolds during the first hour or so of production until the mold reaches the intended temperature.

Some molders use a colder mold for faster cycles and more profits. In fact, most parts require a slow cool down period especially when molding crystalline materials. This normally means a warm mold and long cooling cycles to create the highest level of physical strength in the final part. Cold molds will negatively affect the physical properties of a molded part, while warm molds will enhance those same properties. There are few situations where cold molds should be used. If manifold systems are in place, there is a tendency for the molder to routinely lower the mold temperature to the temperature of the manifold system, which is usually around 50°F (10°C). This may result in faster cycles, but can create a major quality problem if not understood and controlled.

0.23.2.2 Laminar Flow versus Turbulent Flow

There are two different types of flow that water can experience when traveling through a waterline of a mold: laminar or turbulent. Figure 3.43 shows the differences between the two conditions. Both conditions will remove heat from the surrounding mold metal, but the laminar flow is not nearly as effective as the turbulent flow. Note that in the laminar flow diagram the water travels in separate layers. The layers nearest the outside are next to the mold metal and are in direct contact with the heat that needs to be removed. These layers move slowly (due to friction) and transfer some of that heat to the faster-moving inner layers. However, the very center layer, moving fastest of all, receives no heat at all. In the turbulent flow model, the water is constantly being tumbled and mixed. All of the water is in contact with the mold metal at one time or another and all of it is used to remove heat from the mold metal. This is the desired effect.

The creation of turbulence is a function of flow rate, waterline diameter, water viscosity, water temperature, and velocity of the water as it travels through the

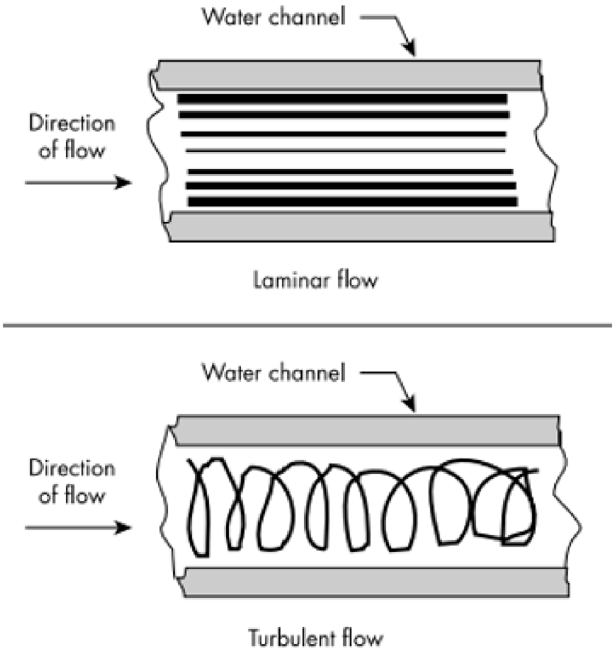


Figure 76: Laminar flow versus turbulent flow.

Table 12: Water Viscosity versus Temperature.

Water Temperature (°F [°C])	Viscosity (n)(centistokes)
32 (0)	1.79
50 (10)	1.30
68.4 (20.2)	1.00
100 (37.8)	.68
150 (65.6)	.43
212 (100)	.28

channels. Whether concerning laminar or turbulent flow, these conditions are characterized by a ratio known as the Reynolds number. Conditions causing a Reynolds number of 2,000 or less will result in laminar flow. Ideal turbulence is found when conditions create a Reynolds number of 3,500 or more. In between exists a transition area that fluctuates between laminar and turbulent flow.

Determining the existing Reynolds number can be achieved by using the following formula:

$$R = KQ/Dn$$

Where:

K = *3,160*
Q = *flow rate (gpm)*
D = *diameter of waterline (inches)*
n = *water viscosity (centistokes), see Table* 3.3

You can easily detect whether or not a mold temperature is being properly maintained by noting the temperature difference between water going in and water coming out. Contrary to popular belief, there should not be more than a 10°F (5.5°C) difference between the two temperatures. If outgoing water is hotter than incoming water, it means that there is too much heat retained in the mold and the water is not bringing it out fast enough. An ideal condition is one where the heat is removed as fast as it is created, which would result in the water temperature being exactly the same going in as coming out. While this may not be entirely possible, there should be no more than a 10° F (5.5°C) difference. If the waterlines have been designed for the proper Reynolds number and the return waterline is hotter, scale buildup may be occurring in the lines or some other item is plugging the flow.

0.23.2.3 Determining Location of Waterlines.

An easy statement to make is that the waterlines should be located as close as possible to the surface of the cavity image forming the molded product. While that is easy to state, it is not easily accomplished. The reason is that drilled waterlines must follow straight paths, but most molded products have three-dimensional qualities and are not flat and straight. In many cases, drilled lines are placed so that they surround the part as much as possible, but do not take the exact configuration, as shown in Figure 3.44.

The round-shape cavity is surrounded by a square-shape waterline pattern. This is not efficient and causes uneven cooling in the molded part. This is due to the uneven location of the water being used to pull heat from the plastic. The uneven cooling will result in a tendency for the flat part to warp and bow as some areas cool down quickly while others cool at a slower rate.

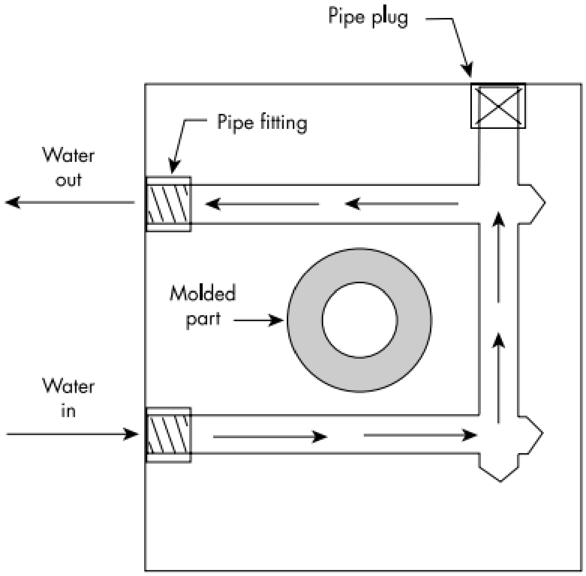


Figure 77: Improper waterline pattern.

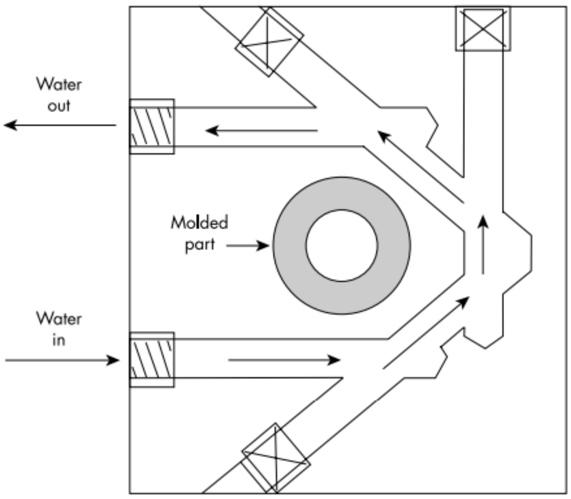


Figure 78: Acceptable waterline pattern.

Figure 3.45 shows a more accepted waterline pattern for this product. The coolant follows a pattern that is much closer to the actual shape of the product being molded. However, to create this pattern, a complicated system of drilled and plugged waterlines must be created. This is expensive, and still does not form an ideal pattern.

A better pattern may be the one shown in Figure 3.46, where the water flows in a pattern that very closely matches the shape of the product being molded. However, to incorporate this pattern requires a system of open-faced channels, connected together, and sealed with O-rings to eliminate leakage. While this provides the most acceptable pattern, it is expensive to create and requires constant maintenance to keep leaks from forming and damaging the molded parts, as well as the mold itself.

A further problem is that the steel used for making the mold must contain the high pressure initiated by the injection phase of the molding process. Therefore, the waterlines cannot be too close to the cavity or they will create a breakthrough of the cavity steel. A rule-of-thumb suggests that waterlines be no closer than 1.5 times their diameter, but a safer rule-of-thumb states they should be no closer than a full two diameters from the cavity, as shown in Figure 3.47. The diameter is determined by what is required to provide the proper Reynolds number value. However, that does not mean there should only be a single waterline. If depth allows, additional layers of waterlines can be used.

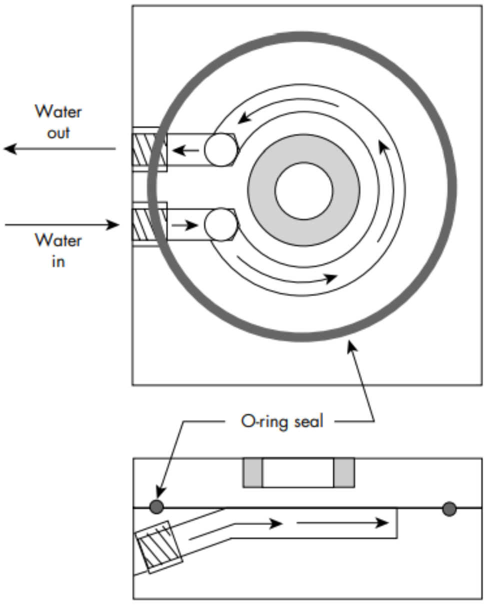


Figure 79: Better waterline pattern.

When locating waterlines, other items that make up the construction of the mold may cause interference with the waterlines. For example, bolts are used to hold the cavity blocks in place. These come from behind the cavity blocks and might interfere with any waterlines running under these blocks. Also, ejector pins for the part and the runner must travel through the B side of the mold, and these also might interfere with waterline locations, as shown in Figure 3.48. It is critical that the mold designer lay these items out with two primary thoughts in mind: first, locate the cavity blocks as close as possible to the center of the mold (to minimize flow travel of the incoming plastic); and second, locate waterlines as close as possible to the contour of the cavity image. These two concepts must be brought to bear on each other until a compromise is created that satisfies both requirements.

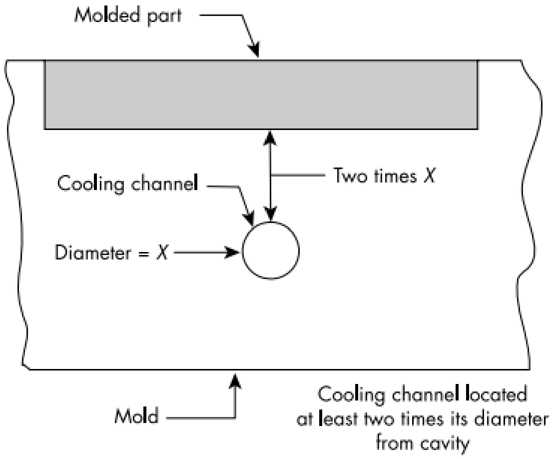


Figure 80: Waterline distance to cavity.

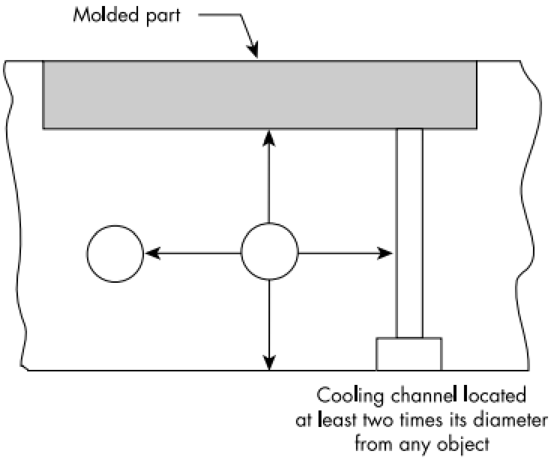


Figure 81: Waterlines and interference items.

0.24 Which Mold Material?

There are dozens of materials that can be used for making molds for producing plastic products, including many types of aluminum, brass and copper, epoxy, and many others, as well as combinations of these. The following section describes some of the more common materials and the role they play in the making of molds.[24]

1020 carbon steel. This steel is used for ejector plates and ejector retainer plates and is easily machined and welded. Not usually hardened because of distortion and warp, this material must be first carburized if hardening is preferred.

1030 carbon steel. Used for mold bases, ejector housings, and clamp plates, this steel has 25% greater tensile strength than 1020 and can be easily machined and welded. It can be hardened to Rockwell hardness C scale (R_c) 20 to 30.

1040 carbon steel. Commonly used for support pillars, this tough steel has good compressive strength and can be hardened to R_c 20 to 25.

4130 alloy steel. This is a high-strength steel used primarily for cavity and core retainer plates, support plates, and clamping plates, and is supplied at 26 to 35 R_c .

6145 alloy steel. Primary use for this type of steel is for sprue bushings and it is supplied at 42 to 48 R_c . S-7 tool steel. Shock resistant with good wear resistance, this steel is used for interlocks and latches and hardened to 55 to 58 R_c .

O-1 tool steel. This is a general purpose, oil-hardening steel used for small inserts and cores and hardened to 56 to 62 R_c .

A-2 alloy tool steel. This steel has good dimensional stability and abrasion resistance, and is used for hobs and slides and is hardened to 55 to 58 R_c .

A-6 tool steel. A general purpose oil-hardening steel with good dimensional stability and high hardness, its primary use is for optical quality cavities and cores and it is hardened to 56 to 60 R_c .

D-2 tool steel. This steel has high chromium and high carbon content, and is difficult to grind, but has excellent abrasion resistance. It is used for gate inserts, lifters, and slides, and is hardened to 58 to 60 R_c .

H-13 tool steel. This is a very high toughness, low-hardness steel used for high quality cavity and core requirements. It is primarily used for ejector pins, return pins, sprue pullers, leader pins, and slide-actuating angle pins, and supplied annealed at 15 to 20 R_c , but can be hardened to 60 R_c with little distortion.

P-20 tool steel. This is a modified 4130, commonly referred to as prehard. It is supplied at a R_c hardness of 28 to 40, which provides moderately high hardness, good machinability, and exceptional polishability. It is used primarily for cavities and cores, as well as stripper plates.

420 stainless steel. Used in applications requiring exceptional chemical resistance (such as molding PVC resins), this steel is usually supplied in an annealed condition (15 to 25 Rc), but can be hardened to 55 to 60 R_c . Its primary use is as a steel for cores and cavities.

0.25 Conclusion

For a good design of a plastic injection mould, the designer has to follow all the necessary steps, because the best choice of design parameters guarantees a good quality finished product.

CALCULATIONS AND MATERIAL RESISTANCE VERIFICATIONS

0.26 Introduction

Before any design is finalized, there are many factors to consider, including various calculations that need to be done in order to confirm that the design is practical. In this chapter, we will through go all the necessary calculations needed in our case including the machine selection requirements and the material resistance verifications.

0.27 Machine selection

Selecting the right plastic injection machine is one of the most important criteria in making quality parts consistently and profitably.
There are many factors to consider when choosing the right machine such as:

- Injection capacity.
- Clamping force.
- Recovery rate.
- Tie bar spacing.
- mold height.

0.27.1 Theoretical injection capacity

The injection capacity depends on the shot size/weight which is best defined as the maximum amount of plastic that the injection machinery is capable of injecting into the molding cavity during one molding cycle.

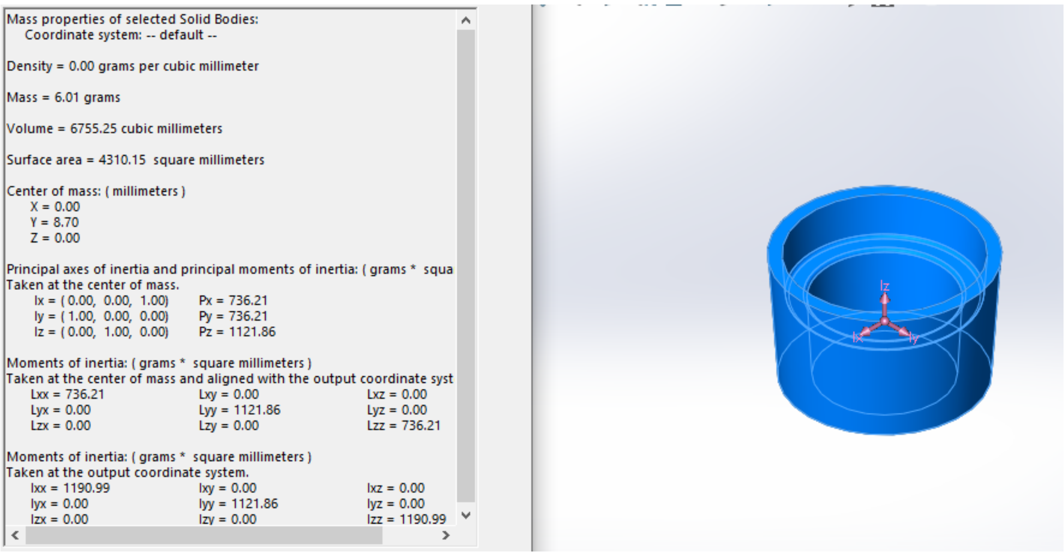
Table 13: injection capacity of some machine.

Machine clamping force	KN(T) Shot size cm^3	Shot weight of PP
300 (30)	12	10
500 (50)	12	10
800 (80)	25	22
1000 (100)	49	44
1300 (130)	49	44
1800 (180)	97	87
2160 (216)	226	203
2750 (275)	349	314
3440 (344)	349	314
4420 (442)	687	618
5400 (540)	1026	923
6380 (638)	2328	2095

0.27.1.1 Weight of plastic part

The weight of our product is determined using the CAD Software SolidWorks.

$$m_p = 6.01g \tag{1}$$

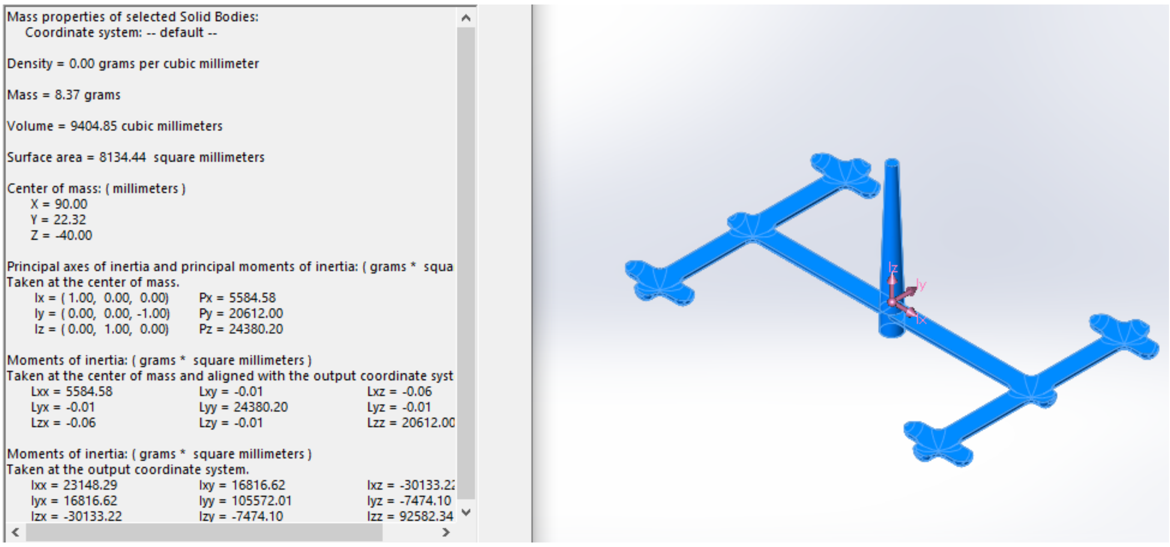


```
Mass properties of selected Solid Bodies:
   Coordinate system: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 6.01 grams

Volume = 6755.25 cubic millimeters

Surface area = 4310.15  square millimeters

Center of mass: ( millimeters )
     X = 0.00
     Y = 8.70
     Z = 0.00

Principal axes of inertia and principal moments of inertia: ( grams *  squa
Taken at the center of mass.
      Ix = ( 0.00,  0.00,  1.00)       Px = 736.21
      Iy = ( 1.00,  0.00,  0.00)       Py = 736.21
      Iz = ( 0.00,  1.00,  0.00)       Pz = 1121.86

Moments of inertia: ( grams *  square millimeters )
Taken at the center of mass and aligned with the output coordinate syst
     Lxx = 736.21                 Lxy = 0.00                 Lxz = 0.00
     Lyx = 0.00                   Lyy = 1121.86              Lyz = 0.00
     Lzx = 0.00                   Lzy = 0.00                 Lzz = 736.21

Moments of inertia: ( grams *  square millimeters )
Taken at the output coordinate system.
     Ixx = 1190.99                Ixy = 0.00                 Ixz = 0.00
     Iyx = 0.00                   Iyy = 1121.86              Iyz = 0.00
     Izx = 0.00                   Izy = 0.00                 Izz = 1190.99
```

Figure 82: Weight of plastic part (Mass properties - SolidWorks)

0.27.1.2 Weight of feed system

$$m_f = 8.37g \tag{2}$$



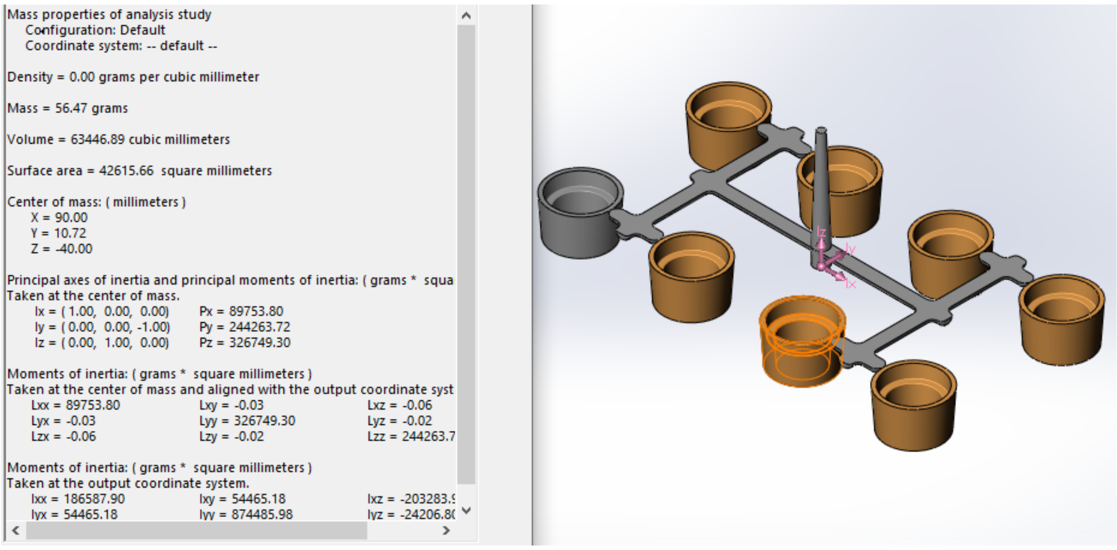
```
Mass properties of selected Solid Bodies:
    Coordinate system: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 8.37 grams

Volume = 9404.85 cubic millimeters

Surface area = 8134.44  square millimeters

Center of mass: ( millimeters )
     X = 90.00
     Y = 22.32
     Z = -40.00

Principal axes of inertia and principal moments of inertia: ( grams *  squa
Taken at the center of mass.
      Ix = ( 1.00,  0.00,  0.00)       Px = 5584.58
      Iy = ( 0.00,  0.00, -1.00)       Py = 20612.00
      Iz = ( 0.00,  1.00,  0.00)       Pz = 24380.20

Moments of inertia: ( grams *  square millimeters )
Taken at the center of mass and aligned with the output coordinate syst
     Lxx = 5584.58                 Lxy = -0.01                 Lxz = -0.06
     Lyx = -0.01                   Lyy = 24380.20              Lyz = -0.01
     Lzx = -0.06                   Lzy = -0.01                 Lzz = 20612.00

Moments of inertia: ( grams *  square millimeters )
Taken at the output coordinate system.
     Ixx = 23148.29                Ixy = 16816.62              Ixz = -30133.2
     Iyx = 16816.62                Iyy = 105572.01             Iyz = -7474.10
     Izx = -30133.22               Izy = -7474.10              Izz = 92582.34
```

Figure 83: weight of feed system (Mass properties - SolidWorks)

0.27.1.3 Shot weight of the injected material

Our mold produces 8 parts plus their feeding system each cycle. Therefore, the machine's injection capacity has to exceed the shot weight of one cycle.

$$M = (8 \times 6.01) + 8.37 = 56.45g \tag{3}$$



```
Mass properties of analysis study
   Configuration: Default
   Coordinate system: -- default --

Density = 0.00 grams per cubic millimeter

Mass = 56.47 grams

Volume = 63446.89 cubic millimeters

Surface area = 42615.66  square millimeters

Center of mass: ( millimeters )
     X = 90.00
     Y = 10.72
     Z = -40.00

Principal axes of inertia and principal moments of inertia: ( grams *  squa
Taken at the center of mass.
      Ix = ( 1.00,  0.00,  0.00)       Px = 89753.80
      Iy = ( 0.00,  0.00, -1.00)       Py = 244263.72
      Iz = ( 0.00,  1.00,  0.00)       Pz = 326749.30

Moments of inertia: ( grams *  square millimeters )
Taken at the center of mass and aligned with the output coordinate syst
     Lxx = 89753.80                    Lxy = -0.03                    Lxz = -0.06
     Lyx = -0.03                       Lyy = 326749.30                Lyz = -0.02
     Lzx = -0.06                       Lzy = -0.02                    Lzz = 244263.7

Moments of inertia: ( grams *  square millimeters )
Taken at the output coordinate system.
     Ixx = 186587.90                   Ixy = 54465.18                 Ixz = -203283.
     Iyx = 54465.18                    Iyy = 874485.98                Iyz = -24206.8
```

Figure 84: Shot weight of the injected material (Mass properties - SolidWorks)

The machines which are capable of injecting the needed shot weight are >180T

0.27.2 Clamping force (Tonnage)

with plastic injection machines, tonnage is measured by how many tons with which the machine is capable of pressing together the platens, which holds the mold cavity and forms the plastic that is injected into said cavity to produce the desired part.

$$F = P \times S \times K \tag{4}$$

Where:

F: clamping force (T)
P: injected pressure (P=0.4 for PP)
S: the pressure projected area (given by SolidWorks)
K: factor of safety.

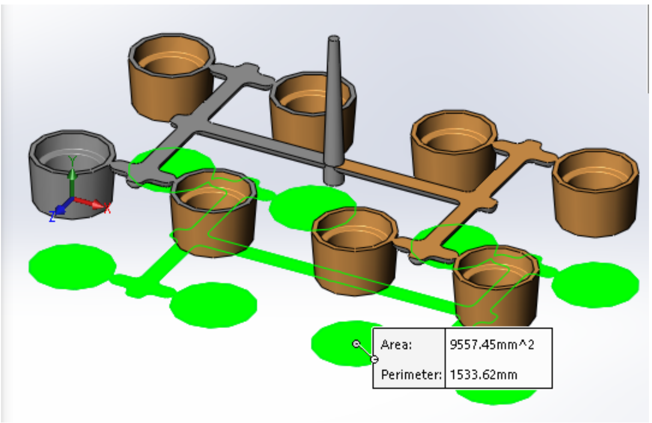


Figure 85: The pressure projected area (Measure analysis - SolidWorks)

$$0.4 \times 95.57 \times 2 = 76.456(T) \tag{5}$$

In conclusion, we need a machine that is capable of injecting a shot weight of 56.45g and exerting a clamping force of >76.456(T). therefore, the best machine choice is the 180(T) press.

0.27.3 Recovery rate

Even though the 180T press is capable of injecting 56.45g, we still need to verify its recovery rate compatibility with our product.

$$C = \frac{Shotsize}{Cycletime} \tag{6}$$

Where:

C: mass of recovered material per hour.
Cycle time: (see 4.2.1)

Our machine has to solidify:

$$\frac{56.45}{\frac{39.47}{3600}} = 5148.72g/h = 5.1kg/h \tag{7}$$

Our press's recovery rate is 74kg/h. So, this condition is verified.

0.27.4 Tie bar spacing

Tie bar spacing is defined as the space between the horizontal tie-bars on an injection molding machine. Basically, the measurement, along with the platen max spacing determines the maximum size of molds that can be placed in the molding machine.

Our mold's dimensions are:
400mm in length.
260mm in width.

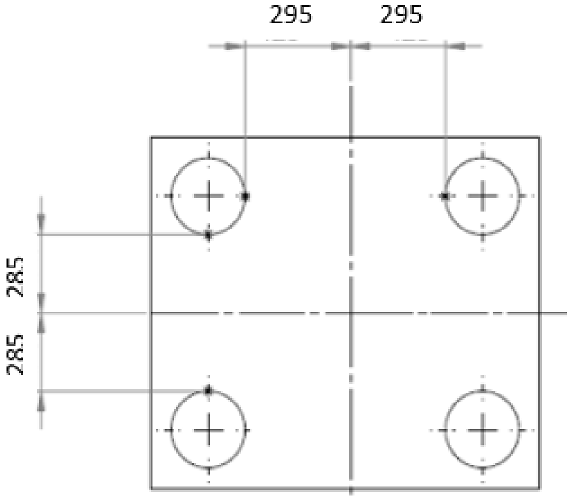


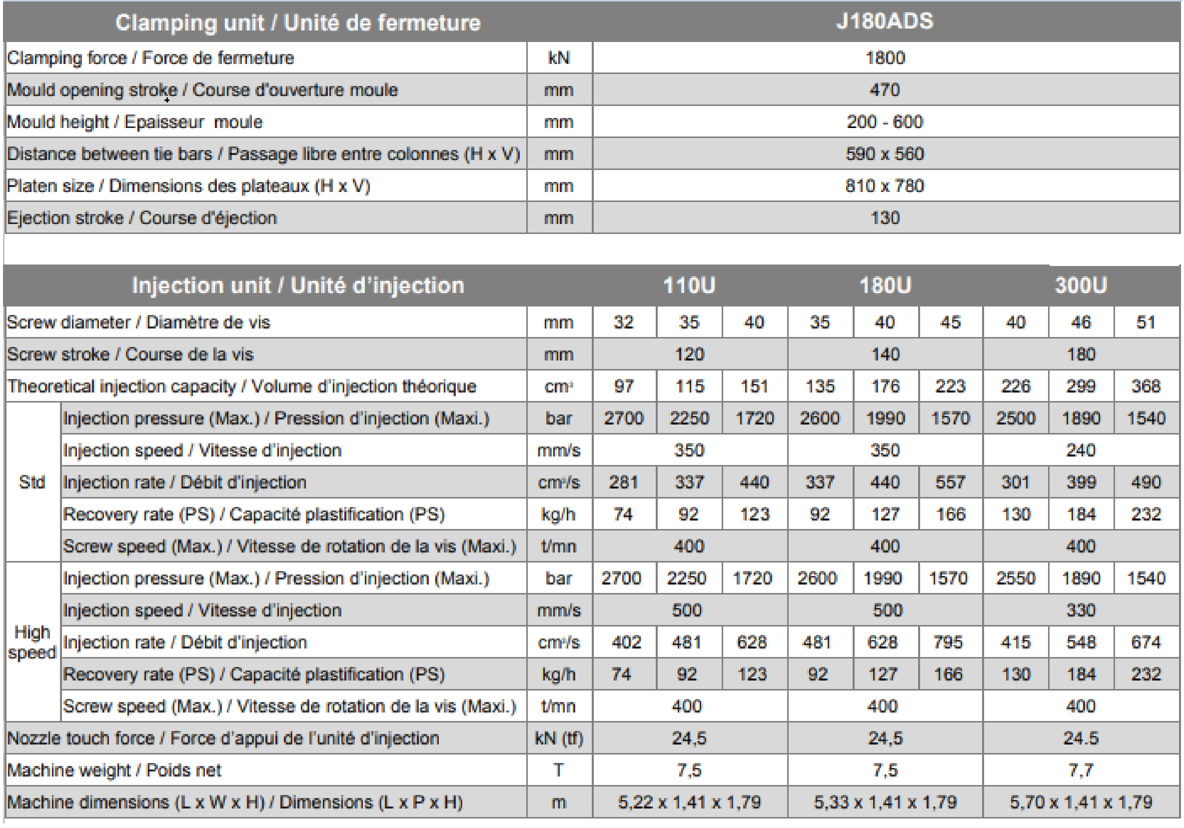
Figure 86: Tie bar spacing of 180(T) press

0.27.5 Mold height

The mold height of our 180T press is (200-600) mm (see Figure ??).
Our mold has a height of 204.5 mm
So, we are well under the limits.

0.27.6 Technical specifications of the machine

Table 14: Technical specifications of the machine.



Clamping unit / Unité de fermeture		J180ADS
Clamping force / Force de fermeture	kN	1800
Mould opening stroke / Course d'ouverture moule	mm	470
Mould height / Epaisseur moule	mm	200 - 600
Distance between tie bars / Passage libre entre colonnes (H x V)	mm	590 x 560
Platen size / Dimensions des plateaux (H x V)	mm	810 x 780
Ejection stroke / Course d'éjection	mm	130

Injection unit / Unité d'injection			110U			180U			300U		
Screw diameter / Diamètre de vis		mm	32	35	40	35	40	45	40	46	51
Screw stroke / Course de la vis		mm	120			140			180		
Theoretical injection capacity / Volume d'injection théorique		cm^3	97	115	151	135	176	223	226	299	368
Std	Injection pressure (Max.) / Pression d'injection (Maxi.)	bar	2700	2250	1720	2600	1990	1570	2500	1890	1540
	Injection speed / Vitesse d'injection	mm/s	350			350			240		
	Injection rate / Débit d'injection	cm^3/s	281	337	440	337	440	557	301	399	490
	Recovery rate (PS) / Capacité plastification (PS)	kg/h	74	92	123	92	127	166	130	184	232
	Screw speed (Max.) / Vitesse de rotation de la vis (Maxi.)	t/mn	400			400			400		
High speed	Injection pressure (Max.) / Pression d'injection (Maxi.)	bar	2700	2250	1720	2600	1990	1570	2550	1890	1540
	Injection speed / Vitesse d'injection	mm/s	500			500			330		
	Injection rate / Débit d'injection	cm^3/s	402	481	628	481	628	795	415	548	674
	Recovery rate (PS) / Capacité plastification (PS)	kg/h	74	92	123	92	127	166	130	184	232
	Screw speed (Max.) / Vitesse de rotation de la vis (Maxi.)	t/mn	400			400			400		
Nozzle touch force / Force d'appui de l'unité d'injection		kN (tf)	24,5			24,5			24.5		
Machine weight / Poids net		T	7,5			7,5			7,7		
Machine dimensions (L x W x H) / Dimensions (L x P x H)		m	5,22 x 1,41 x 1,79			5,33 x 1,41 x 1,79			5,70 x 1,41 x 1,79		

0.28 Heat balance

During this study, we make an assumption to simplify the calculations:

We assume that the heat transfer fluid alone must evacuate all the energy provided by the polymer.

0.28.1 Cycle time estimation

The cycle time is the total time required to complete all the stages of the injection molding cycle.

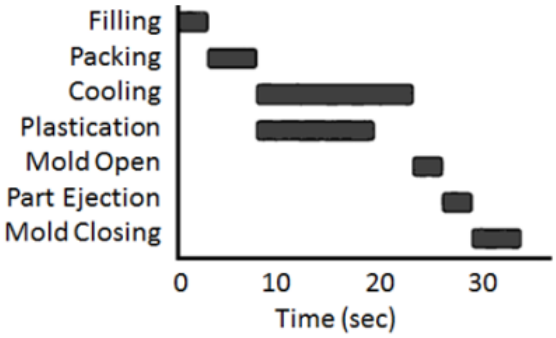
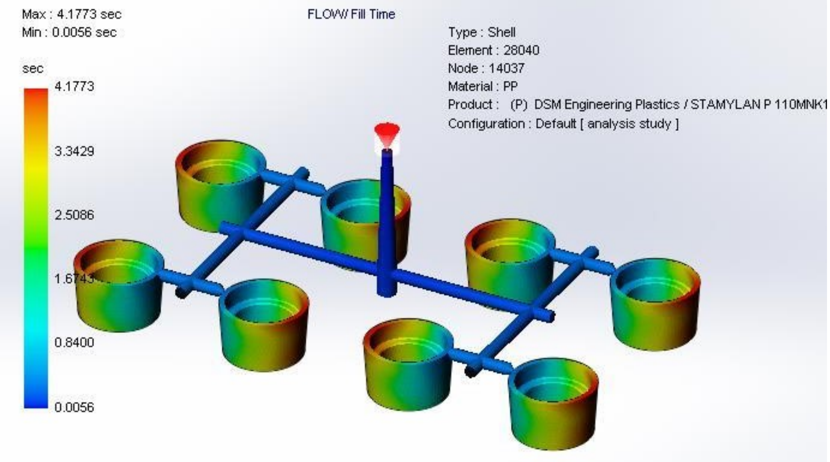


Figure 87: Diagram of stage-times for the injection molding cycle

The cycle time is made up of:
1- Mold closing time (5s)
2- Filling time (given by SolidWorks plastics simulation 4.19s)
3- Pack time (5s)
4- Cooling time (20.28)
5- Mold ejection time (5s)


Max : 4.1773 sec
Min : 0.0056 sec

FLOW/ Fill Time

Type : Shell
Element : 28040
Node : 14037
Material : PP
Product : (P) DSM Engineering Plastics / STAMYLAN P 110MNK1
Configuration : Default [analysis study]

Figure 88: Fill time (SolidWorks Plastics Simulation)

$$5 + 4.19 + 5 + 20.28 + 5 = 39.47s \tag{8}$$

0.28.1.1 Theoretical calculation of cooling time

$$Cot = \frac{e^2}{\Pi^2 \times D} \ln[(\frac{8}{\Pi^2}) \times (\frac{T_i - T_m}{T_e - T_m})] \tag{9}$$

Where:

e: average thickness of plastic part (4.7mm)
D: thermal diffusivity of PP ($\frac{\lambda}{\rho} \times c$)
T$_i$: injection temperature (230°C)
T$_e$: ejection temperature (110°C)
T$_m$: mold temperature (40°C)
λ : Thermal conductivity of PP (0.147 W/m.K)
ρ : Mass density of PP (890 Kg/m^3)
c : Specific heat of PP (1881 J/kg.k)

$$D = \frac{0.147}{1881 \times 890} \tag{10}$$

D = 8.7× $10^{-}8$ m^2/s

$$Cot = 20.28s \tag{11}$$

0.28.2 Calculation of the heat quantity to be extracted from the part

$$Qh = M \times N(H_i - H_e) \tag{12}$$

Where:

M: mass of the injected plastic (56.45g)
N: number of cooling cycles per hour (N=3600×Cot)
H$_i$: specific heat capacity at injection
H$_e$: specific heat capacity at ejection

Ti= 230°C ... Hi= 143.4 kcal/kg (see Figure 4-9)
Te= 110°C ... He= 47.8 kcal/kg

$$Qh = 541.47kcal/h \tag{13}$$

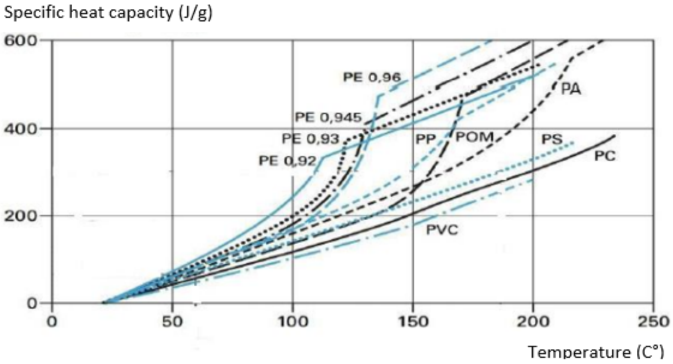


Fig 93 Variation of the specific heat capacity of polymers as a function of

Figure 89: ariation of the specific heat capacity of polymers as a function of temperature.

0.28.3 Hourly liquid consumption

$$C_h = \frac{Q_h}{C_f(T_s - T_e)} \tag{14}$$

Where:

C$_h$: hourly liquid consumption (kcal/h)
Q$_h$: extracted heat quantity (541.47 kcal/h)
C$_f$: heat capacity of the coolant (water = 1kcal/kg)
T$_e$: water entry temperature (20°C)
T$_s$: water exit temperature (25°C)

$$C_h = 108.29kg/h \tag{15}$$

0.28.4 Cooling system sizing

The dimensioning of the cooling channel must take into account the need of a turbulent flow and that of a large exchange surface with the cavity.

$$L_c = \frac{Q_h}{h \times \Pi \times d(T_c - T_f)} \tag{16}$$

Where:

L$_c$: length of the cooling channel.
Q$_h$: extracted heat quantity (541.47 kcal/h)
h: heat transfer coefficient.

d: channel diameter (6mm).
T_c: channel wall temperature.
T_f: fluid temperature at the center of the channel.

0.28.4.1 Calculation of heat transfer coefficient

$$R_e = \frac{V_f \times d}{\nu} \tag{17}$$

Where:

V_f: fluid speed (Vf=1850m/h)
R_e: reynold's number.
d: channel diameter.
ν: kinematic viscosity of water at Tf. (0.8x10$^-6$ m^2/s)(see appendix of water properties)

$$T_f = \frac{T_m + T_c}{2} \tag{18}$$

Where:

T_m: average fluid temperature along cooling channel.
T_c: channel wall temperature.

$$T_f = 31.25°C \tag{19}$$

$$Re = 3854.16 \tag{20}$$

Re>3500 turbulent flow:

$$h = 0.04 \times (R_e.P_r)^{0.75}(\frac{y}{d}) \tag{21}$$

Where:

P_r: prandtl number of water at 25°C (Pr=6.13)
Y: water conductivity at 25°C (y= 0.521kcal/h.m.°C)

$$h = 6618.95 kcal/h.m^2.°C \tag{22}$$

$$L_c = 0.87m \tag{23}$$

0.29 Material resistance verifications

0.29.1 Mold components resistance to clamping pressure

The press exerts a clamping force of 180T $\simeq$ 1800000N.

Resistance condition to high pressure:

$$\sigma = \frac{F}{S} \leq R_{pc} \tag{24}$$

$$R_{pc} = \frac{R_e}{s} \tag{25}$$

Where:
R_{pc}: Practical resistance to compression.
R_e: Material's Elastic resistance to compression.
s: Safety coefficient (s= 2) F: Exerted clamping force.
S: Surface area.

0.29.1.1 Surface area resistance

$$S_{pj} \geq \frac{F}{0.2 \times R_e} \tag{26}$$

Where:
S$_{pj}$: Surface area.(Calculated with SolidWorks Measure)

$$S_{pj} \geq \frac{1800000}{0.2 \times 880} = 10227.27mm^2 \tag{27}$$

S$_{pj}$ is the minimum surface that can support the clamping force, in our case, the contact surface between the two blocks is 39185.89 mm^2 largely superior.

0.29.1.2 Fixed part resistance verification

0.29.1.2.1 Core holder block.

$$\sigma = \tfrac{F}{S} \leq R_{pc}$$

Where:
R_e of 25CrMo4 is: 700 Mpa (see appendix of materials properties)
F = 1800000 N
S = 85549.85mm^2 (Calculated with SolidWorks)

$$R_{pc} = \frac{700}{2} = 350N/mm^2 \tag{28}$$

$$\sigma = \frac{1800000}{85549.85} = 21.04N/mm^2 < R_{pc} \tag{29}$$

Resistance verified.

0.29.1.2.2 Top clamp plate

$$\sigma = \frac{F}{S} \leq R_{pc}$$

Where:
R_e of C30 is: 350
F = 1800000 N
S = 101757.38mm^2

$$R_{pc} = \frac{350}{2} = 175 N/mm^2 \tag{30}$$

$$\sigma = \frac{1800000}{101757.38} = 17.69 N/mm^2 < R_{pc} \tag{31}$$

Resistance verified.

0.29.1.3 Moving part resistance verification

0.29.1.3.1 Cavity holder block

$$\sigma = \frac{F}{S} \leq R_{pc}$$

Where:
R_e of 25CrMo4 is: 700 Mpa
F = 1800000 N
S = 39185.89 mm^2

$$R_{pc} = \frac{700}{2} = 350 N/mm^2 \tag{32}$$

$$\sigma = \frac{1800000}{39185.89} = 45.93 N/mm^2 < R_pc \tag{33}$$

Resistance verified.

0.29.1.3.2 Spacer plates

$$\sigma = \frac{F}{S} \leq R_pc$$

Where:
R_e of C30 is: 350 Mpa
F = 1800000 N
S = 10136.56mm^2

$$R_{pc} = \frac{350}{2} = 175 N/mm^2 \tag{34}$$

$$\sigma = \frac{1800000}{10536.56} = 170.83 N/mm^2 < R_pc \tag{35}$$

Resistance verified.

0.29.2 Leader pins resistance to shear stress verification

The leader pins are subjected to a shear stress force applied by the weight of the moving half of the mold.

Resistance condition:

$$\tau = \frac{F}{n \times S} \leq R_{pg} \tag{36}$$

$$R_{pg} = \frac{R_{eg}}{s} = \frac{R_e}{s} \times 0.8 \tag{37}$$

Where:

R_{pg}: Practical resistance to shear stress.
R_{eg}: Material's elastic resistance to shear stress
R_e: Material's elastic resistance to compression
s: safety factor (s=2).
F: Weight of moving half of the mold.
S: Pin section mm^2 (S = $\frac{\Pi \times d^2}{4}$)
n: number of sections under shear stress.

$$R_{pg} = \frac{1230}{2} \times 0.8 = 492 N/mm^2 \tag{38}$$

$$S = \frac{3.14 \times 20^2}{s} = 314 mm^2 \tag{39}$$

$$\tau = \frac{24074.06}{4 \times 314} = 45.88 N/mm^2 \tag{40}$$

Where:
R_e of 12MnCr5 is: 1230 Mpa
F: (F = 57623.03 N)(Calculated with SolidWorks Measure)
d = 20mm.
n = 4.
Resistance verified.

0.29.3 Return pin resistance to shear stress verification

The return pins are subjected to a shear stress force applied by the weight of the ejection system.

Resistance condition:

$$\tau = \tfrac{F}{n \times S} \leq R_{pg}$$

$$R_{pg} = \frac{520}{2} \times 0.8 = 208N/mm^2 \tag{41}$$

Where:

R_e: of 100Cr6 is: 520 Mpa
F: weight of the ejection system (F=10097.22N)
n = 4.
d = 14mm.
s = 2.

$$S = \frac{3.14 \times 14^2}{4} = 153.86mm^2 \tag{42}$$

$$\tau = \frac{10097.22}{4 \times 153.86} = 16.40N/mm^2 \tag{43}$$

Resistance verified.

0.29.4 Screws resistance to shear stress verifications

0.29.4.1 Fixing screws of top clamp plate with core holder block

Resistance condition:

$$\tau = \tfrac{F}{n \times S} \leq R_{pg}$$

$$R_{pg} = \frac{335}{2} \times 0.7 = 117.25N/mm^2 \tag{44}$$

Where:

R_e of C35 is 335 Mpa
F: weight of the core holder block and its components (F= 26546.62N)
n = 4.
d = 10mm.
k = 2.

$$S = \frac{3.14 \times 10^2}{4} = 78.5mm^2 \tag{45}$$

$$\tau = \frac{26546.62}{4 \times 78.5} = 84.54N/mm^2 \tag{46}$$

Resistance verified.

0.29.4.2 Fixing screws of cavity holder block and cavity plate

$$\tau = \tfrac{F}{n \times S} \leq R_{pg}$$

$$R_{pg} = 117.25 N/mm^2 \tag{47}$$

Where:

F: weight of the cavity plate and its components (F=9432.11N)
n = 4.
d = 8mm.

$$S = \frac{3.14 \times 8^2}{4} = 50.24 mm^2 \tag{48}$$

$$\tau = \frac{9432.11}{4 \times 50.24} = 46.94 N/mm^2 \tag{49}$$

Resistance verified.

0.29.4.3 Fixing screws of bottom clamp plate and spacer plates, cavity holder block and cavity plate and their components

$$\tau = \tfrac{F}{n \times S} \leq R_{pg}$$

$$R_{pg} = 117.25 N/mm^2 \tag{50}$$

Where:

F: weight of the cavity plate + cavity holder block + spacer plates + their components (F= 52288.43N))
n = 4.
d = 12mm.

$$S = \frac{3.14 \times 12^2}{4} = 113.04 mm^2 \tag{51}$$

$$\tau = \frac{52288.43}{4 \times 113.04} = 115.64 N/mm^2 \tag{52}$$

Resistance verified.

0.30 Spring selection

We use springs for the main purpose to ensure the return ejection system to its initial position in order to avoid the closing force of the mold to be applied on the return pins.

The choice of springs depends mainly on :

- The ejection stroke which ensures the demolding of the parts (130mm)
- The load to be supported (weight of the ejection system P= 10068.74g)
- The overall dimensions.

The return force (F) that a spring must exert is equal to the total weight of the ejection system over the number of springs used, so:

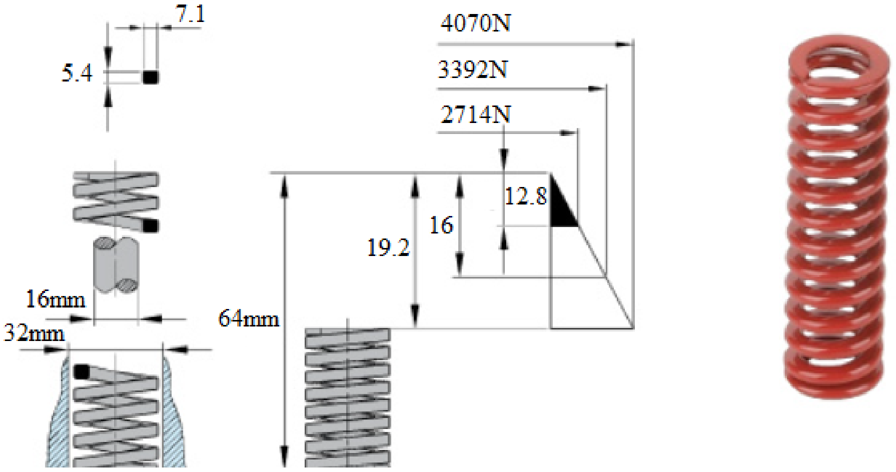


Figure 90: Spring dimensions.

$$F = \frac{10068.74}{4} = 2517,185N \tag{53}$$

For the dimensioning of the spring that will support the force (F), it is necessary to consult charts. These charts classify the springs by color, which means the type of load as shown in the following figure.

Figure 91: Spring clasification by color.

The spring that meets our requirements is a red spring.

0.31 Conclusion

This part allowed us to choose the press to be used and also to check the calculation of the dimensioning of the mold, the verification of the resistance of the various elements acting during the opening and closing of the mold which will ensure a good realization.

SIMULATIONS

0.32 Introduction to CAD/CAM

Welcome to the world of CAD/CAM (Computer Aided Design/Computer Aided Manufacturing). CAD/CAM systems have revolutionized design and manufacturing techniques. Designers no longer have to solve mathematical equations to calculate tangents, intersections, positions, or complex surfaces. The use of computers for geometric design and the generation of numerical control (NC) programs provides almost immediate realization and modification.

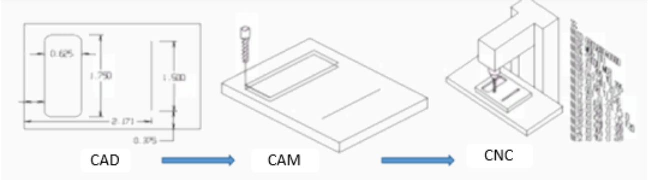


Figure 92: Transition from CAD to NC.

0.32.1 Computer Aided Design (CAD)

0.32.1.1 Definition of CAD

The computer aided design (CAD) is the set of software and techniques of geometric modeling that allows to design and test with a computer, digital simulation techniques of numerical simulation, to carry out manufactured products and the tools to manufacture them.

0.32.1.2 Areas of CAD

The use of CAD applies to all sectors, regardless of the production tools. Regardless of the discipline, CAD software is business-oriented with integrated functionalities.

Therefore, CAD software can be found in areas such as:

- Mechanics,
- Electronics,
- Architecture and construction,
- Furniture,
- Clothing,
- Orthopedics and prosthetics... etc.

0.32.1.3 The use of CAD

The CAD allows to conceive and thus to draw in 3 dimensions simple basic elements, then to elements, then to assemble them in order to realize more or less complex sets. All these basic elements are stored in databases and can therefore be reused in each new study by all the users of the system. It allows standardization of the products. It requires, however, to well organize and the referencing of the sets and sub-sets thus created.

During the design process, the products are visualized in 3D. The visual control of the assemblies is thus facilitated, especially since there are simulation modules that to validate the chosen solutions.

As all the elements that make up the product are contained in the files generated by the CAD software, it is then possible to automatically create the documents necessary for manufacturing, i.e. drawings and bills of material for sub-assemblies and assemblies.

The CAD files can then be exported to CAM software to create NCM (Numerically Controlled Machine Tools) control programs.

0.32.1.4 The benefits of CAD

The use of a tool such as Computer Aided Design does not necessarily mean to have all the other digital tools. CAD can be a first step towards digital tools for a small structure which starts to be equipped. It only requires the acquisition of computers and a CAD software.

It will thus allow to conceive the products by profiting from the advantages of the CAD such as :

- Design of complete sets integrating the management of assemblies in 3D,
- Simulation allowing the validation of adopted solutions,
- Collaborative data management, standardization,
- Drawing and generation of bills of material, CAM links.

0.32.1.5 Hardware and software used for CAD

There are countless CAD tools, the ideal being to have a CAD/CAM tool that integrates design and manufacturing (Computer Aided Design and Manufacturing).

Example:
- PTC CREO PARAMETRIC,
- ESPRIT,
- SOLID EDGE,
- ALIBRE DESIGN,
- CATIA, DRAFTSIGHT AND SOLIDWORKS,
- FREECAD,
- INVENTOR,
- KOMPAS 3D V10... etc.

0.32.2 Computer Aided Manufacturing (CAM)

0.32.2.1 Definition of CAM

CAM (Computer Aided Manufacturing) consists in defining tool paths (machining) on a geometry created in CAD, specifying the tools and machining parameters required. The advantage of this method is that it eliminates most programming errors thanks to the advanced machining verification functions (simulation, solid verification).

0.32.2.2 Areas of CAM

The use of CAM applies to all sectors of the trade with numerically controlled machines.

We will therefore find it in industries such as:
- Metals and alloys (cutting, all machining),
- Wood (cutting, all machining),
- Stone (cutting, all machining),
- Textile, leather (cutting optimization),
- Glass (cutting, all machining),
- Plastics (cutting, all machining),
- Electronics (printed circuits, insertion and soldering of components)
- Assemblies...

0.32.2.3 The use of CAM

The design of the part to be manufactured is carried out using a 3D software package of Computer Aided Design. The resulting file is called a ”3D model”.

This three-dimensional model of the part to be manufactured is then exported, i.e. translated into a language that can be understood by CAM software. Some CAM tools are able to directly read back files from major CAD suppliers. In other cases, CAD and CAM are completely integrated and do not require any transfer. For These software packages are called CAD/CAM (Computer Aided Design and Manufacturing).

Once the 3D model has been imported into the CAM software package and then read back into it, it is possible to move on to programming the toolpaths, which is the core activity of CAM. From the machine data that was initially entered, the software creates the toolpaths, respecting the choice of tool, cutting and feed speeds and the machining operations to be carried out.

It is now possible to fully model the machine tool and to visualize the movements of its moving parts (head, table, rotary axes) during the machining simulation. This possibility is invaluable when verifying and validating 5-axis toolpaths where the risks of collision are high.

The last step consists in generating the instructions for the machine tool and transmitting them to the machine tool and transmit them to the machine via a floppy disk, USB key or via a digital network. The instructions are finally executed by this machine, after the phase of necessary adjustments.

0.32.2.4 The benefits of CAM

The main advantage of CAM software lies in the fact that once the design of the products is completed and validated, the dimensions and shapes of the parts to be machined from one software to another in a completely automatic way, without any risk of modification or error at the time of transcription.

The machining phases are also automated, except for the loading and unloading of the parts (which can sometimes be automated) and of course the adjustments and calibrations which always require the presence of an operator.

The implementation of CAD/CAM corresponds to a search for productivity. It allows, for example in the case of material flow, to limit significantly the losses of raw material.

0.32.2.5 Hardware and software used for CAM

There are innumerable CAM tools, specific to each sector, because they are often recommended by the machine manufacturers. The ideal is to have a CAD/CAM tool that integrates design and manufacturing, we quote :

- CAMWORKS,
- CATIA,
- POWERMILL,
- SURFCAM,
- GTL from MISSLER,
- MARTCAM,
- CADKEY.

0.32.3 Numerical Controlled Machines

Numerical control is a technique that uses data composed of alphanumeric codes to represent the geometric and technological instructions necessary to drive a machine or a process. It is also a method of automating the functions of machines, the main characteristic of which is that it is very easy to adapt to different jobs.

As such, NC is one of the best examples of the penetration of information processing in production activities. By making full use of the possibilities of microcomputing, all data is processed in real time, i.e. as it is generated, so that the results of the processing also help to control the process.

After a first generation of NC systems with hardwired logic, we have seen the emergence of computer numerical controls (CNC), which integrate one or more specific computers to perform all or part of the control functions. All CNC systems on the market today contain at least one microprocessor.

Currently the main manufacturers of NCMs are :
- NUM,
- FANUC,
- DMU,
- HEIDENHAIN,
- MAKINO,
- MAZAK,
- DMG,
- HERMIL,
- CINCINNATI.

0.33 Injection molding simulation

0.33.1 SolidWorks Plastics

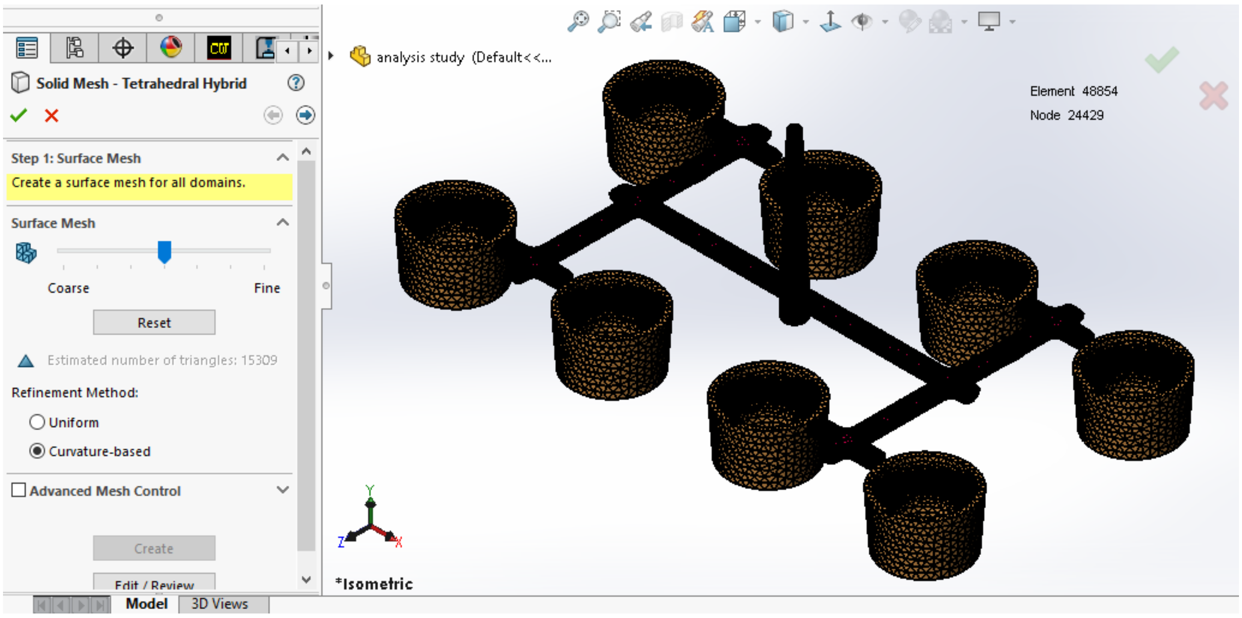
SolidWorks is a computer program for solid modeling in computer-aided design (CAD) and computer-aided engineering (CAE). The simulation of the injection

molding process is performed by SolidWorks Plastics. This module's major goal is to predict manufacturing flaws in parts and molds. Three types of analysis are used in the injection molding simulation:
1. Flow analysis: This predicts how the material will flow into the mold cavity.
2. Pack analysis: This simulates the solidification of the material in the mold cavity.
3. Warp analysis: Estimates the shape and dimensions of the part after it has been ejected and cooled.

0.33.2 Mesh Generation

The first stage of the simulation preparation consists of the part's meshing. The meshing procedure is of critical importance for the accuracy of the simulation.

SolidWorks Plastics provides two options for meshing. The 'Auto' option selects the default size of the solid element based on the cavity size and thickness, and the 'Manual' option for a full control over the meshing parameters in a step-by-step process, we choose the 'Auto' option.



Solid Mesh - Tetrahedral Hybrid

Step 1: Surface Mesh

Create a surface mesh for all domains.

Surface Mesh

Estimated number of triangles: 15309

Refinement Method:

○ Uniform

● Curvature-based

☐ Advanced Mesh Control

analysis study (Default<<...

Element 48854

Node 24429

Figure 93: Mesh

0.33.3 Material

In our study, the polymer used for the injection molding of the part is PP (Polypropylene).

SolidWorks Plastics includes a comprehensive database with numerous plastics sorted by company or by polymer family. We choose the polymer from the database.

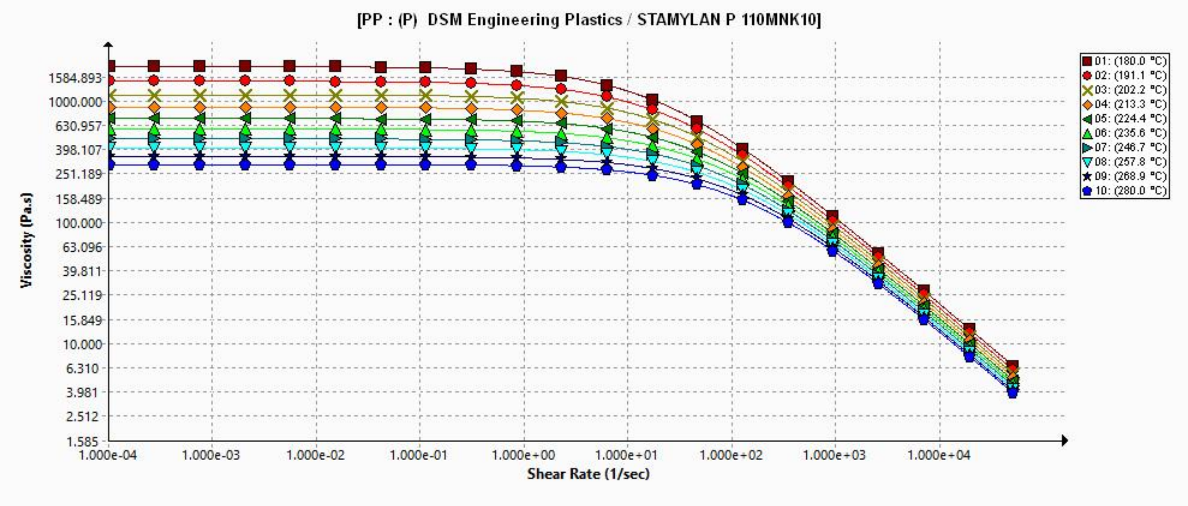
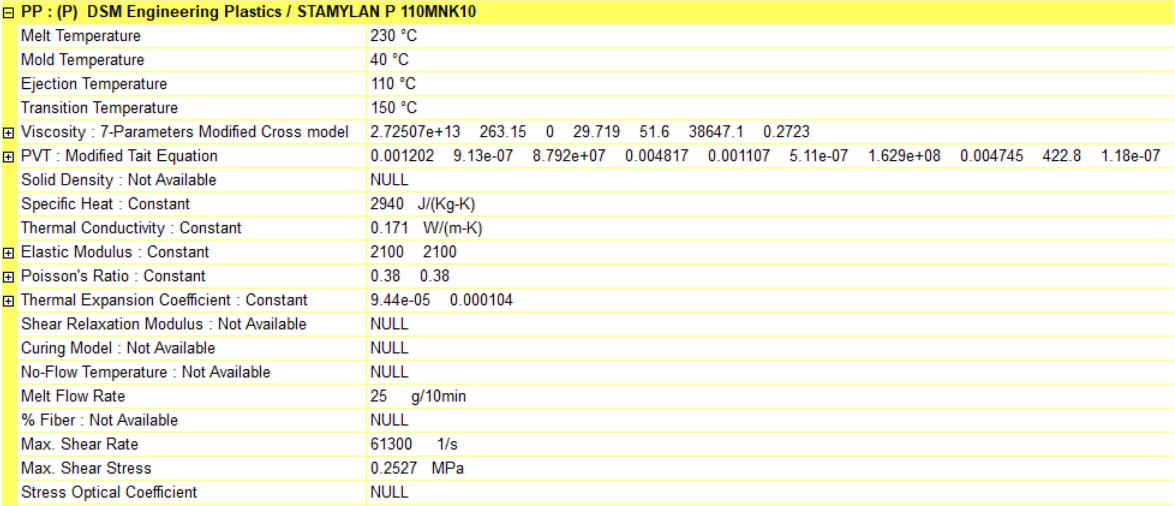


Figure 94: Polypropylene viscosity - Shear stress diagram

The polymer material parameters are given below.



⊟ PP : (P) DSM Engineering Plastics / STAMYLAN P 110MNK10	
Melt Temperature	230 °C
Mold Temperature	40 °C
Ejection Temperature	110 °C
Transition Temperature	150 °C
⊞ Viscosity : 7-Parameters Modified Cross model	2.72507e+13 263.15 0 29.719 51.6 38647.1 0.2723
⊞ PVT : Modified Tait Equation	0.001202 9.13e-07 8.792e+07 0.004817 0.001107 5.11e-07 1.629e+08 0.004745 422.8 1.18e-07
Solid Density : Not Available	NULL
Specific Heat : Constant	2940 J/(Kg-K)
Thermal Conductivity : Constant	0.171 W/(m-K)
⊞ Elastic Modulus : Constant	2100 2100
⊞ Poisson's Ratio : Constant	0.38 0.38
⊞ Thermal Expansion Coefficient : Constant	9.44e-05 0.000104
Shear Relaxation Modulus : Not Available	NULL
Curing Model : Not Available	NULL
No-Flow Temperature : Not Available	NULL
Melt Flow Rate	25 g/10min
% Fiber : Not Available	NULL
Max. Shear Rate	61300 1/s
Max. Shear Stress	0.2527 MPa
Stress Optical Coefficient	NULL

Figure 95: Polymer material parameters

0.33.4 Process Parameters

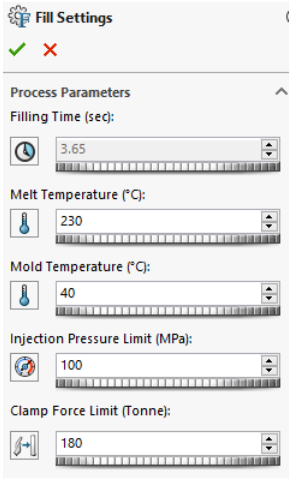
Process parameters can be modified and displayed through this submenu.

0.33.4.1 Fill Settings

The Fill Settings define the injection molding process parameters such as Filling time, Melt temperature, Mold temperature, and Injection pressure limit.
SolidWorks Plastics calculates the volume of the part and advises the user on how long the molten plastic will take to inject and completely fill the mold. The resin manufacturer's advice on how to operate the injection molding machine with this particular material are contained in the melt and mold temperature boxes. The user can either accept the default values or change them manually. The Filling time is accepted with the default estimated value for our study, and the remaining parameters are altered according to the manufacturer's specifications.

0.33.4.2 Pack Settings

The goal of packing is to produce a part with uniform weight and dimensional integrity. A successful packing improves the part quality.
SolidWorks Plastics estimates "Pressure Holding Time" required for the packing stage and "Cooling Time" required for the pure cooling stage during the simulation process.
The user can either accept those default values or manually adjust those. For our study, the default calculated values are accepted for the verification analysis of the part.



Filling Time (sec): 3.65

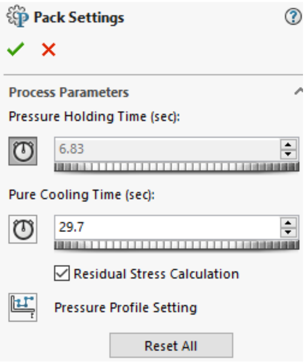
Melt Temperature (°C): 230

Mold Temperature (°C): 40

Injection Pressure Limit (MPa): 100

Clamp Force Limit (Tonne): 180

Figure 96: Fill Settings



Pressure Holding Time (sec): 6.83

Pure Cooling Time (sec): 29.7

☑ Residual Stress Calculation

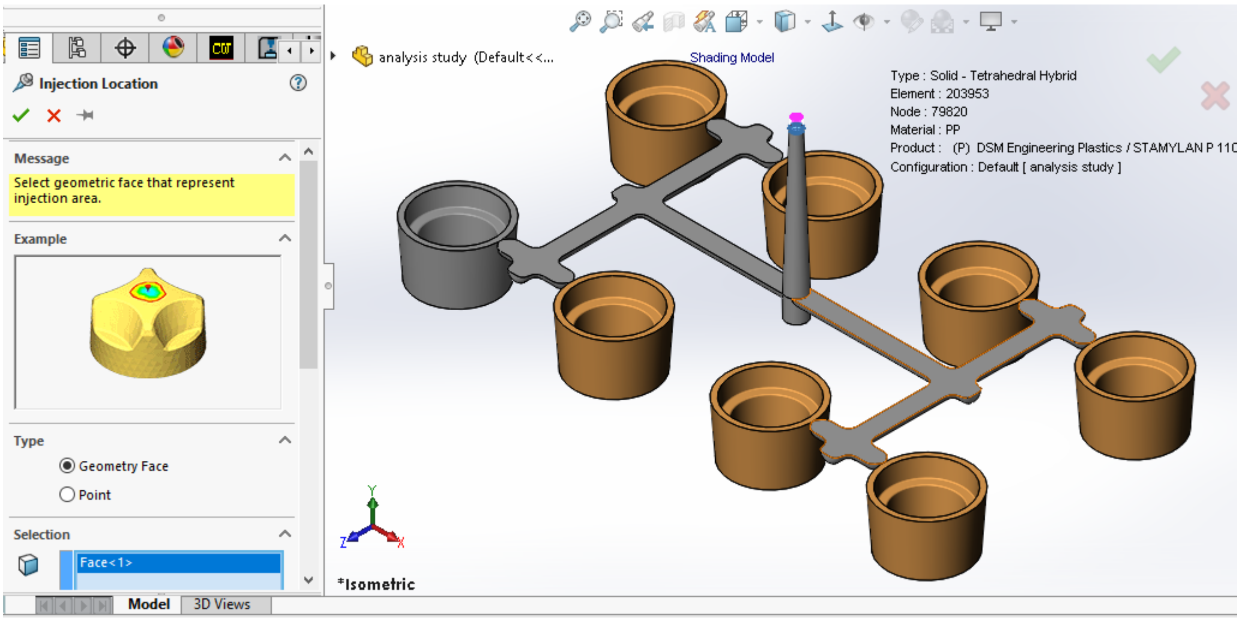
Figure 97: Pack Settings

0.33.5 Boundary Conditions

In our study, the boundary condition required for the simulation is to define the injection location.

0.33.5.1 Injection Location

Polymer material at the specified Melt Temperature is introduced into the cavity through injection locations.



Message

Select geometric face that represent injection area.

Type

● Geometry Face

○ Point

Selection

Face<1>

Type : Solid - Tetrahedral Hybrid
Element : 203953
Node : 79820
Material : PP
Product : (P) DSM Engineering Plastics / STAMYLAN P 11C
Configuration : Default [analysis study]

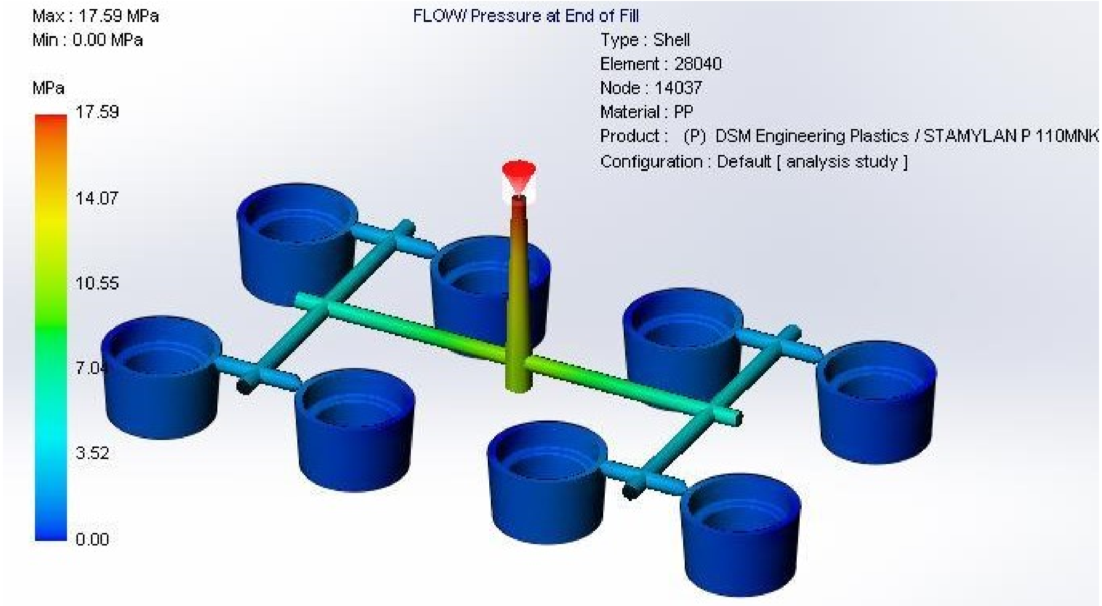
Figure 98: Injection Location

0.33.6 Results

Based on the simulation results, the injection was successful with a maximum pressure of 17.58 Mpa and the filling time took around 4.18s.

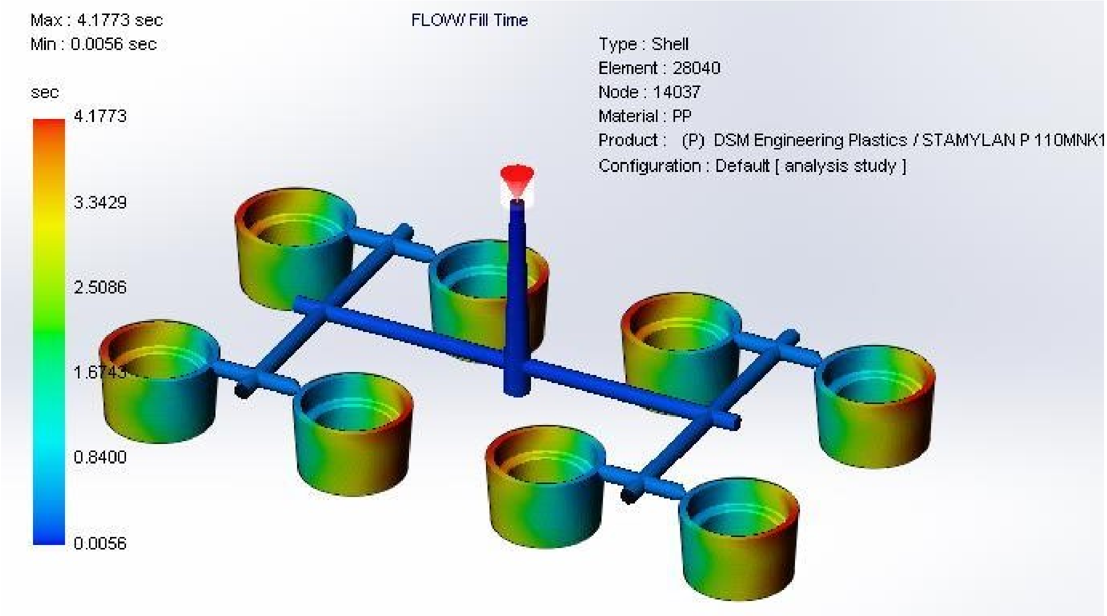
Figures 5.8 5.9 demonstrate the previous results obtained with SolidWorks platics.

The full results report of the injection simulation generated with SolidWorks Plastics is found in the appendices section.



Type : Shell
Element : 28040
Node : 14037
Material : PP
Product : (P) DSM Engineering Plastics / STAMYLAN P 110MNK
Configuration : Default [analysis study]

Figure 99: Injection pressure at end of fill



Type : Shell
Element : 28040
Node : 14037
Material : PP
Product : (P) DSM Engineering Plastics / STAMYLAN P 110MNK1
Configuration : Default [analysis study]

Figure 100: Injection fill time

0.34 Stress analysis

0.34.1 ABAQUS

Today, product simulation is often being performed by engineering groups using niche simulation tools from different vendors to simulate various design attributes. The use of multiple vendor software products creates inefficiencies and increases costs. SIMULIA delivers a scalable suite of unified analysis products that allow all users, regardless of their simulation expertise or domain focus, to collaborate and seamlessly share simulation data and approved methods without loss of information fidelity.

The Abaqus Unified FEA product suite offers powerful and complete solutions for both routine and sophisticated engineering problems covering a vast spectrum of industrial applications. In the automotive industry engineering work groups are able to consider full vehicle loads, dynamic vibration, multibody systems, impact/crash, nonlinear static, thermal coupling, and acoustic-structural coupling using a common model data structure and integrated solver technology. Best-in-class companies are taking advantage of Abaqus Unified FEA to consolidate their processes and tools, reduce costs and inefficiencies, and gain a competitive advantage.

0.34.2 Analysis steps

1-Importing parts:

During this step we import the parts exported from the Software SolidWorks into ABAQUS including (the 2 clamp plates, the 2 holder plates and the 2 spacer plates)
To do that we follow the steps below:
File → Import → File filter (.step) → Select Part → OK → OK

2-Creating materials:

In this step, we create two materials for our parts, C30 for the clamp plates and the spacer plates and the 25CrMo4 for the holder plates.
To do that we follow the steps below:

Property → Create material → Name (Low alloyed steel C) → General → Density (7700E-12) → Mechanical → Elasticity → Elastic → Young's Modulus (200E3) → Poisson's ratio (0.3)

Property → Create material → Name (Low alloyed steel Cr) → General → Density (7800E-12) → Mechanical → Elasticity → Elastic →

Young's Modulus (210E3) → Poisson's ratio (0.3)

3-Creating sections:

In this step, we create two materials for our parts using the steps below:

Create section → Name (Clamp and Spacer plates) → Category (Solid) → Type (Homogeneous) → Continue → Material (low alloyed steel C) → Ok

Create section → Name (Cavity and Core holder plates) → Category (Solid) → Type (Homogeneous) → Continue → Material (low alloyed steel Cr) → Ok

4-Assign sections:

In this step, we assign each section to its proper part using the steps below:

Assign section → select the Part → Done (Solid) → Section (select material)→ ok

5-Assembly:

In this step, we add parts into the assembly and move them around with the constraints untill they are in their proper place using the steps below:

Assembly → Create instance → Select part → Instance type (Dependent)→ ok

6-Create Step:

In this step, we create a static step for our analysis using the steps below:

Step → Create step → Name (Step-1) → type (Static, General)→ Continue Basic (Nlgeom ON) → Ok

7-Interactions:

In this step, we specify the interactions between the parts surfaces in the assembly using the steps below:

Interactions → Create interaction property → Name (IntProp-1) → type (Contact)→ Continue → Mechanical → Tangential behavior → Friction Formulation (Penalty) → Friction Coeff (0.2) → Ok

Create Interaction → Name (Int-1) → Step (Initial) → Type (General contact)

8-Load:

In this step, we specify the interactions between the parts surfaces in the assembly using the steps below:

Create Load → Name (load-1) → Step (Step-1) → Category (Mechanical) → Type (Pressure)→ Continue → Select bottom surface of the bottom clamp plate → Done → Magnitude (1.8) → Ok

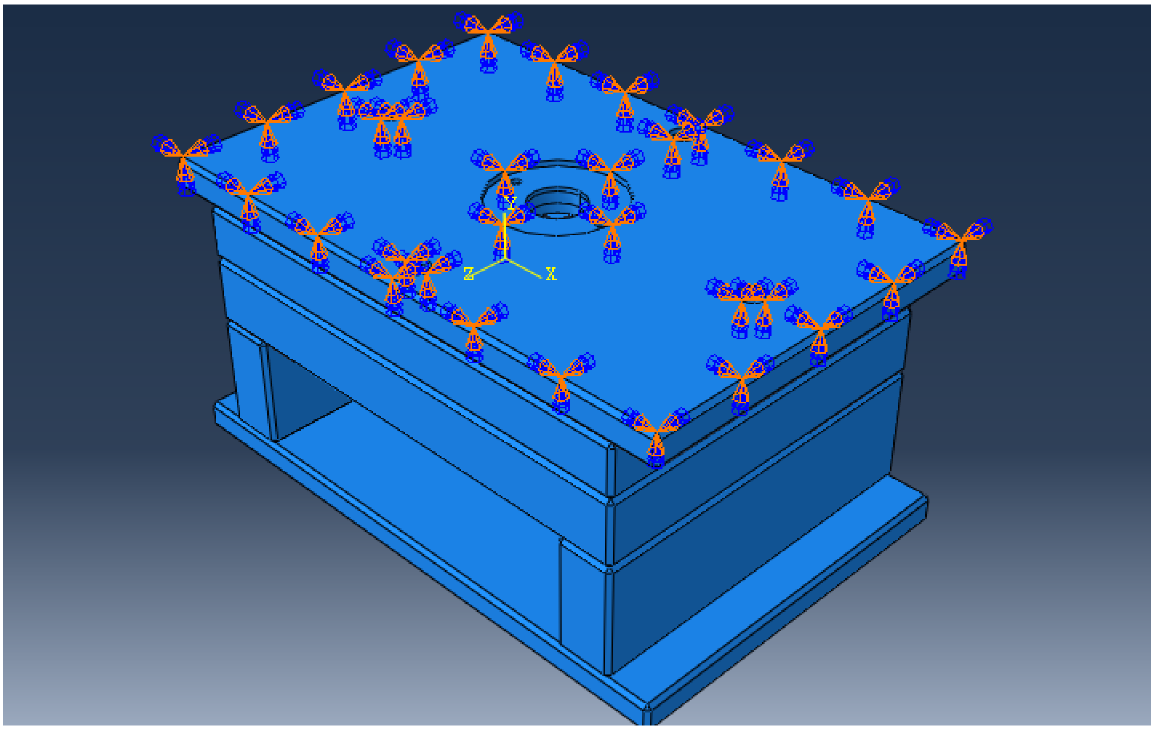


Figure 101: Load on assembly

Create boundary condition → Name (BC-1) → Step (Initial) → Category (Symmetry/Antisymmetry/ancastre) → Continue → Select surface → Done → Check ENCASTRE → Ok

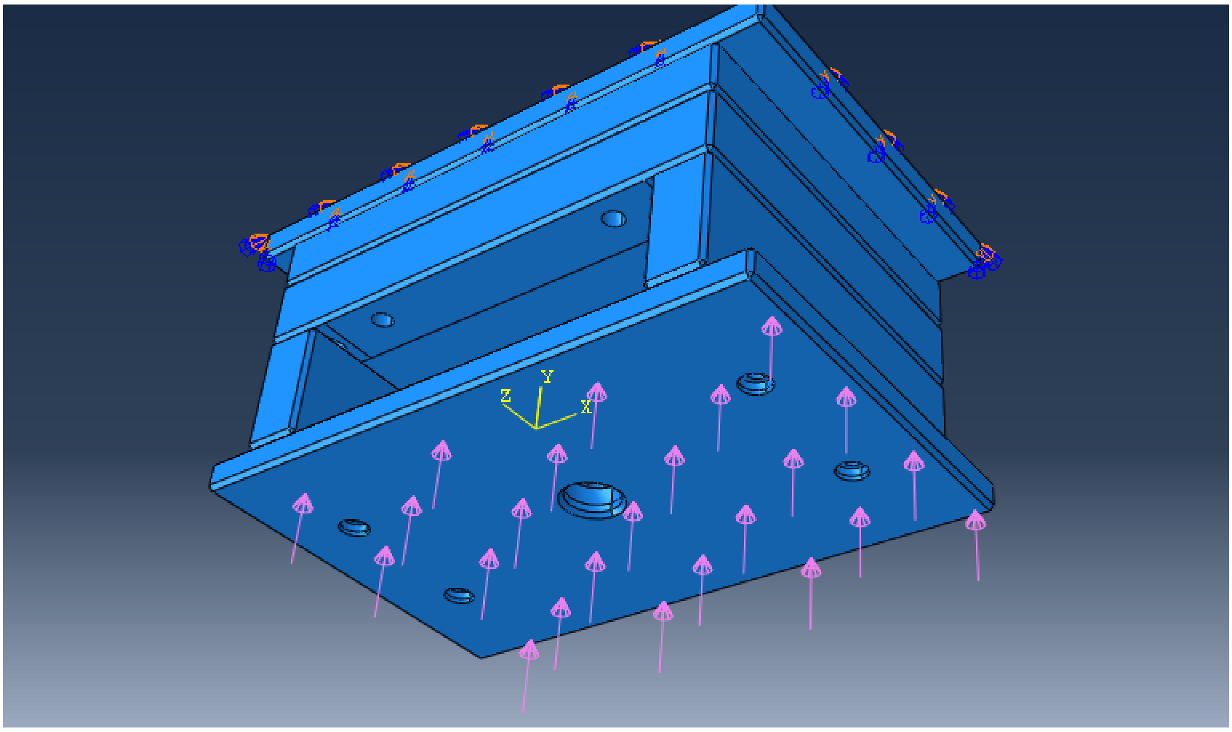


Figure 102: Boundary condition on assembly

9-Mesh:

In this step, we mesh every part individualy after creating the proper partitions.
the steps below demostrate the general way of meshing our complex part:

Mesh → Create partition → Type (cell) → Method (Define cutting plane)

Seed part → Global size (20) → Ok

Mesh part → Yes

10-Job:

The final step in the analysis is to create a job and submit it following the steps below:

Job → Create Job → Name (Job-1) → Continue → Ok

Job Manager → Submit

0.34.3 Results

0.34.3.1 Displacement results

The first set of results that we were interested in were the displacement values of the mold components when under clamping pressure. These values were important to verify if the displacement value is negligible compared to the component thickness.

The bottom clamp plate was the component that showed the largest displacement which reached 0.9mm which was negligible compared to its thickness. Same case was found for the rest of the components.

Pictures from 5.12 through 5.17 show the displacement values for each components of the assembly.

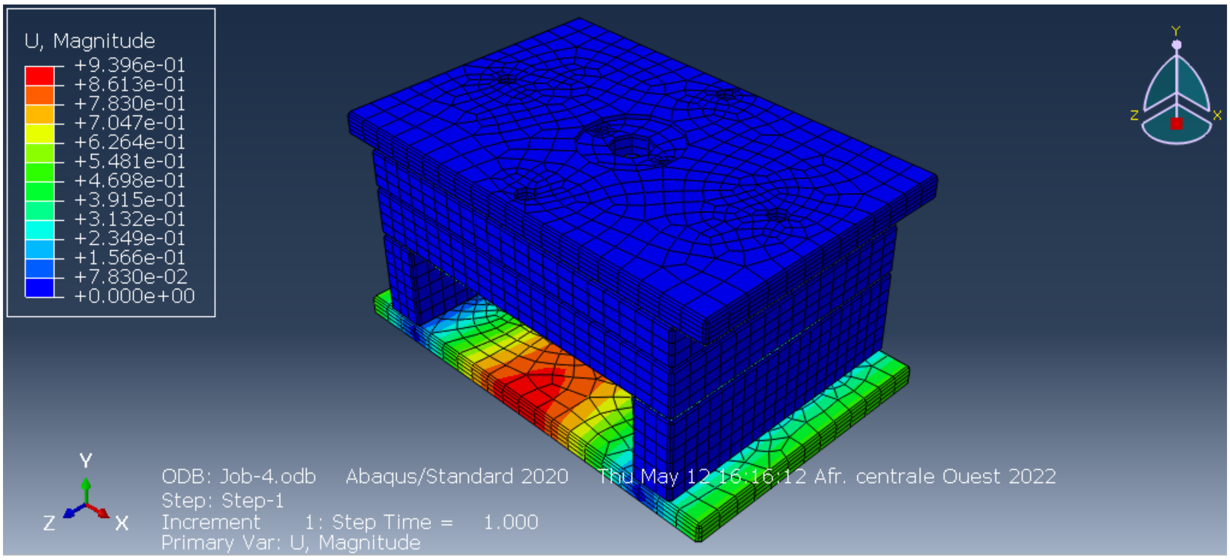


Figure 103: Displacement

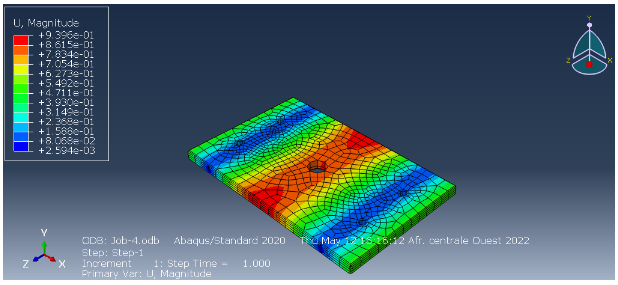


Figure 104: Bottom calamp plate displacement

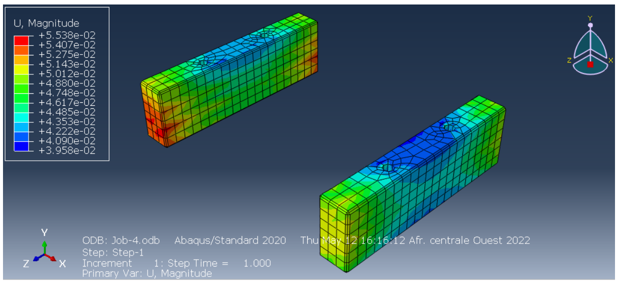


Figure 105: Spacer plates displacement

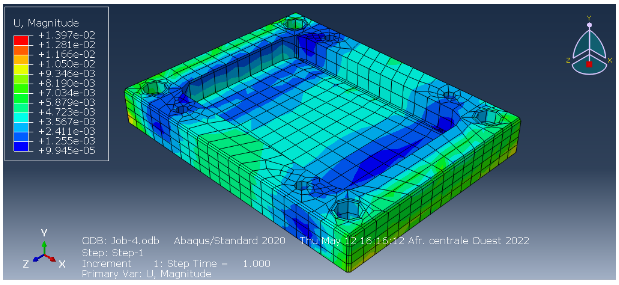


Figure 106: Holder block displacement

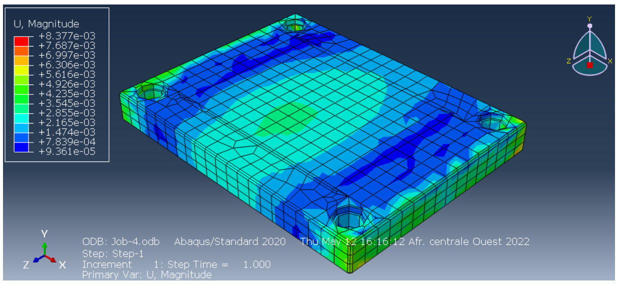


Figure 107: Holder plate displacement

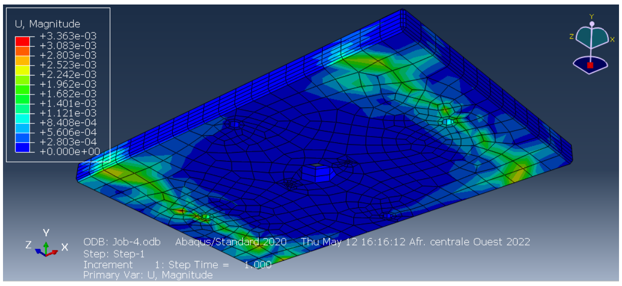


Figure 108: Top clamp plate displacement

0.34.3.2 Stress results

Another important results we needed to verify were the displacement values for all of the mold components.

The displacement analysis was important to find the stress of a components under the clamping force with taking into consideration the proper boundary condition area, the component on which the load is applied and the contact between all the mold components.

Pictures from 5.18 through 5.23 show the stress on each component of the assembly.

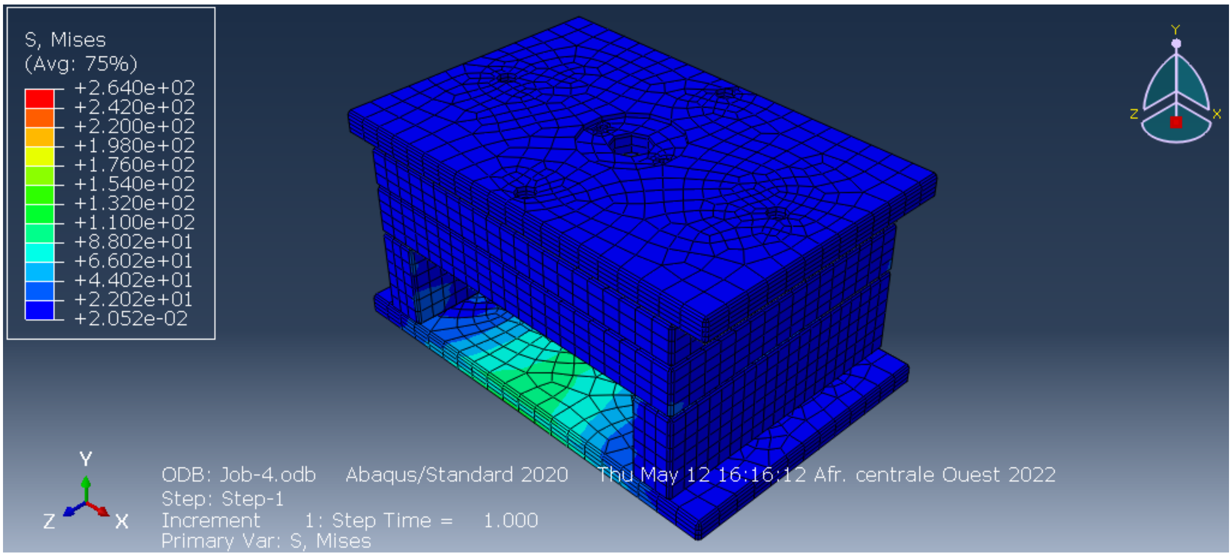


Figure 109: Stress

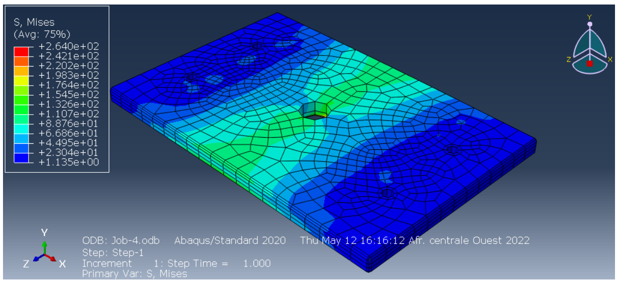


Figure 110: The strss on the botoom clamp plate

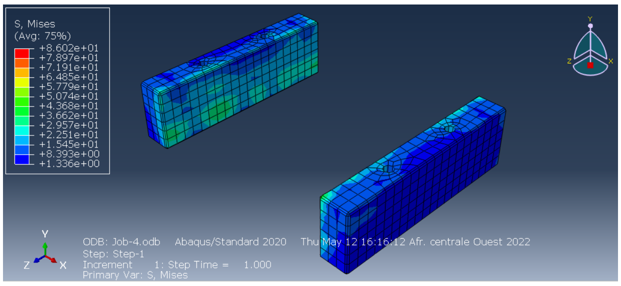


Figure 111: The strss on the spacer plates

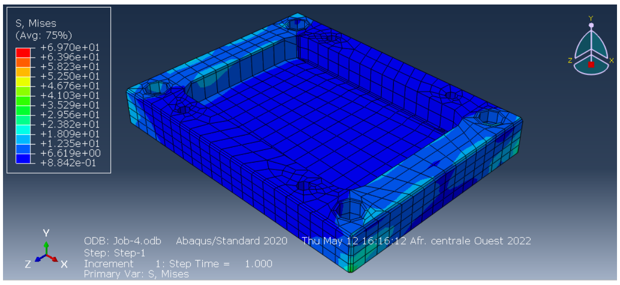


Figure 112: The strss on the holder block

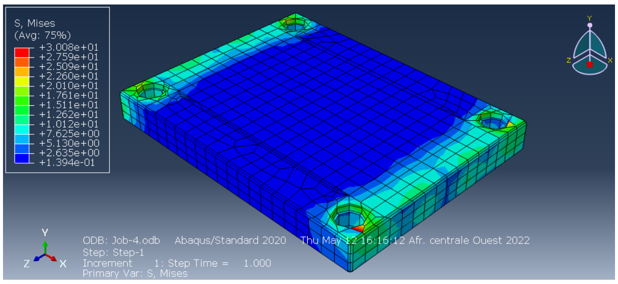


Figure 113: The strss on the holder plate

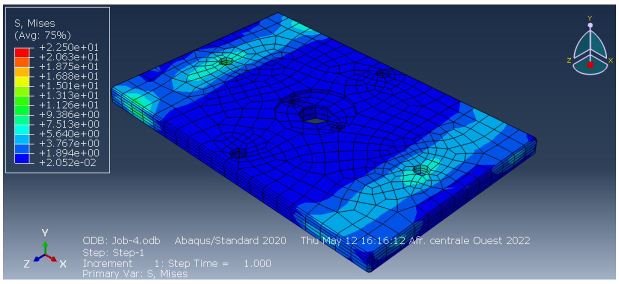


Figure 114: The strss on the top clamp plate

0.34.3.3 Reaction forces results

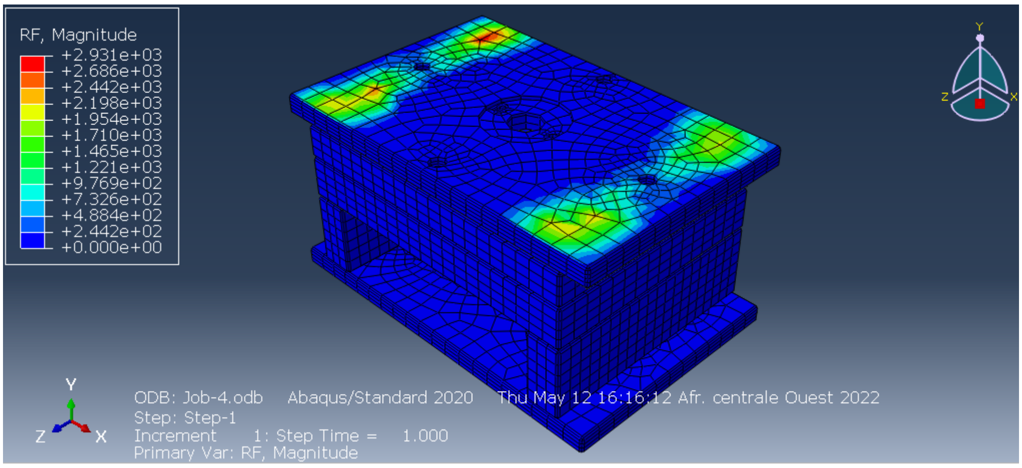


Figure 115: Reaction forces

0.35 Computer-aided manufacturing

0.35.1 CAMWorks

CAMWorks was the first fully integrated computer-aided manufacturing (CAM) solution designed exclusively to operate in SOLIDWORKS and the first to offer knowledge-based, feature recognition, and associative machining capabilities within SOLIDWORKS.CAMWorks uses the same SOLIDWORKS geometry to generate toolpaths to ensure the part you machine is the same part you've modeled. Toolpath simulation allows users to compare "as designed" parts to "as machined" parts. The part model-to-toolpath associativity in CAMWorks increases machining productivity and lowers manufacturing costs by eliminating time-consuming CAM system rework due to design updates.CAMWorks eliminates the drudgery of CNC programming with Intelligent Machining through automation. Pioneered by Geometric Technologies, this suite of tools automates the generation of toolpaths based on a knowledge-based database. It eliminates hours of complex programming through Automatic Feature Recognition (AFR) that automatically defines prismatic machinable features, while the TechDBTM (Technology Database) defines machining operations to automatically generate accurate toolpaths at the click of a button.CAMWorks machining modules include: 2 Axis Mill, 3 Axis Mill, Turning, Mill-Turn with 5 Axis Simultaneous Machining Support, 4/5 Axis Simultaneous Machining, and Wire EDM[25].

0.35.2 Calculations of speeds and feeds

When manufacturing and machining any product, it is important to know the time used for the work and the feeds and speeds. Usually, machines automatically calculate the estimated time spent at work. Engineers and manufacturers are usually interested in calculations of speeds and feeds. The most desired values are cutting speed, spindle speed and table feed. Cutting speed can be calculated when spindle speed n (r/min) and the tool diameter are known. Spindle speed is frequency of rotation of the machine it rotates around its axis, it is measured by revolutions per minute. Table feed, feed rate or just commonly referred as feed, indicates the speed at which the cutter grips the part. It is measured in (mm/min)[26].

$$CuttingSpeedV_c = \frac{\Pi \times D \times n}{100} \tag{54}$$

$$TableFeedV_f = n \times f_z \times Z \tag{55}$$

Where:

n: Spindle Speed.
f$_z$: Feed per tooth. D: Tool diameter. Z: No. of flutes.

0.35.3 Simulation Steps

1-Machine selection:

CAMWorks NC Manager → Machine → Edit definition → Available machines (Mill-Metric) → OK

2-Stock manager:

Stock manager → Edit definition → Material (P20) → Bounding Box Offset ((X+: 2.5 X-: 2.5 / Y+:1 Y-:4 / Z+: 2.5 Z-: 2.5) → Ok

3-Coordinate system:

Method (User defined →Origin (Stock bounding box vertex) Ok

4-Create Feature example:

Mill part setup1 → Part Peremiter Feature → Feature Type (Open Pocket) → Feature Strategy (Rough-Finish) → End Condition (Bottom of the part) → Ok

5-Generate Operation Plan:

Perimeter Boss Finish → Generate Operation Plan

6-Feature Parameters:

Contour Mill → Edite Definition

Tool → Tool Crib → From library → Add → Tool Type (Flat End 18mm) → Ok

F/S → Spindle SMM(210) → Feed rates (Feed per tooth 0.1mm)

Contour → Side Parameters Allowance (0mm)) → Depth Parameters (First cut amt 2mm Max cut amt 2mm) → Ok

CAMWorks NC Manager → Generate Toolpath

7-Simulation:

CAMWorks NC Manager → Simulate Toolpath

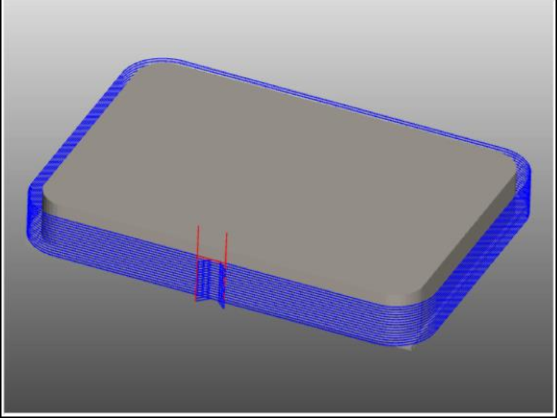
Cavity Plate
Setup Sheet

Material	P20	Author	Marwa DAKICHE	Units	MM
Stock Size	265.00, 35.00, 185.00	Keywords		CNC Mach	Mill - Metric
Company	SOFICLEF	Comments		Programmer	
Date/Time	05/05/2022 20:26:34	Title	Injection Mold	Subject	

Setup No.	**Setup Name**	**Setup Origin**
1	Group1	90.00, 21.23, 40.00

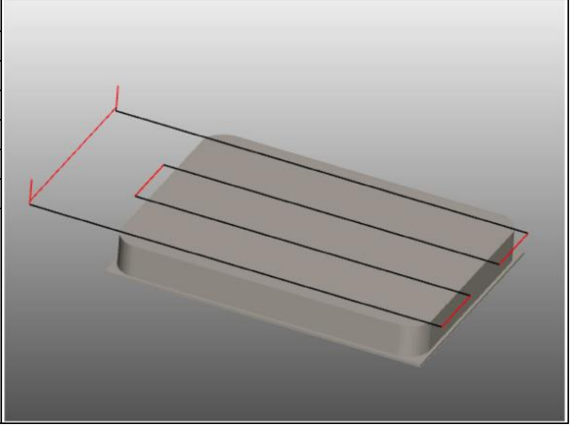
Operation	Contour Mill1
Cutting Speed Vc	210
Feed per tooth (mm)	0.1mm
Z Feed Rate	371.36
Tool Station no.	14
Tool Description	18MM CRB 4FL 38 LOC
Mach Depth	31.00

	Minimum	Maximum
X:	-139.00	139.00
Y:	-99.00	106.15
Z:	-32.00	25.00



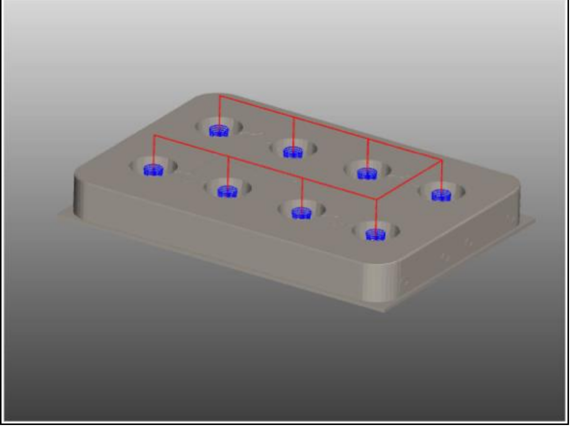
Operation	Face Mill1
Cutting Speed Vc	500
Feed per tooth (mm)	0.1
Z Feed Rate	125.00
Tool Station no.	15
Tool Description	80MM 8FL FACE MILL
Mach Depth	1.00

	Minimum	Maximum
X:	-204.50	140.50
Y:	-76.50	76.50
Z:	-1.00	25.00



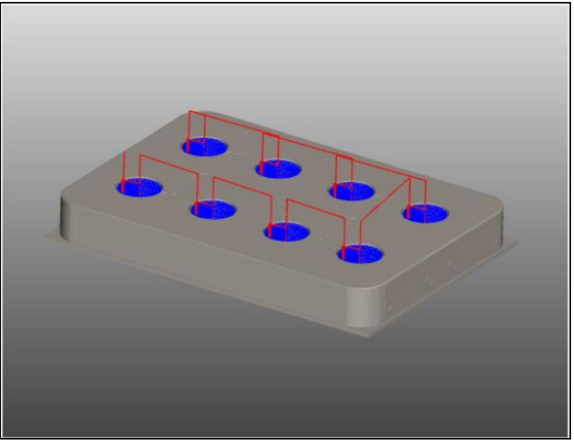
Operation	Rough Mill1
Cutting Speed Vc	210
Feed per tooth (mm)	0.1mm
Z Feed Rate	167.11
Tool Station no.	5
Tool Description	20MM CRB 2FL 38 LOC
Mach Depth	20.40

	Minimum	Maximum
X:	-96.28	96.28
Y:	-46.28	46.28
Z:	-21.40	25.00



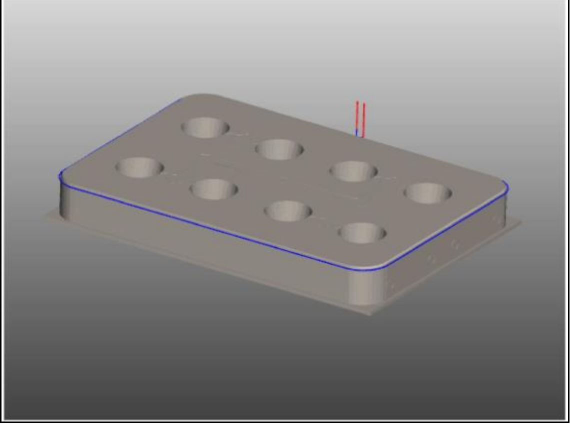
Operation	Contour Mill2
Cutting Speed Vc	150
Feed per tooth (mm)	0.1
Z Feed Rate	1200.00
Tool Station no.	7
Tool Description	4MM CRB 4FL BM 14 LOC
Mach Depth	20.40

	Minimum	Maximum
X:	-104.69	104.69
Y:	-54.69	54.69
Z:	-21.40	25.00



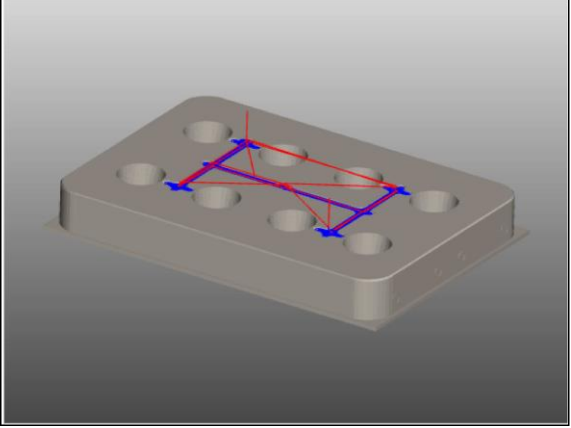
Operation	Contour Mill3
Cutting Speed Vc	180
Feed per tooth (mm)	0.1
Z Feed Rate	1200.00
Tool Station no.	16
Tool Description	5MM HSS 90DEG COUNTERSINK
Mach Depth	10.00

	Minimum	Maximum
X:	-131.00	131.00
Y:	-91.00	92.98
Z:	-3.00	25.00



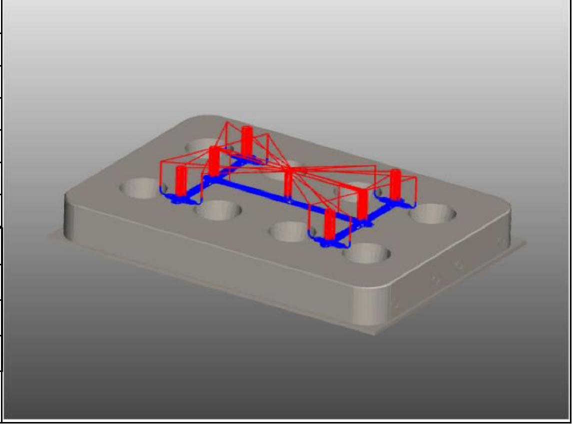
Operation	Area Clearance1
Cutting Speed Vc	113
Feed per tooth (mm)	0.1
Z Feed Rate	960.00
Tool Station no.	17
Tool Description	3MM CRB 2FL 12 LOC
Mach Depth	N.A.

	Minimum	Maximum
X:	-96.25	96.25
Y:	-50.82	50.82
Z:	-23.92	25.00



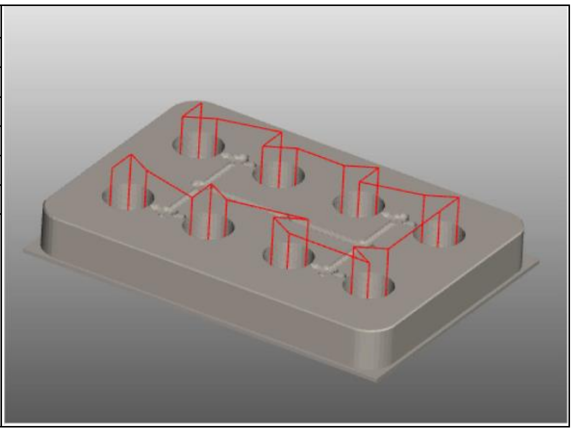
Operation	Z Level1
Cutting Speed Vc	75
Feed per tooth (mm)	0.1
Z Feed Rate	3600.00
Tool Station no.	18
Tool Description	2MM CRB 4FL BM 4 LOC
Mach Depth	N.A.

	Minimum	Maximum
X:	-71.03	71.03
Y:	-46.95	46.95
Z:	-4.70	25.00



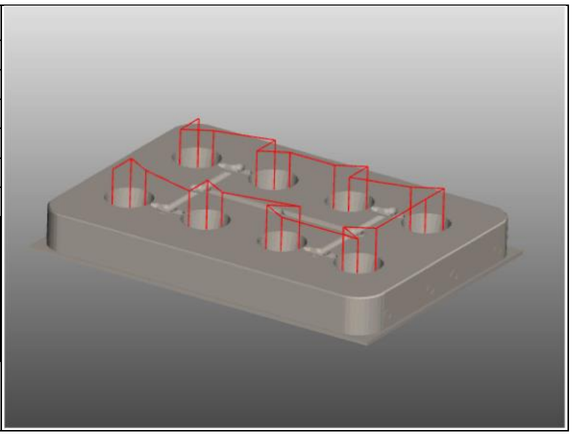
Operation	Center Drill1
Cutting Speed Vc	150
Feed per tooth (mm)	N.A.
Z Feed Rate	61.61
Tool Station no.	6
Tool Description	6MM X 60DEG HSS CENTERDRILL
Mach Depth	2.52

	Minimum	Maximum
X:	-76.63	76.61
Y:	-48.16	48.16
Z:	-4.70	25.00



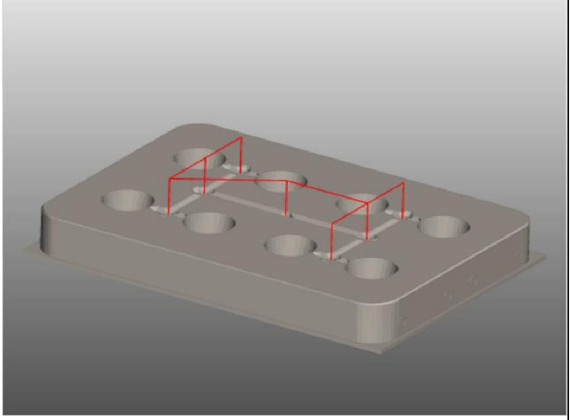
Operation	Drill1
Cutting Speed Vc	213
Feed per tooth (mm)	N.A.
Z Feed Rate	112.37
Tool Station no.	22
Tool Description	3.0mm SCREW MACH DRILL
Mach Depth	10.22

	Minimum	Maximum
X:	-60.00	60.00
Y:	-40.00	40.00
Z:	-32.80	25.00



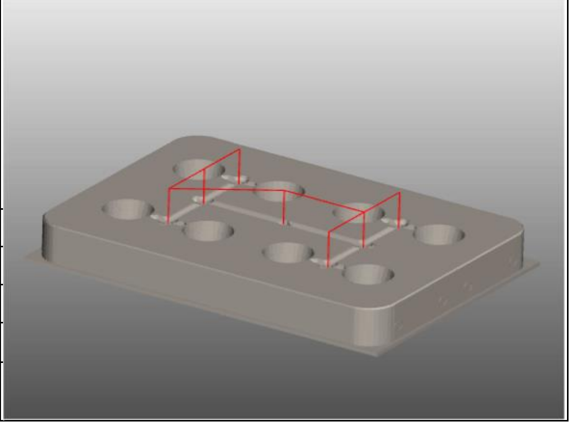
Operation	Center Drill2
Cutting Speed Vc	180
Feed per tooth (mm)	N.A.
Z Feed Rate	61.61
Tool Station no.	6
Tool Description	6MM X 60DEG HSS CENTERDRILL
Mach Depth	2.70

	Minimum	Maximum
X:	-96.25	96.25
Y:	-50.82	50.82
Z:	-31.62	25.00



Operation	Drill2
Cutting Speed Vc	225
Feed per tooth (mm)	N.A.
Z Feed Rate	84.36
Tool Station no.	23
Tool Description	6.0mm JOBBER DRILL
Mach Depth	31.80

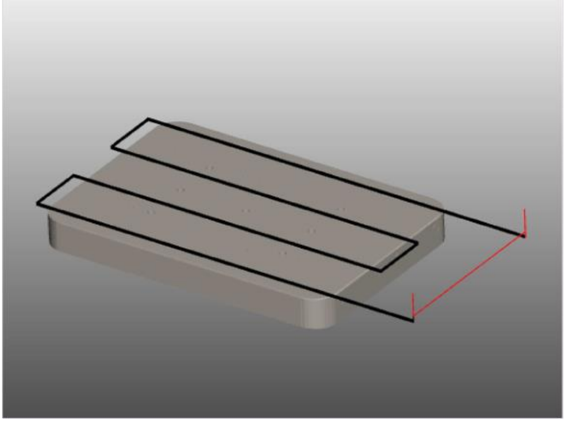
	Minimum	Maximum
X:	-60.00	60.00
Y:	-40.00	40.00
Z:	-3.70	25.00



Setup No.	Setup Name	Setup Origin
2	Group2	90.00, 21.23, 40.00

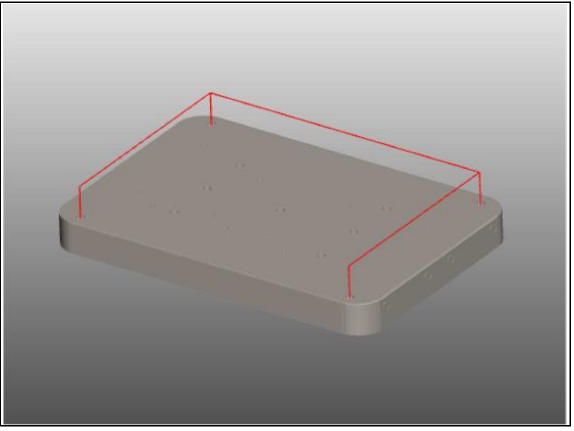
Operation	Face Mill2
Cutting Speed Vc	500
Feed per tooth (mm)	0.1
Z Feed Rate	125.00
Tool Station no.	15
Tool Description	80MM 8FL FACE MILL
Mach Depth	3.50

	Minimum	Maximum
X:	-204.50	140.50
Y:	-76.50	76.50
Z:	31.00	60.00



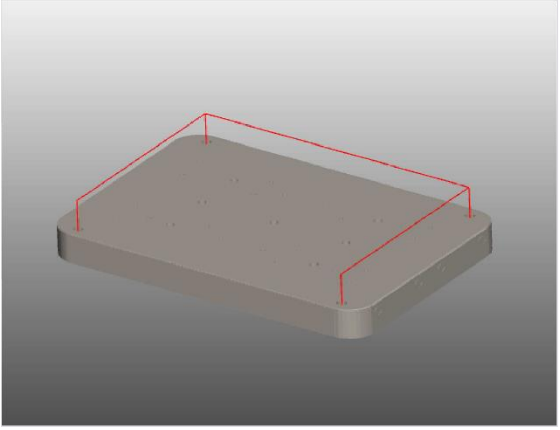
Operation	Center Drill3
Cutting Speed Vc	180
Feed per tooth (mm)	N.A.
Z Feed Rate	61.61
Tool Station no.	6
Tool Description	6MM X 60DEG HSS CENTERDRILL
Mach Depth	7.50

	Minimum	Maximum
X:	-110.00	110.00
Y:	-80.00	80.00
Z:	23.50	60.00



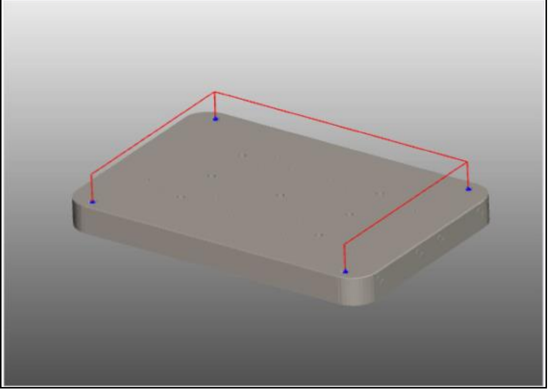
Operation	Drill3
Cutting Speed Vc	250
Feed per tooth (mm)	N.A.
Z Feed Rate	94.63
Tool Station no.	28
Tool Description	7.0mm JOBBER DRILL
Mach Depth	22.10

	Minimum	Maximum
X:	-110.00	110.00
Y:	-80.00	80.00
Z:	8.90	60.00



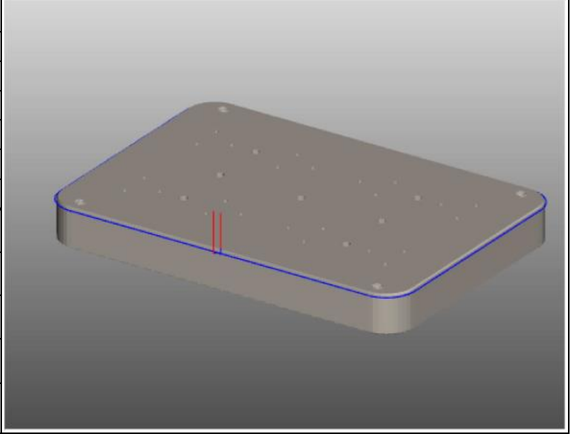
Operation	Thread Mill1
Cutting Speed Vc	180
Feed per tooth (mm)	0.2
Z Feed Rate	2302.30
Tool Station no.	29
Tool Description	M6-M7 CRB SP THREAD MILL
Mach Depth	20.00

	Minimum	Maximum
X:	-131.00	131.00
Y:	-92.98	91.00
Z:	29.00	60.00



Operation	Contour Mill4
Cutting Speed Vc	180
Feed per tooth (mm)	0.1
Z Feed Rate	64.07
Tool Station no.	11
Tool Description	5MM HSS 90DEG COUNTERSINK
Mach Depth	30.00

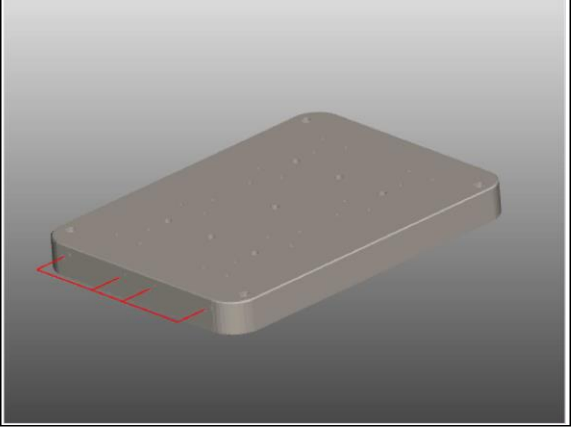
	Minimum	Maximum
X:	-111.57	111.57
Y:	-81.57	81.57
Z:	10.87	60.00



Setup No.	Setup Name	Setup Origin
3	Group3	90.00, 21.23, 40.00

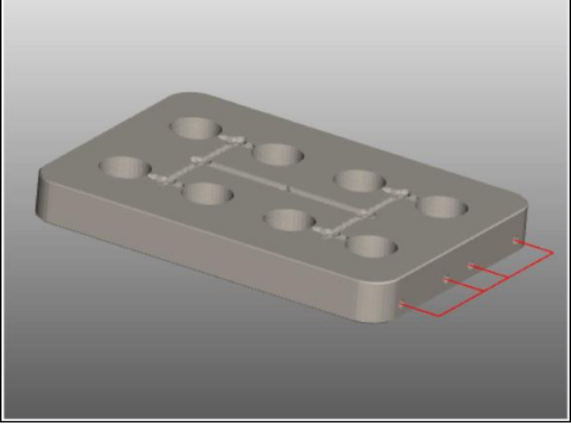
Operation	Center Drill6
Cutting Speed Vc	180
Feed per tooth (mm)	N.A.
Z Feed Rate	61.61
Tool Station no.	6
Tool Description	6MM X 60DEG HSS CENTERDRILL
Mach Depth	2.70

	Minimum	Maximum
X:	-64.99	64.99
Y:	-21.40	-21.40
Z:	132.05	157.50



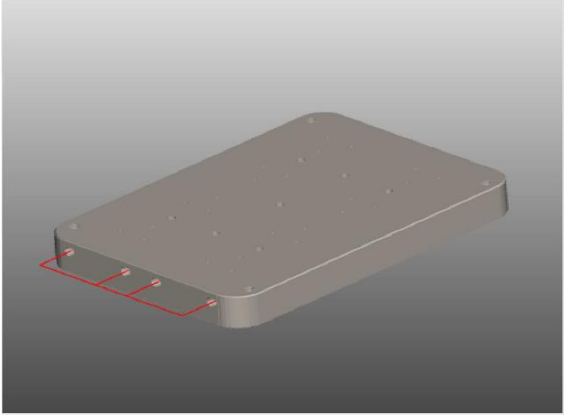
Operation	Drill6
Cutting Speed Vc	225
Feed per tooth (mm)	N.A.
Z Feed Rate	84.36
Tool Station no.	30
Tool Description	6.0mm JOBBER DRILL
Mach Depth	135.00

	Minimum	Maximum
X:	-64.99	64.99
Y:	-21.40	-21.40
Z:	-0.25	157.50



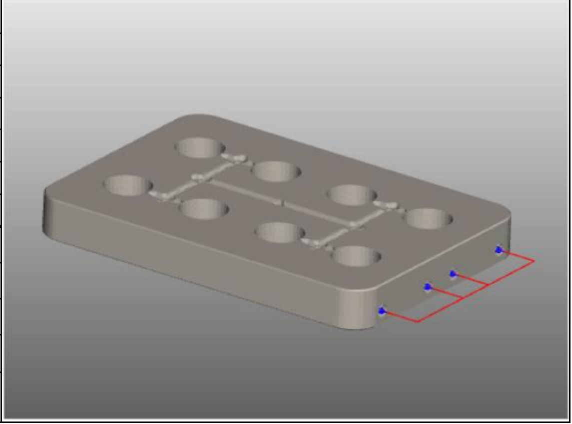
Operation	Drill7
Cutting Speed Vc	250
Feed per tooth (mm)	N.A.
Z Feed Rate	73.60
Tool Station no.	33
Tool Description	9.0mm JOBBER DRILL
Mach Depth	12.70

	Minimum	Maximum
X:	-64.99	64.99
Y:	-21.40	-21.40
Z:	117.30	157.50



Operation	Thread Mill2
Cutting Speed Vc	225
Feed per tooth (mm)	0.2
Z Feed Rate	2302.30
Tool Station no.	27
Tool Description	M8-M9 CRB SP THREAD MILL
Mach Depth	10.00

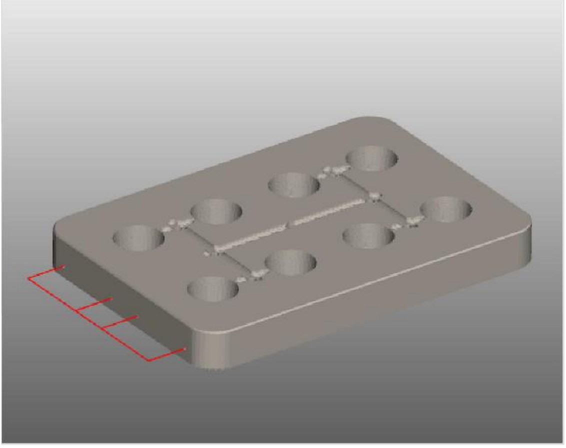
	Minimum	Maximum
X:	-66.79	66.79
Y:	-23.19	-19.61
Z:	119.84	157.50



Setup No.	Setup Name	Setup Origin
4	Group4	90.00, 21.23, 40.00

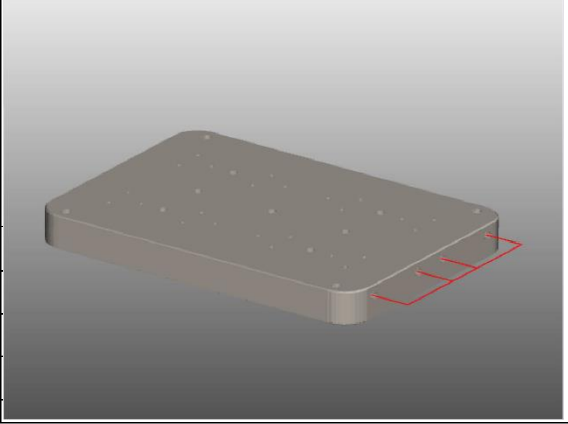
Operation	Center Drill8
Cutting Speed Vc	180
Feed per tooth (mm)	N.A.
Z Feed Rate	61.61
Tool Station no.	6
Tool Description	6MM X 60DEG HSS CENTERDRILL
Mach Depth	2.70

	Minimum	Maximum
X:	-64.99	64.99
Y:	-21.40	-21.40
Z:	128.05	157.50



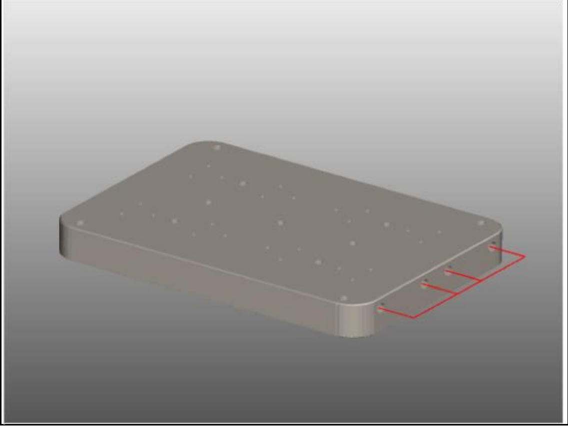
Operation	Drill8
Cutting Speed Vc	225
Feed per tooth (mm)	N.A.
Z Feed Rate	84.36
Tool Station no.	30
Tool Description	6.0mm JOBBER DRILL
Mach Depth	136.80

	Minimum	Maximum
X:	-64.99	64.99
Y:	-21.40	-21.40
Z:	117.30	157.50



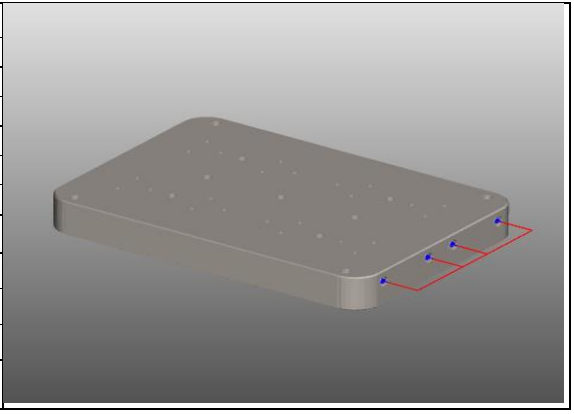
Operation	Drill9
Cutting Speed Vc	250
Feed per tooth (mm)	N.A.
Z Feed Rate	73.60
Tool Station no.	33
Tool Description	9.0mm JOBBER DRILL
Mach Depth	12.70

	Minimum	Maximum
X:	-64.99	64.99
Y:	-21.40	-21.40
Z:	-6.05	157.50



Operation	Thread Mill3
Cutting Speed Vc	225
Feed per tooth (mm)	0.2
Z Feed Rate	2302.30
Tool Station no.	27
Tool Description	M8-M9 CRB SP THREAD MILL
Mach Depth	10.00

	Minimum	Maximum
X:	-66.79	66.79
Y:	-23.19	-19.61
Z:	119.88	157.50



0.36 Conclusion

Simulations enabled us to learn from our mistakes, prevent errors and optimize responses in critical situations, therefore it played a great role for us to come up with the best possible results for our design. It also had a huge impact on the work time and resources needed while keeping them to the minimum.

CONCLUSION

This project has allowed us to acquire new knowledge, to test already acquired notions, but also to know how to think and react to practical problems.

Through this project, we wanted to look for solutions that were as broad as possible to solve our problem. However, we were guided by the concern to respect the specifications as well as the technological constraints of the system to be designed.

This study gave us the opportunity to make the technical drawings of the mold elements, to get familiar with the computer-aided manufacturing using the CNC machine, and offered us the chance to get acquainted with advanced computer tools (SolidWorks, CAMWorks, SolidWorks plastics and ABAQUS...) and to learn a lot of information about plastics and the methods of working with these materials.

The creation of this work allowed us to expand the knowledge and the 'know-hows' we gained during our years of study at University M'Hamed Bougara of Boumerdes while also preparing for our integration into professional life and positioning ourselves on the industry market.

At the end of this work, we believe that the realisation of the plastic injection mould of a wheelbarrow's wheel spacer will meet the objectives set by SOFICLEF and will be a positive contribution, especially on the economic side, and will contribute to its development.

Bibliography

[1] Material selection guide. [Online]. Available: https://www.curbellplastics.com/Research-Solutions/Technical-Resources/Technical-Resources/Plastic-Material-Selection-Guide

[2] D. V. Rosato, *Plastics processing data handbook*. Springer Science & Business Media, 2012.

[3] Extrusion blow molding. [Online]. Available: https://www.silganplastics.com/capabilities-technologies/extrusion

[4] Injection blow molding process. [Online]. Available: http://ns-plastic.blogspot.com/2011/01/types-of-blow-molding-process.html

[5] Injection stretch blow moulding. [Online]. Available: https://robinsonpackaging.com/technical-capabilities/injection-stretch-blow-moulding/

[6] Plastics extrusion. [Online]. Available: https://suthaimachine.com/2016/10/25/nulla-metus-metus-ullamcorper-vel-tincidunt/

[7] Compression molding png images. [Online]. Available: https://www.pngwing.com/en/search?q=Compression+molding

[8] D. D. Kopeliovich. Transfer molding of polymers. [Online]. Available: http://www.substech.com/dokuwiki/doku.php?id=transfer_molding_of_polymers

[9] Rotation molding process. [Online]. Available: https://www.bpf.co.uk/plastipedia/processes/Rotational_Moulding.aspx

[10] A. BARRETT. Thermoforming with biobased plastics for greater sustainability. [Online]. Available: https://bioplasticsnews.com/2019/05/04/thermoforming-with-biobased-plastics-for-greater-sustainability/

[11] Main process : Calendering machine. [Online]. Available: http://www.thainam.com/cam.html

[12] The coating process. [Online]. Available: https://www.klieverik.com/laminating-coating-calender/coating-process/

[13] M. A. A. Khan, A. K. Sheikh, and B. S. Al-Shaer, "Evolution of metal casting technologies—a historical perspective," in *Evolution of Metal Casting Technologies.* Springer, 2017, pp. 1–43.

[14] R. J. Crawford and P. Martin, *Plastics engineering.* Butterworth-Heinemann, 2020.

[15] Soficlef. [Online]. Available: https://www.soficlef.com

[16] A. Shrivastava, *Introduction to plastics engineering.* William Andrew, 2018.

[17] R. Klein, *Laser welding of plastics: materials, processes and industrial applications.* John Wiley & Sons, 2012.

[18] S. A. Benidir Smail, "etude et conception d'un moule d'injection plastique pour port-cable d'un refrigérateur eniem," 2013-2014.

[19] K. Alemayehu, "Design, simulation and analysis for injection moulding of manual juice maker," 2014.

[20] Plastic processing technique. [Online]. Available: https://www.scribd.com/document/394610436/Plastic-Processing-Technique

[21] S. S. Teklehaimanot, "Simulation and design of a plastic injection mold," 2012.

[22] M. E.-P. Corporation. Molding. [Online]. Available: https://www.m-ep.co.jp/en/pdf/product/iupi_nova/molding.pdf

[23] L. A. E. Group. Engineering plastics part and mold design. [Online]. Available: LANXESS

[24] D. M. Bryce, *Plastic injection molding: Mold design and construction fundamentals.* Society of Manufacturing Engineers, 1998.

[25] Camworks. [Online]. Available: https://www.solidworks.com/partner-product/camworks

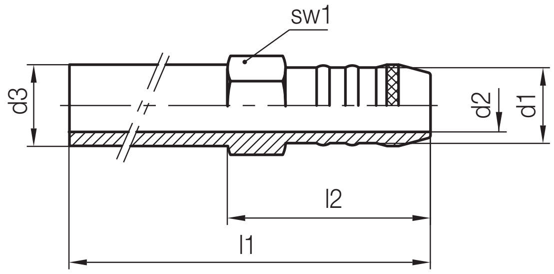
[26] P. Lappalainen, "Computer aided manufacturing with solidworks cam and mastercam," 2020.

Appendices

Z91/...

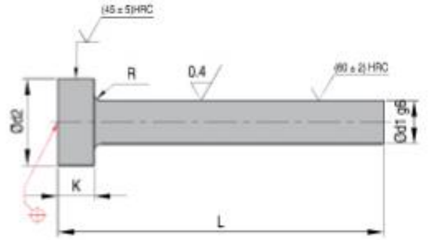
Verlängerungstülle
Extension tube
Raccord mouliste long

Mat.: 2.0401



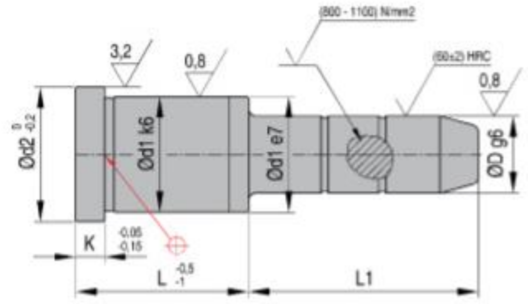
◀ Z85/...
Z851/...
Z852 /...
Z8525 /...
Z853/...
Z854/...

sw1	l2	d3	d2	**d1**	**l1**	**Nr./No.**
7	18,5	5	3	**5**	**100**	Z91/ 9x100
11	29,5	10	6	**9**	**120**	9x120
					240	9x240
					360	9x360
15	35	14	9	**13**	**150**	13x150
					300	13x300
					450	13x450
22	40	21	13	**19**	**500**	19x500



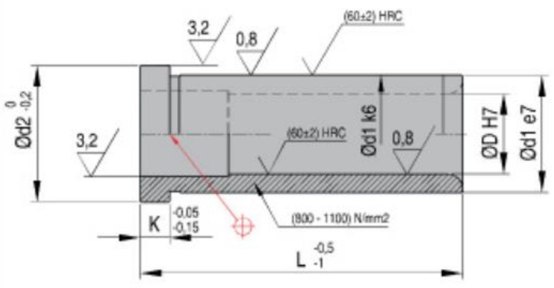
Info
Standard: DIN1530/ISO6751
Mat.: WS (~1.2067)
Max. Temp: 250°C
Surface roughness: Ra

d1 g6	d2 $^{0}_{-0,2}$	K $^{0}_{-0,05}$	R $^{+0,2}_{0}$	L $^{+2}_{0}$													
				0040	0063	0080	0100	0125	0160	0200	0250	0315	0400	0500	0630	0800	1000
01-50	3,0	1,5	0,2	●	●	●	●	●	●	●	●						
01-60	3,0	1,5	0,2	●	●	●	●	●	●	●							
01-70	3,0	1,5	0,2				●	●	●	●							
01-80	3,0	1,5	0,2				●	●	●	●							
02-00	4,0	2,0	0,2	●	●	●	●	●	●	●	●	●					
02-10	4,0	2,0	0,2						●								
02-20	4,0	2,0	0,2			●	●	●	●	●	●	●					
02-50	5,0	2,0	0,3	●	●	●	●	●	●	●	●	●					
02-70	5,0	2,0	0,3			●	●	●	●	●	●	●					
03-00	6,0	3,0	0,3	●	●	●	●	●	●	●	●	●	●	●			
03-10	6,0	3,0	0,3						●								
03-20	6,0	3,0	0,3			●	●	●	●	●	●	●	●				
03-50	7,0	3,0	0,3	●		●	●	●	●	●	●	●	●				
03-70	7,0	3,0	0,3			●	●	●	●	●	●	●	●				
04-00	8,0	3,0	0,3	●	●	●	●	●	●	●	●	●	●	●			
04-10	8,0	3,0	0,3						●	●							
04-20	8,0	3,0	0,3		●	●	●	●	●	●	●	●	●				
04-50	8,0	3,0	0,3			●	●	●	●	●	●	●	●				
04-70	8,0	3,0	0,3			●	●	●	●	●	●	●	●				
05-00	10,0	3,0	0,3	●	●	●	●	●	●	●	●	●	●	●	●	●	
05-10	10,0	3,0	0,3							●							
05-20	10,0	3,0	0,3			●	●	●	●	●	●	●	●	●			
05-50	10,0	3,0	0,3				●	●	●	●	●	●	●	●			
06-00	12,0	5,0	0,5	●	●	●	●	●	●	●	●	●	●	●	●	●	
06-10	12,0	5,0	0,5							●							
06-20	12,0	5,0	0,5			●	●	●	●	●	●	●	●	●	●		
06-50	12,0	5,0	0,5				●	●	●	●	●	●	●	●			
07-00	12,0	5,0	0,5				●	●	●	●	●	●	●	●	●		
08-00	14,0	5,0	0,5	●	●	●	●	●	●	●	●	●	●	●	●	●	●
08-10	14,0	5,0	0,5							●							
08-20	14,0	5,0	0,5		●	●	●	●	●	●	●	●	●	●	●	●	
08-50	14,0	5,0	0,5			●	●	●	●	●	●	●	●	●	●		
09-00	14,0	5,0	0,5				●	●	●	●	●	●	●	●	●		
10-00	16,0	5,0	0,5	●		●	●	●	●	●	●	●	●	●	●	●	●
10-10	16,0	5,0	0,5							●							
10-20	16,0	5,0	0,5			●	●	●	●	●	●	●	●	●	●		
10-50	16,0	5,0	0,5				●	●	●	●	●	●	●	●	●		
11-00	16,0	5,0	0,5				●	●	●	●	●	●	●	●	●		
12-00	18,0	7,0	0,8	●		●	●	●	●	●	●	●	●	●	●	●	●
12-10	18,0	7,0	0,8							●							
12-20	18,0	7,0	0,8				●	●	●	●	●	●	●	●			
12-50	18,0	7,0	0,8				●	●	●	●	●	●	●	●	●		
14-00	22,0	7,0	0,8			●	●	●	●	●	●	●	●	●	●	●	
16-00	22,0	7,0	0,8			●	●	●	●	●	●	●	●	●	●	●	●
18-00	24,0	7,0	0,8				●	●	●	●	●	●	●	●	●		
20-00	26,0	8,0	1,0				●	●	●	●	●	●	●	●	●	●	●



Mat.: 1.7131 60 HRC

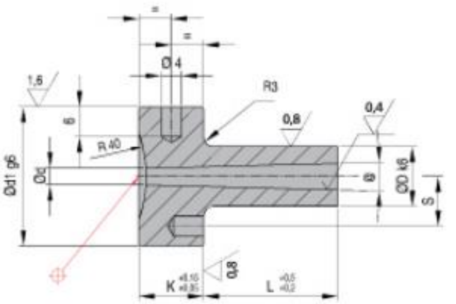
L	D		L1																		d1	d2	K
			020	035	040	045	050	055	060	065	070	075	080	085	095	105	115	125	135	165			
022	18	20	●	●	●	●	●	●	●	●	●		●	●			●				26	31	6
027			●	●	●	●	●	●	●	●	●		●	●		●		●					
036			●	●	●	●	●	●	●	●	●	●	●		●		●		●				
046			●			●				●				●		●			●	●			
056			●	●				●				●			●								
066								●				●			●								
076								●				●			●								
086								●				●			●								
096								●							●								
116																	●						



Mat.: 1.7131

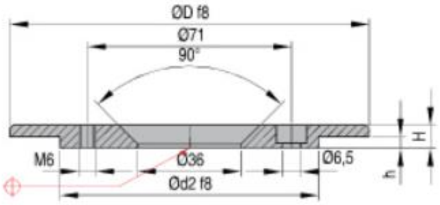
L																D		d1	d2	K
012	017	022	027	036	046	056	066	076	086	096	116	136	156	196	246					
●	●	●	●	●	●	●	●									09	10	14	16	03
●	●	●	●	●	●	●	●	●	●	●						14	15	20	25	06
	●	●	●	●	●	●	●	●	●	●	●					18	20	26	31	06
	●	●	●	●	●	●	●	●	●	●	●	●				22	24	30	35	06
			●	●	●	●	●	●	●	●	●	●	●			30	32	42	47	06
					●	●	●	●	●	●	●	●	●		●	40	42	54	60	10
								●		●	●	●	●	●		50		66	72	10
										●	●	●		●	●	60		80	86	20

Mat.: 1.2826: ca 54 HRC

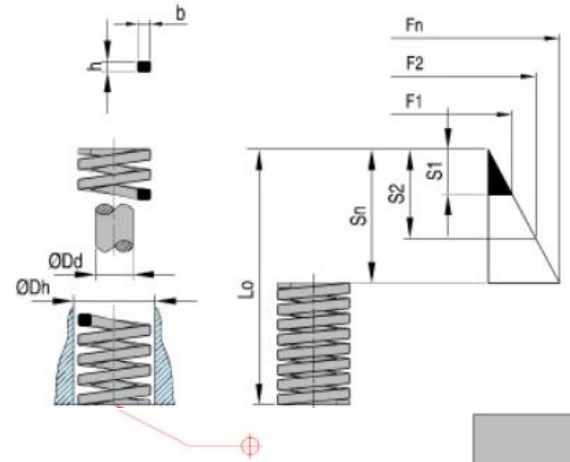


REF	D	L								d	d1	K	S	@
		022	027	036	046	056	076	096	116					
R 078	12		•	•	•					2,5	28	13	11	1°
			•	•	•	•				3,5				
	18				•	•	•	•		3	38	18	15	
					•	•	•	•		4				
DHR 78	12	•	•	•	•					2,5	28	13	11	2°
		•	•	•	•	•				3,5				
	18		•	•	•	•	•	•	•	3	38	18	15	
			•	•	•	•	•	•	•	4				
DHR 79	18		•	•	•	•				3,5	38	18	15	3°
			•	•	•	•	•			4,5				
	24				•	•	•			4,5	48	23	20	
						•	•			6,5				

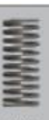
Mat.: 1.1730



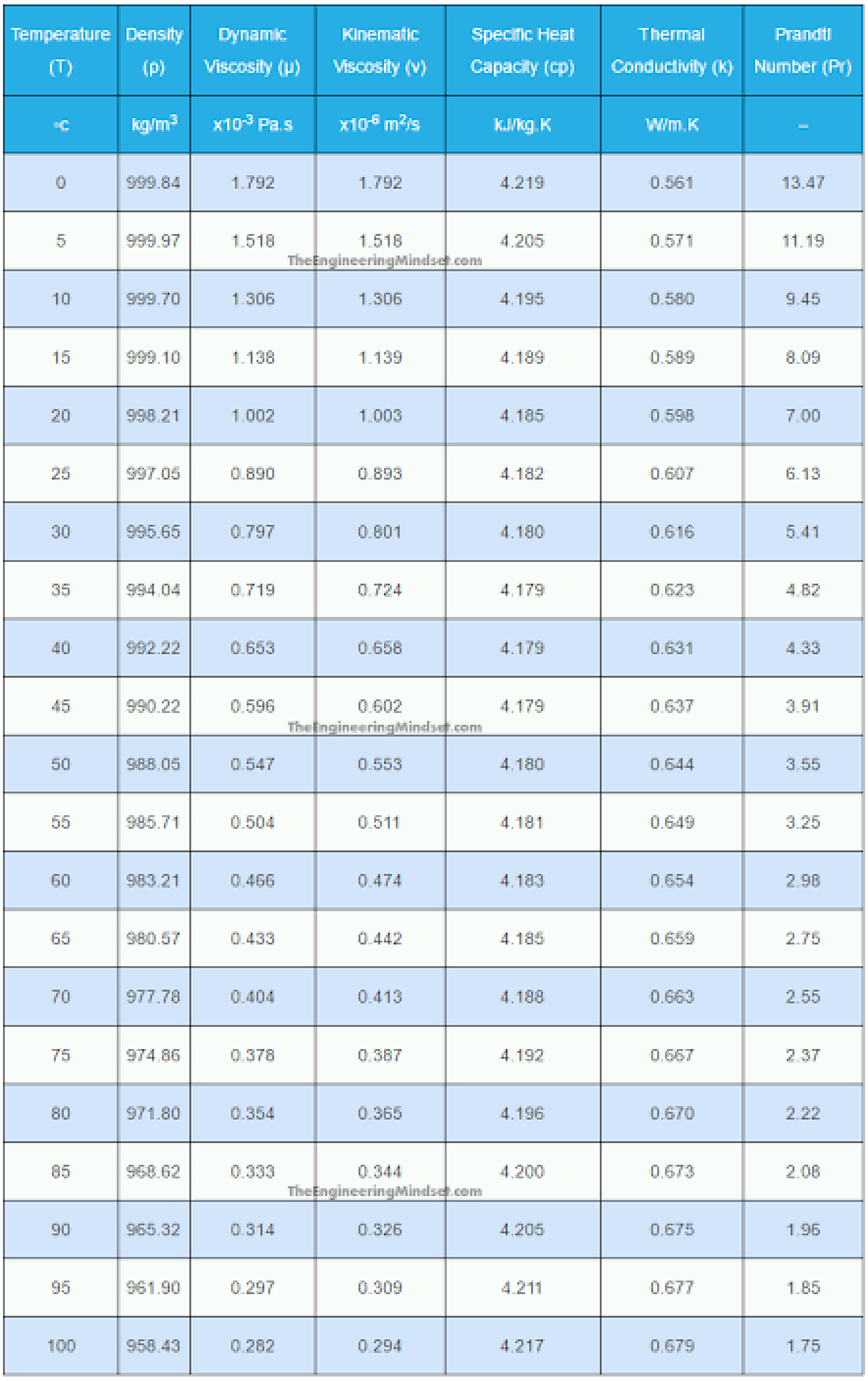
REF	D	H	h	d2
DHR2112008	120	8	4	90



Mat.: Special alloy - Load range: heavy duty.
Important: sufficient initial compression is essential for maximum spring life.
1N = 0.102 Kg (force)
Min. Temp.: -30 °C
Max. Temp.: 120 °C

						Spring rate	 30%		 40%		 50%		 D	
REF	Dh	Lo	Dd	b	h	RO	S1		S2		Sn			
	ø (mm)	mm	ø (mm)	ø (mm)	ø (mm)	N/mm	S1 (mm)	F1 (N)	S2 (mm)	F2 (N)	Sn (mm)	Fn (N)	mm	N
WZ8031E25127RO	25,0	127	12,5	5,5	4,2	57,7	25,4	1466	31,8	1835	38,1	2198	46,2	2666
WZ8031E25139RO	25,0	139	12,5	5,5	4,2	52,7	28,0	1476	35,0	1845	42,0	2213	49,3	2598
WZ8031E25152RO	25,0	152	12,5	5,5	4,2	47,8	30,4	1453	38,0	1816	45,6	2180	55,7	2662
WZ8031E25178RO	25,0	178	12,5	5,5	4,2	41,0	35,6	1460	44,5	1825	53,4	2189	65,1	2669
WZ8031E25203RO	25,0	203	12,5	5,5	4,2	35,8	40,6	1453	50,8	1819	60,9	2180	74,5	2667
WZ8031E25305RO	25,0	305	12,5	5,5	4,2	22,9	61,0	1397	76,3	1747	91,5	2095	110,2	2524
WZ8031E32038RO	32,0	38	16,0	7,1	5,4	388,0	7,6	2949	9,5	3686	11,4	4423	12,5	4850
WZ8031E32044RO	32,0	44	16,0	7,1	5,4	324,0	8,8	2851	11,0	3564	13,2	4277	14,9	4828
WZ8031E32051RO	32,0	51	16,0	7,1	5,4	272,0	10,2	2774	12,8	3482	15,3	4162	17,8	4842
WZ8031E32064RO	32,0	64	16,0	7,1	5,4	212,0	12,8	2714	16,0	3392	19,2	4070	22,4	4749
WZ8031E32076RO	32,0	76	16,0	7,1	5,4	172,0	15,2	2614	19,0	3268	22,8	3922	26,1	4489
WZ8031E32089RO	32,0	89	16,0	7,1	5,4	141,0	17,8	2510	22,3	3144	26,7	3765	30,8	4343
WZ8031E32102RO	32,0	102	16,0	7,1	5,4	122,0	20,4	2489	25,5	3111	30,6	3733	36,8	4490
WZ8031E32115RO	32,0	115	16,0	7,1	5,4	107,0	23,0	2461	28,8	3082	34,5	3692	41,4	4430
WZ8031E32127RO	32,0	127	16,0	7,1	5,4	93,0	25,4	2362	31,8	2957	38,1	3543	44,4	4129
WZ8031E32139RO	32,0	139	16,0	7,1	5,4	86,0	28,0	2408	35,0	3010	42,0	3612	48,5	4171
WZ8031E32152RO	32,0	152	16,0	7,1	5,4	78,0	30,4	2371	38,0	2964	45,6	3557	54,8	4274
WZ8031E32178RO	32,0	178	16,0	7,1	5,4	67,2	35,6	2392	44,5	2990	53,4	3588	63,6	4274
WZ8031E32203RO	32,0	203	16,0	7,1	5,4	59,1	40,6	2399	50,8	3002	60,9	3599	72,5	4285
WZ8031E32254RO	32,0	254	16,0	7,1	5,4	46,4	50,8	2357	63,5	2946	76,2	3536	92,8	4306
WZ8031E32305RO	32,0	305	16,0	7,1	5,4	38,0	61,0	2318	76,3	2899	91,5	3477	111,8	4248
WZ8031E40051RO	40,0	51	20,0	8,4	6,2	350,0	10,2	3570	12,8	4480	15,3	5355	17,0	5950

MATERIALS PROPERTIES					
Part	Material	Density Kg/m^3	Young’s modulus GPa	Poisson’s ratio	Elastic limit MPa
Ejector retainer plate	C22	7700	190	0.29	500-700
Ejector plate					
Spacer Plate	C30	7700	200	0.3	600-780
Bottom Clam plate					
Top Clamp plate					
Screws	C35	7800	210	0.3	600-780
Core holder plate	25CrMo4	7800	210	0.3	700-1100
Cavity holder plate					
Core part	3Cr2Mo	7800	210	0.3	850-1000
Cavity plate					
Leader Pins	12MnCr5	7800	210	0.3	1230
Return Pins	100Cr6	7800	210	0.3	520



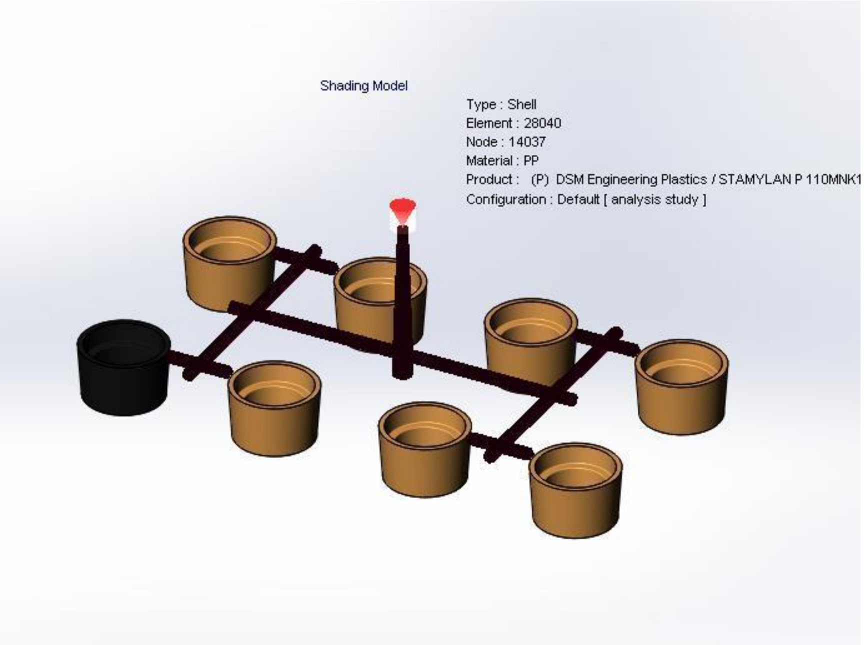
Temperature (T)	Density (ρ)	Dynamic Viscosity (μ)	Kinematic Viscosity (ν)	Specific Heat Capacity (cp)	Thermal Conductivity (k)	Prandtl Number (Pr)
°c	kg/m^3	x10^{-3} Pa.s	x10^{-6} m^2/s	kJ/kg.K	W/m.K	–
0	999.84	1.792	1.792	4.219	0.561	13.47
5	999.97	1.518	1.518	4.205	0.571	11.19
10	999.70	1.306	1.306	4.195	0.580	9.45
15	999.10	1.138	1.139	4.189	0.589	8.09
20	998.21	1.002	1.003	4.185	0.598	7.00
25	997.05	0.890	0.893	4.182	0.607	6.13
30	995.65	0.797	0.801	4.180	0.616	5.41
35	994.04	0.719	0.724	4.179	0.623	4.82
40	992.22	0.653	0.658	4.179	0.631	4.33
45	990.22	0.596	0.602	4.179	0.637	3.91
50	988.05	0.547	0.553	4.180	0.644	3.55
55	985.71	0.504	0.511	4.181	0.649	3.25
60	983.21	0.466	0.474	4.183	0.654	2.98
65	980.57	0.433	0.442	4.185	0.659	2.75
70	977.78	0.404	0.413	4.188	0.663	2.55
75	974.86	0.378	0.387	4.192	0.667	2.37
80	971.80	0.354	0.365	4.196	0.670	2.22
85	968.62	0.333	0.344	4.200	0.673	2.08
90	965.32	0.314	0.326	4.205	0.675	1.96
95	961.90	0.297	0.309	4.211	0.677	1.85
100	958.43	0.282	0.294	4.217	0.679	1.75

SOLIDWORKS

3DEXPERIENCE

Plastic Project Report

Plastic injection simulation of a Wheelbarrow’s wheel spacer product



Shading Model

Type : Shell

Element : 28040

Node : 14037

Material : PP

Product : (P) DSM Engineering Plastics / STAMYLAN P 110MNK1

Configuration : Default [analysis study]

Designer: *Marwa DAKICHE*

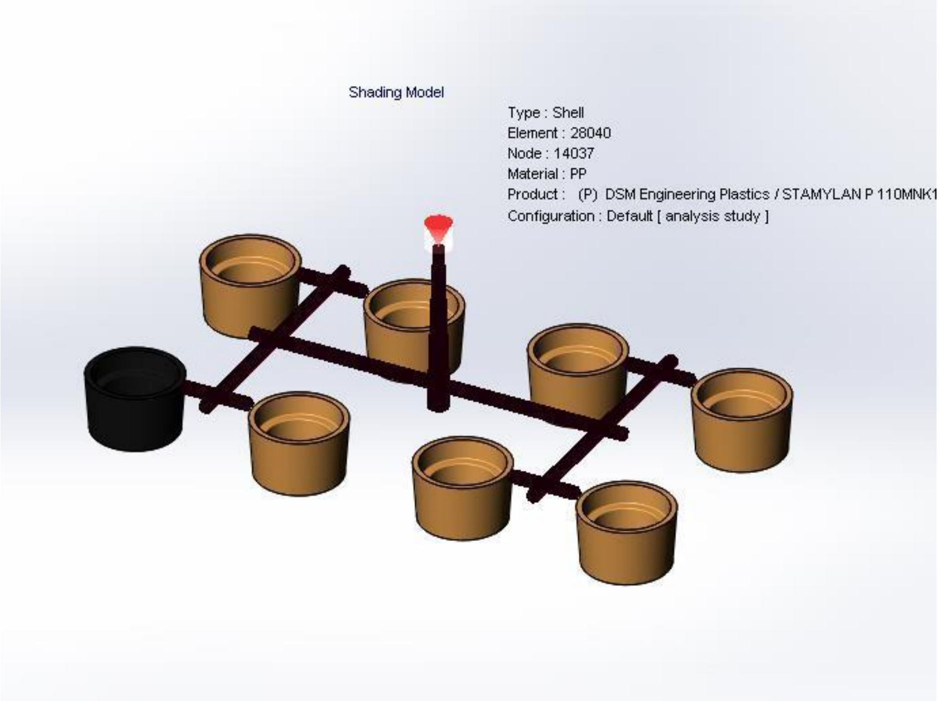
Analysis: Shell/ Solid Based Surface Model

Introduction

This report discusses an injection simulation of an eight-cavity runner system for the production of a plastic Wheelbarrow's wheel spacer.

Table of Contents

Model Information



Shading Model

Type : Shell
Element : 28040
Node : 14037
Material : PP
Product : (P) DSM Engineering Plastics / STAMYLAN P 110MNK1
Configuration : Default [analysis study]

Name	analysis study

Name	Eight-cavity runner system
Element	28040
Node	14037
Volume	54.05 (cm3)
Mass	49.46 (G)
Size	212.82 (mm) x 20.40 (mm) x 112.84 (mm)

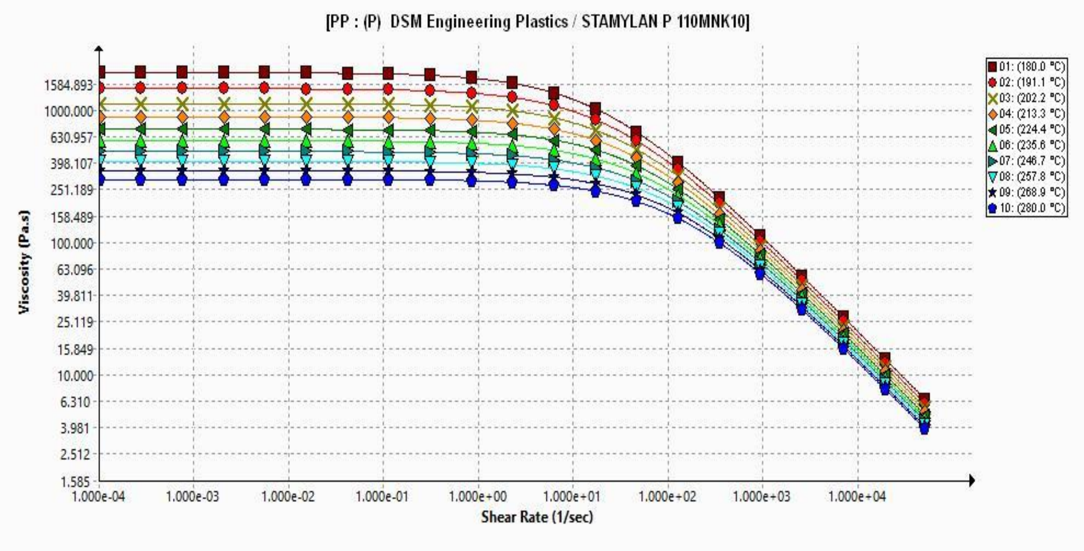
Material Properties

Polymer:

Properties:

Material Name	PP
Product Name	(P) DSM Engineering Plastics / STAMYLAN P 110MNK10
Melt Temperature	230.00 °C
Mold Temperature	40.00 °C
Ejection Temperature	110.00 °C
Transition Temperature	150.00 °C
Specific Heat	2.9400000000e+07 erg/(g-C)
Thermal Conductivity	1.7100000000e+04 erg/(sec-cm-K)
Young Modulus	2.1000000000e+10 dyne/cm2
Poisson's Ratio	3.8000000000e-01

Curve Data:



Polymer Viscosity Graph

Process Parameters

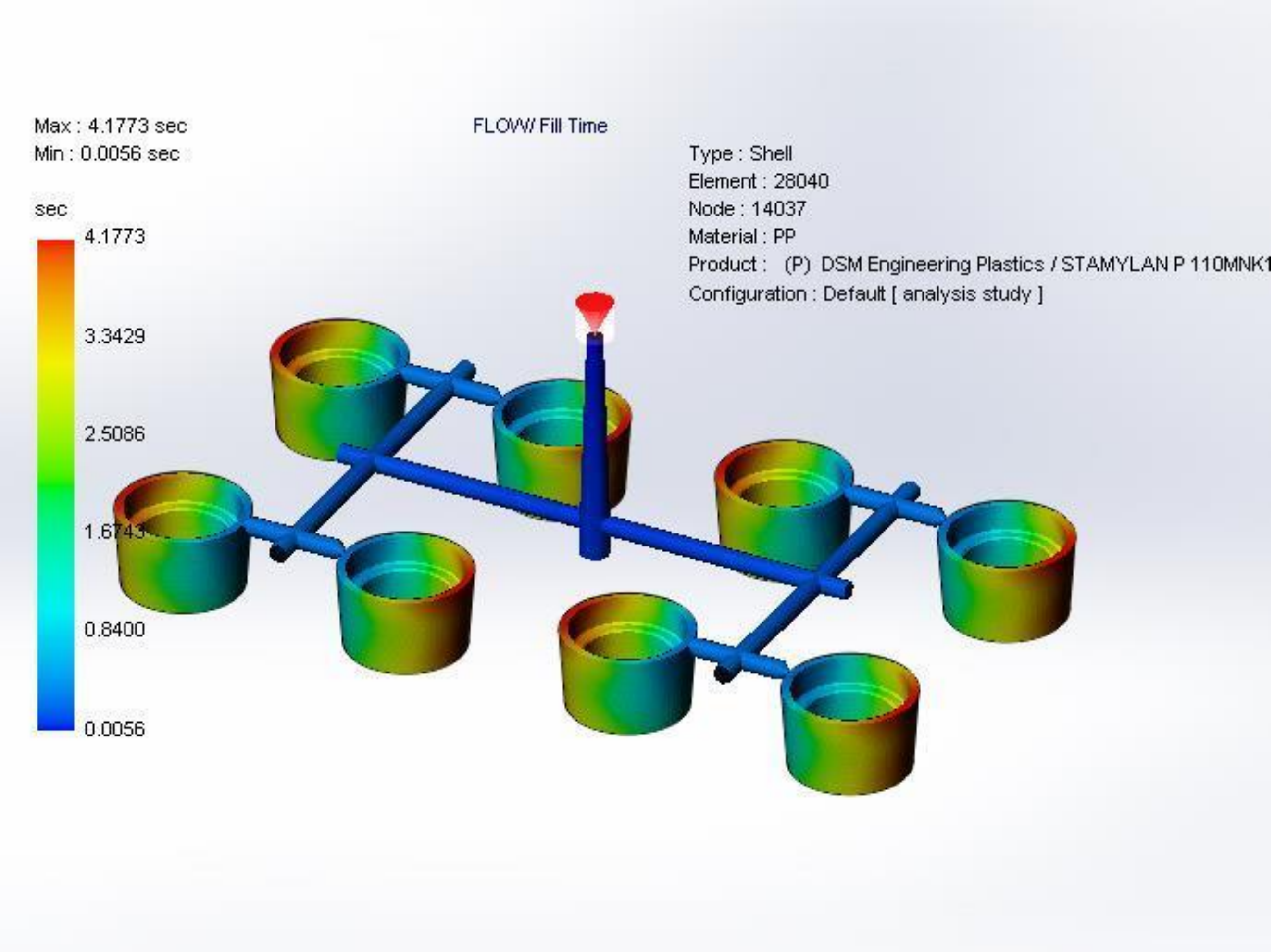
Fill Settings:

Filling Time	4.19 sec
Main Material Melt Temperature	230 °C
Mold Wall Temperature	40 °C
Injection Pressure Limit	100 MPa
Flow Rate Limit	194 cc/s
Flow/Pack Switch Point (% Filled Volume)	100 %
Pressure Holding Time	7.47 sec
Total Time in Pack Stage	41.95 sec
Cavity Initial Air Pressure	0.101 MPa
Cavity Initial Air Temperature	30 °C
Temperature Criteria for Short Shots	150 °C
Clamp Force Limit	100 Tonne

Flow Results

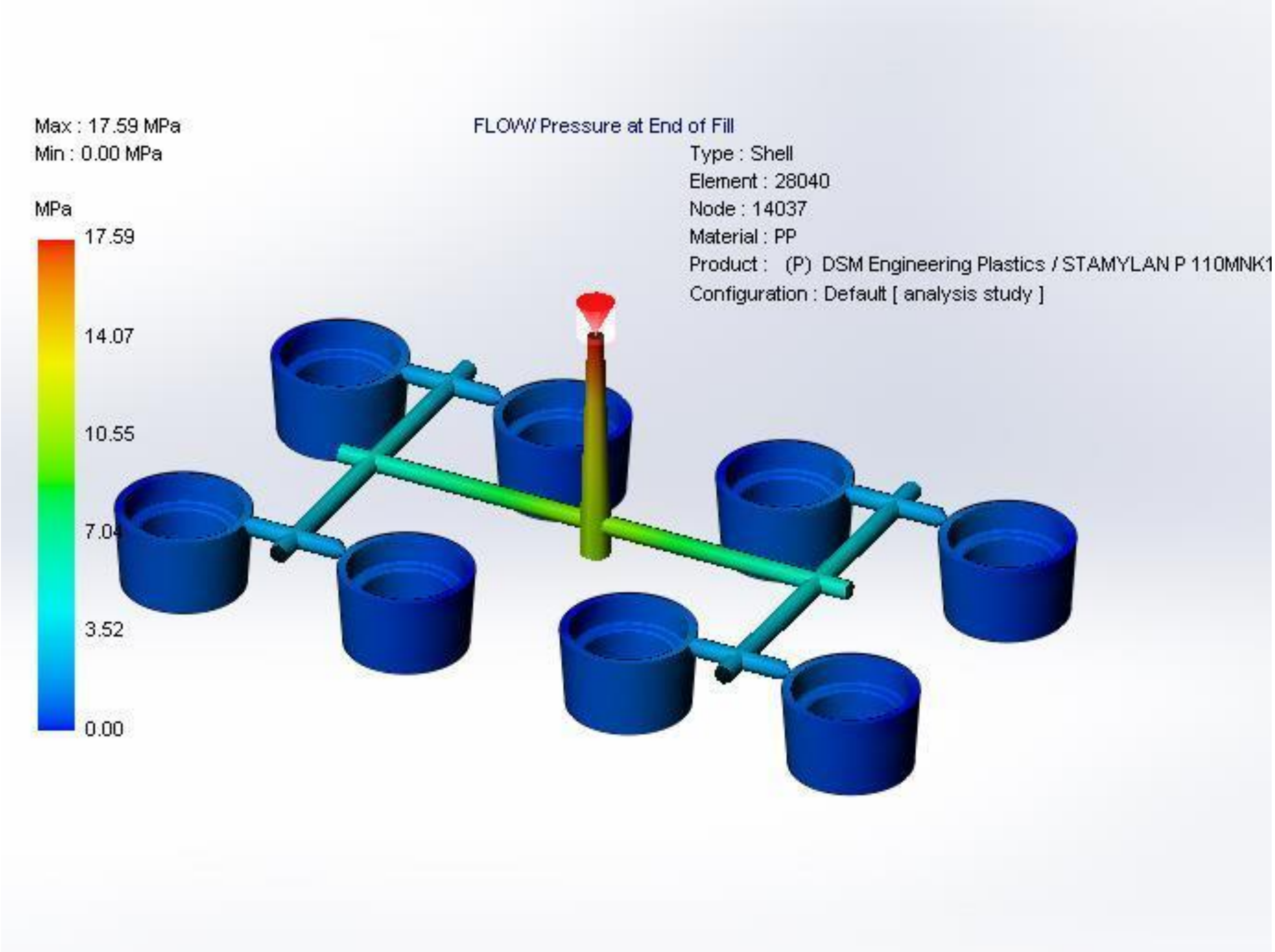
Flow Summary:

Required injection pressure	17.5900 Mpa
Max. central temperature	230.5600 °C
Max. average temperature	205.1300 °C
Max. bulk temperature	232.1400 °C
Max. shear stress	0.2100 Mpa
Max. shear rate	10886.5400 1/sec
Averaged perfect cooling time	26.1900 sec
CPU Time	694.71 sec
Cycle Time	53.23 sec
|- 1. Filling Time	4.18 sec
|- 2. Cooling Time	44.05 sec
|- 3. Mold Open Time	5.00 sec

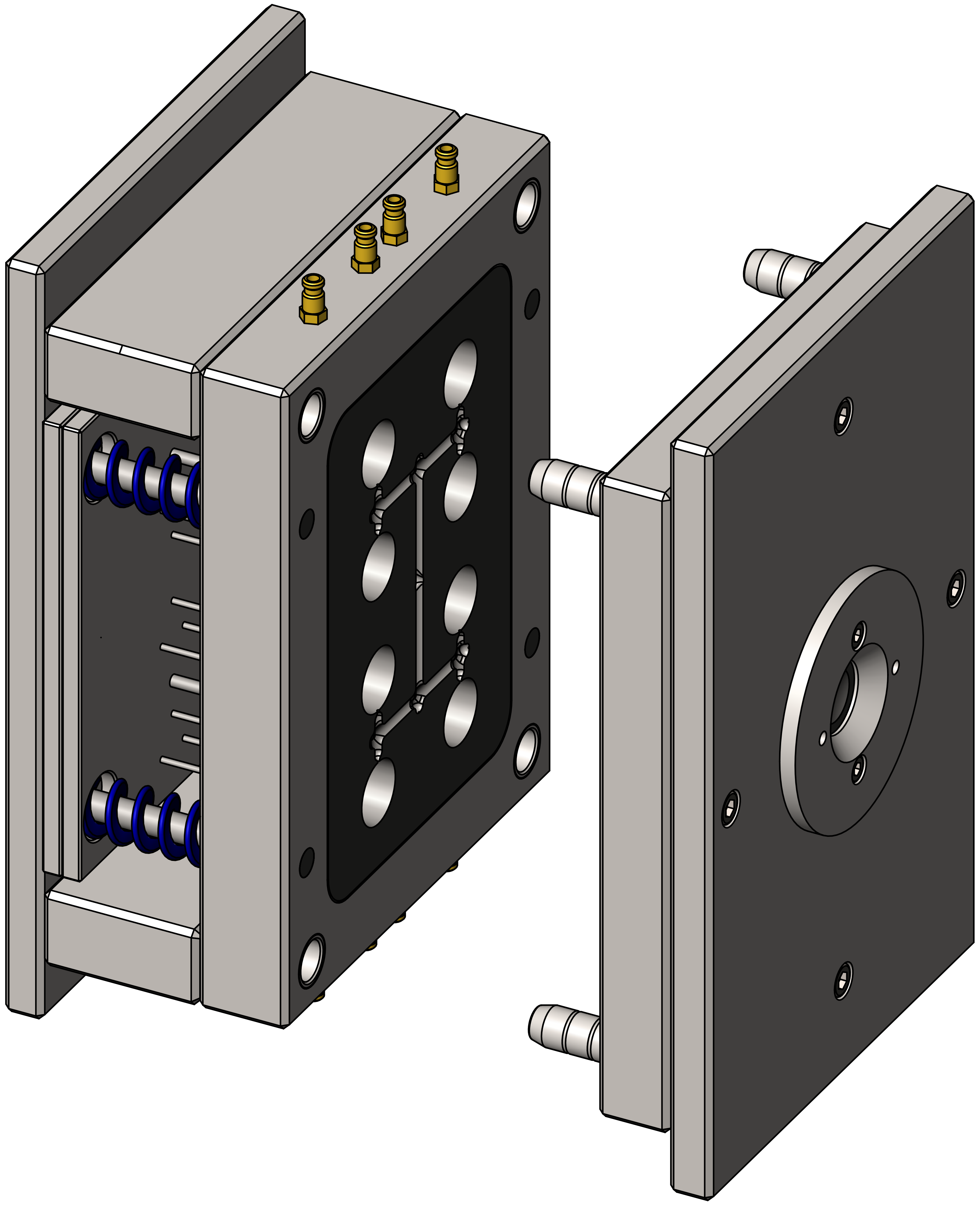


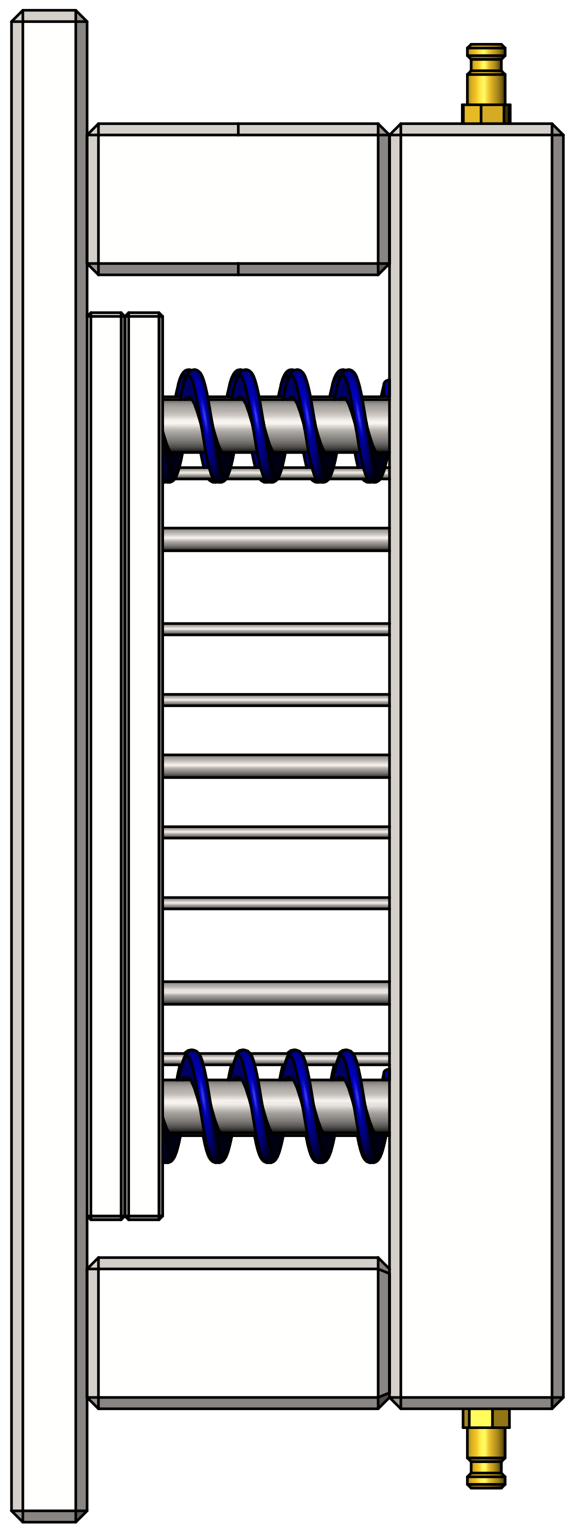
Type : Shell
Element : 28040
Node : 14037
Material : PP
Product : (P) DSM Engineering Plastics / STAMYLAN P 110MNK1
Configuration : Default [analysis study]

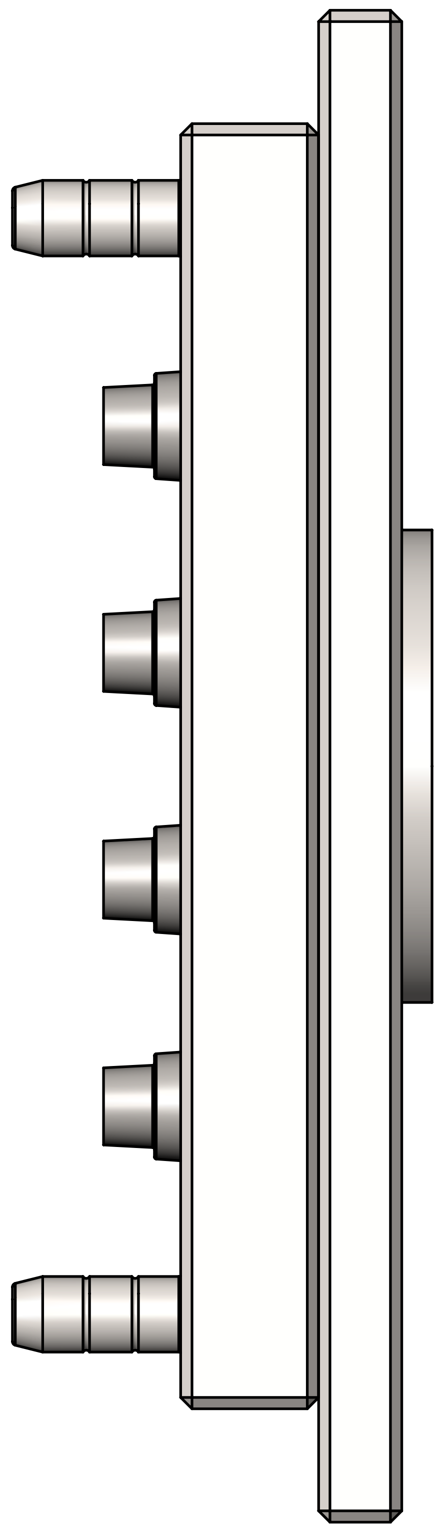
Name	**Type**	**Min**	**Max**
Fill Time	Flow Results	0.005618	4.177279



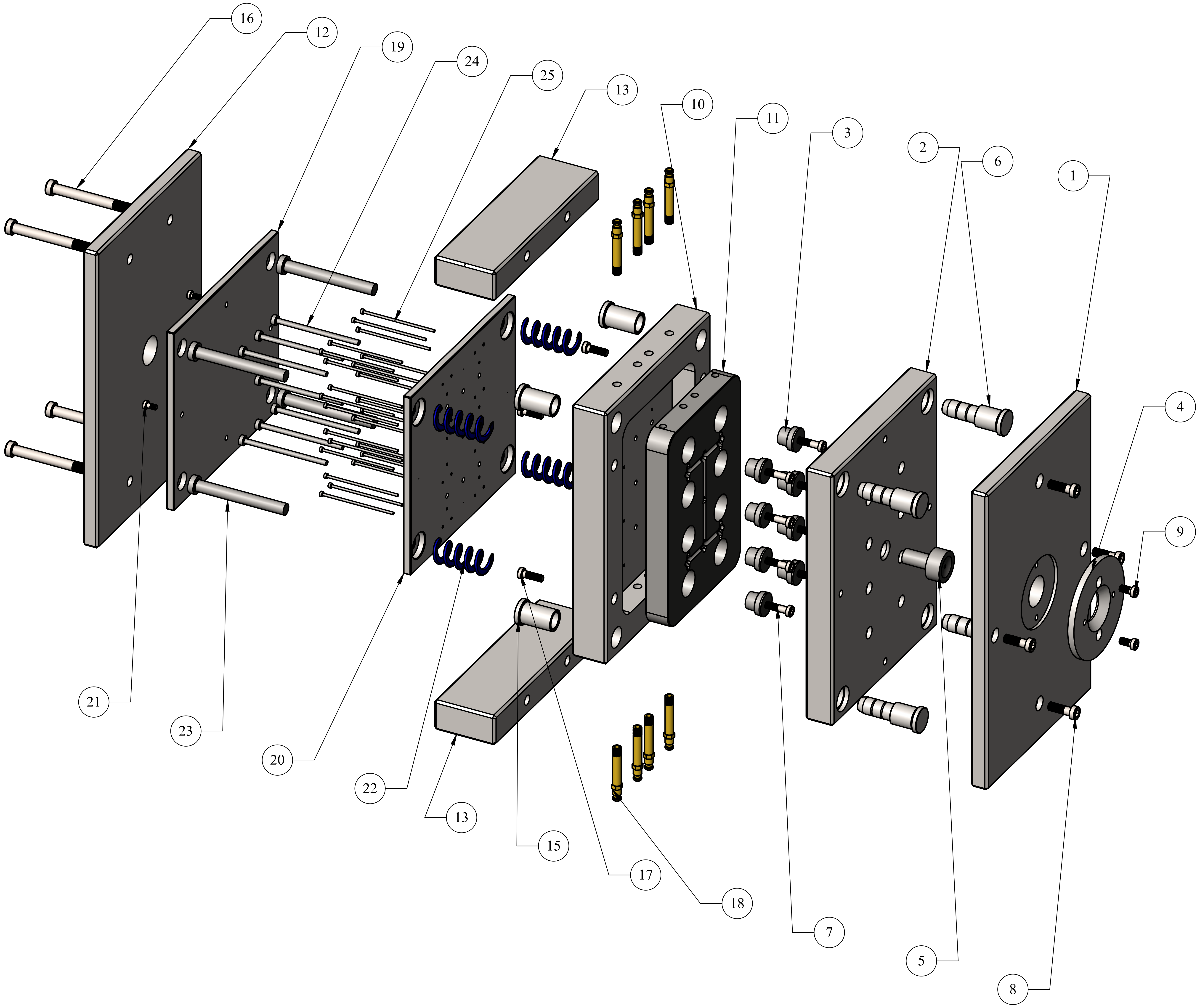
Name	Type	Min	Max
Pressure at End of Fill	Flow Results	0.000000	17.589821







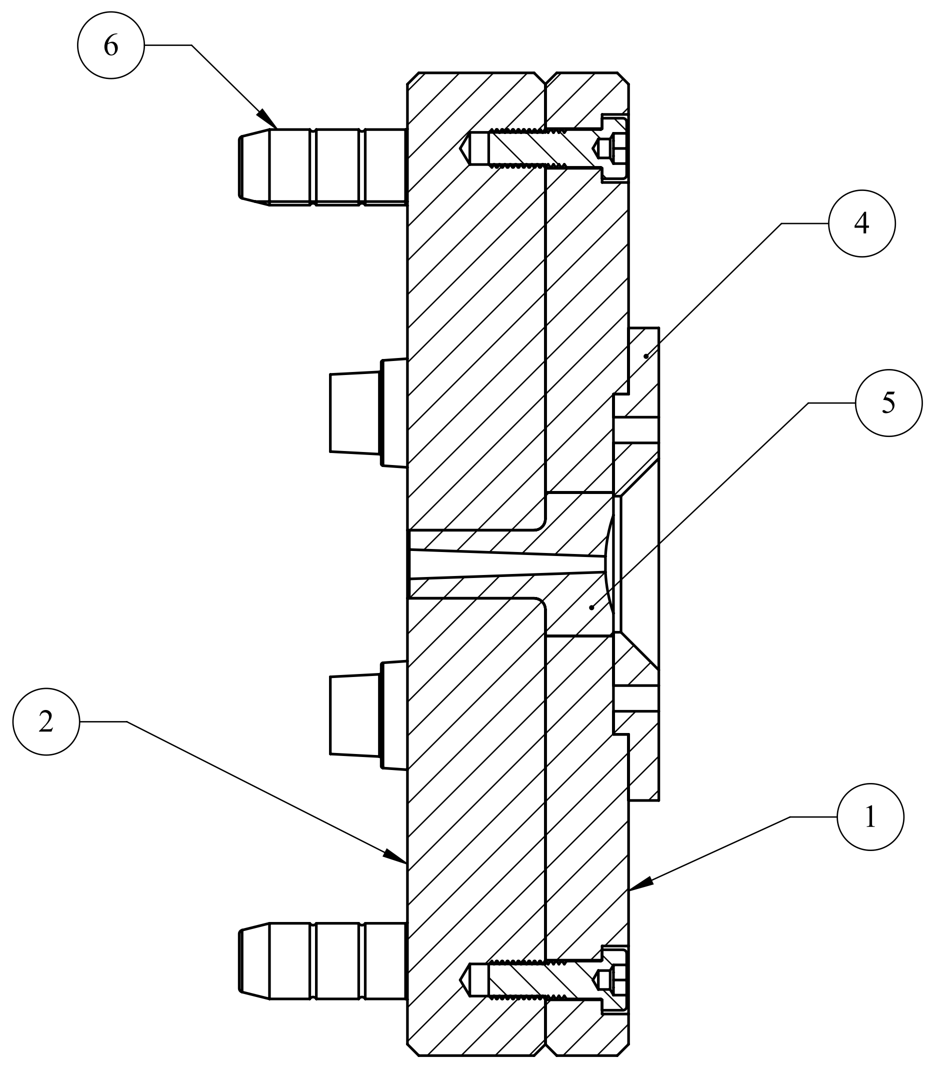
SCALE: 1:2	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A2	UMBB-FT-DGM-CMM	CLASS OF 2022



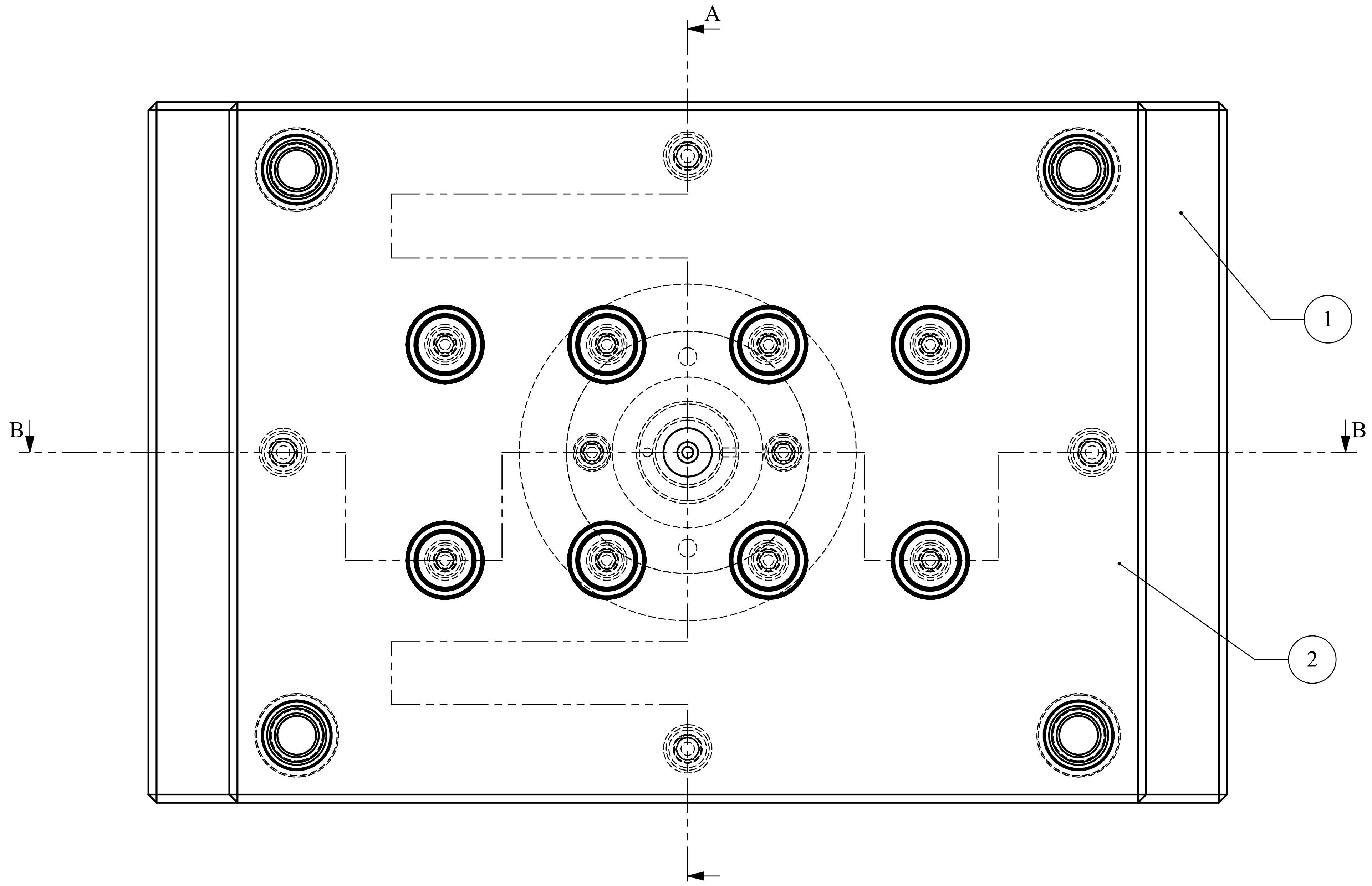
Full mold assemblycomponents

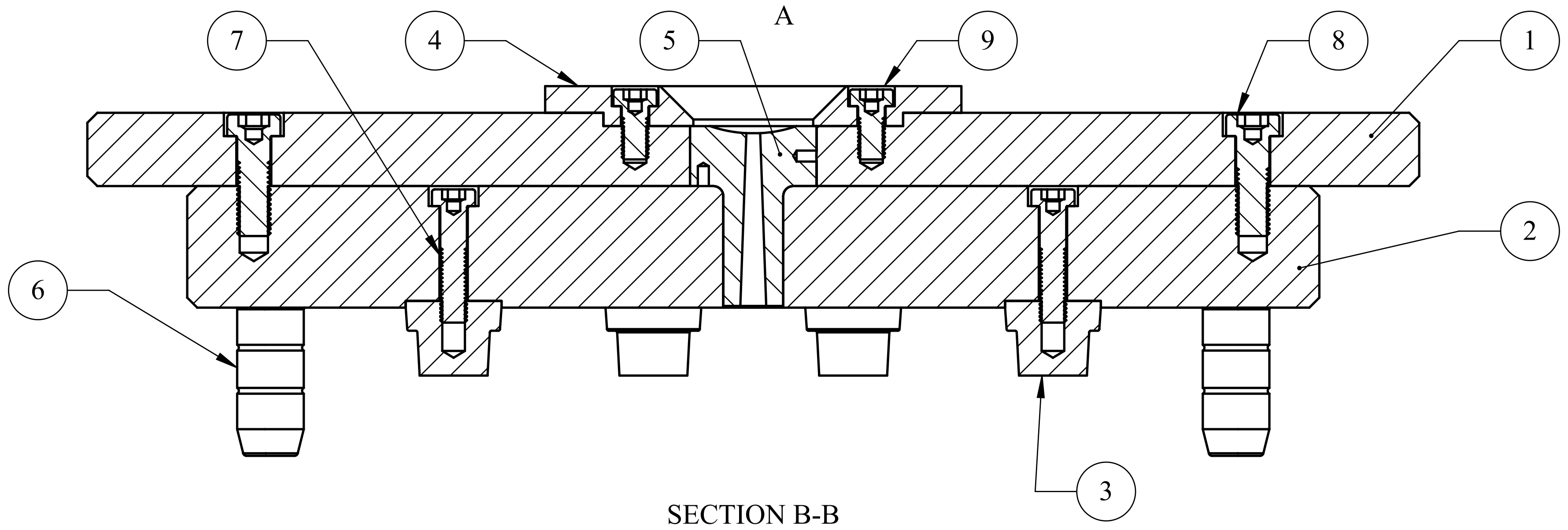
REP	NBR	TITLE	MATERIAL	OBSERVATION
25	24	Cavity ejector pin	100Cr6	
24	7	Runners ejector pin	100Cr6	
23	4	Return pin	100Cr6	
22	4	Blue spring slat round wire		See appendix
21	4	HSH screw M6x12	C35	
20	1	Ejector plate	C22	
19	1	Ejector retainer plate	C22	
18	8	Extension pipe	CuZn39Fb3	
17	4	HSH Screw M8x25	C35	
16	4	HSH screw M12x80	C35	
15	4	Leader pin bushing	16MnCr5	
14	1	Ejection system sub assembly		
13	2	Spacer plate	C30	
12	1	Bottom clamp plate	C30	
11	1	Cavity plate	3Cr2Mo	
10	1	Cavity holder block	25CrMo4	
9	2	HSH Screw M8x16	C35	
8	4	HSH screw M10x20	C35	
7	4	HSH Screw M8x40	C35	
6	4	Leader pin	16MnCr5	
5	1	Sprue bushing	60MnSiCr4	
4	1	Locating ring	C45	
3	1	Core plate	3Cr2Mo	
2	1	Core holder block	25CrMo4	
1	1	Top clamp plate	C30	

SCALE: 1:4	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA** .../07/2022
A2	UMBB-FT-DGM-CMM	CLASS OF 2022

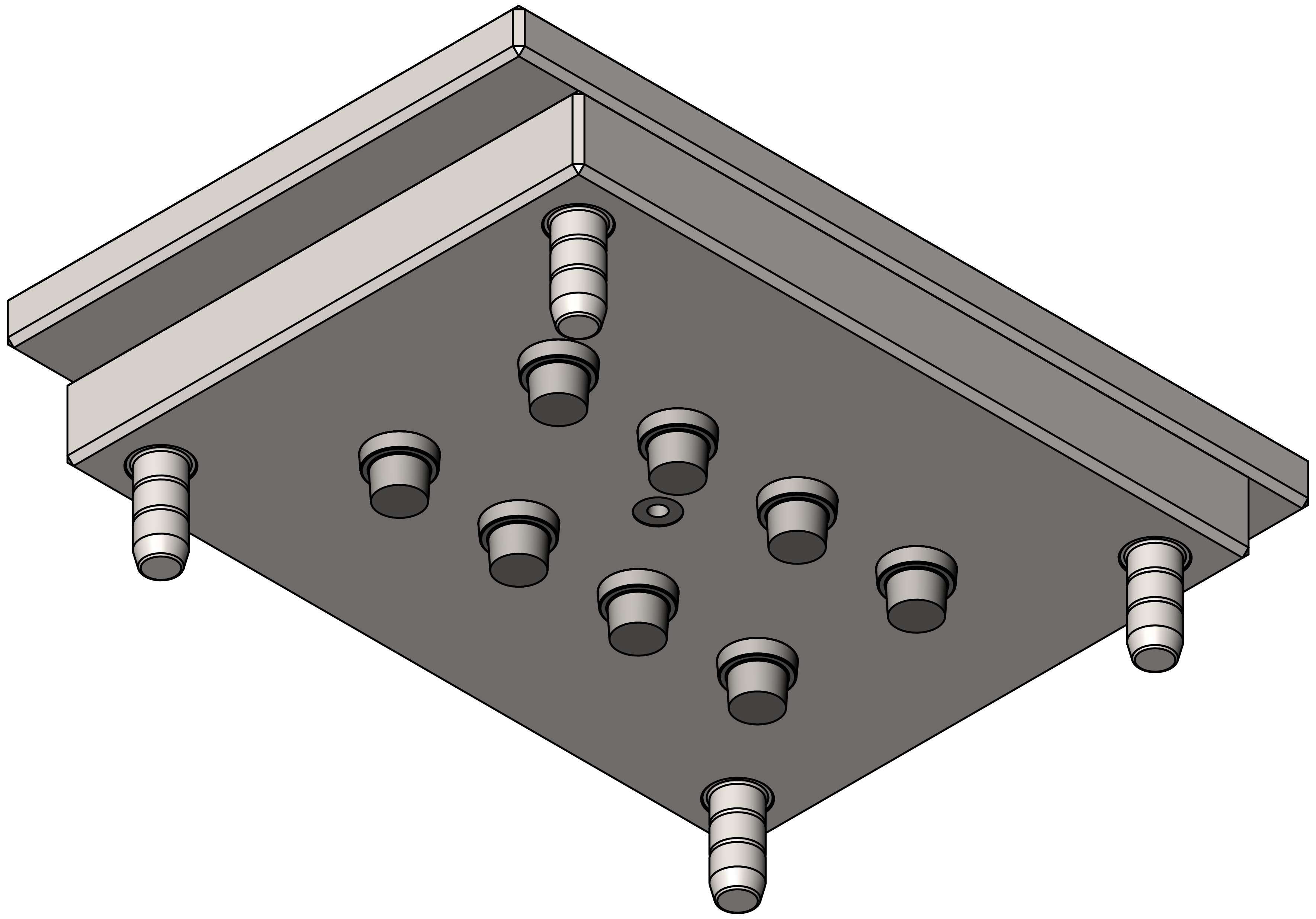


SECTION A-A





SECTION B-B



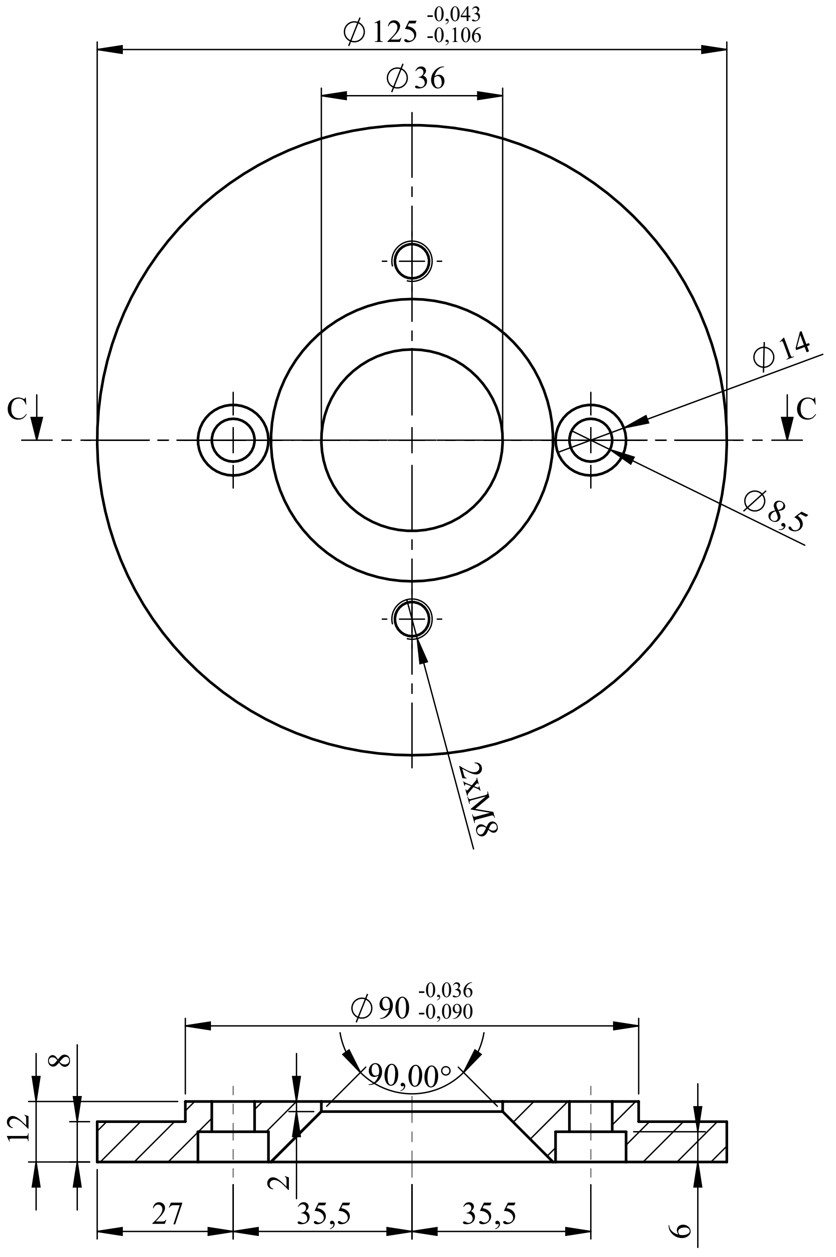
		Fixed part sub assembly		
9	2	HSH Screw M8x16	C35	
8	4	HSH screw M10x20	C35	
7	4	HSH Screw M8x40	C35	
6	4	Leader pin	16MnCr5	
5	1	Sprue bushing	60MnSiCr4	
4	1	Locating ring	C45	
3	8	Core part	3Cr2Mo	
2	1	Core holder block	25CrMo4	
1	1	Top clamp plate	C30	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 1:2

INJECTION MOLD
FOR A WHEELBARROW'S
WHEEL SPACER

PRESENTED BY:
DAKICHE MARWA
.../07/2022

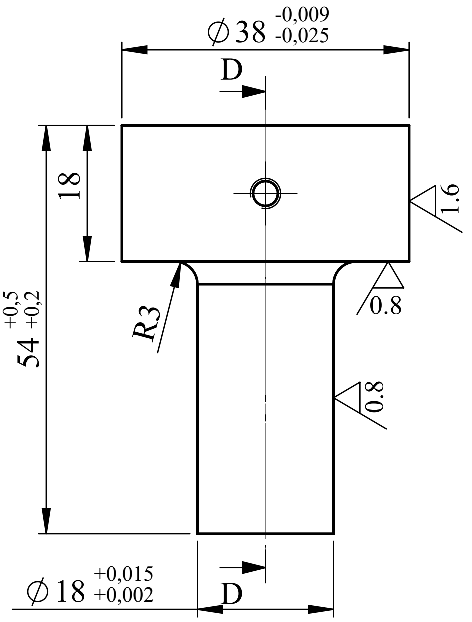
A2
UMBB-FT-DGM-CMM
CLASS OF 2022

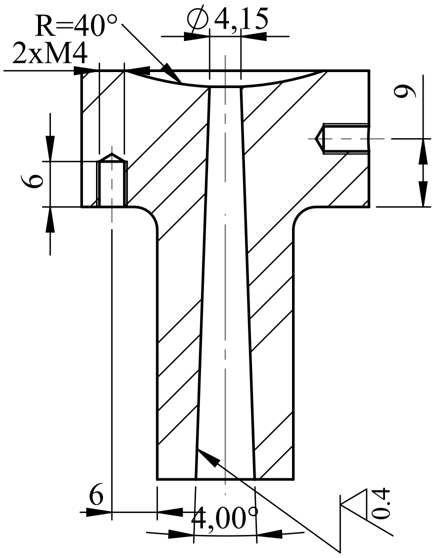


SECTION C-C

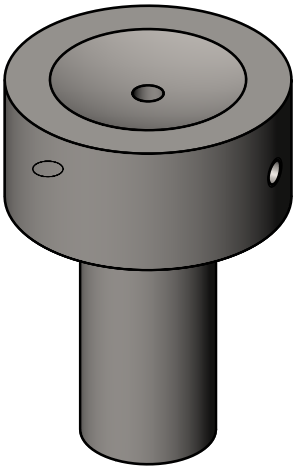
4	1	Locating ring	C45	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 2:3	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022



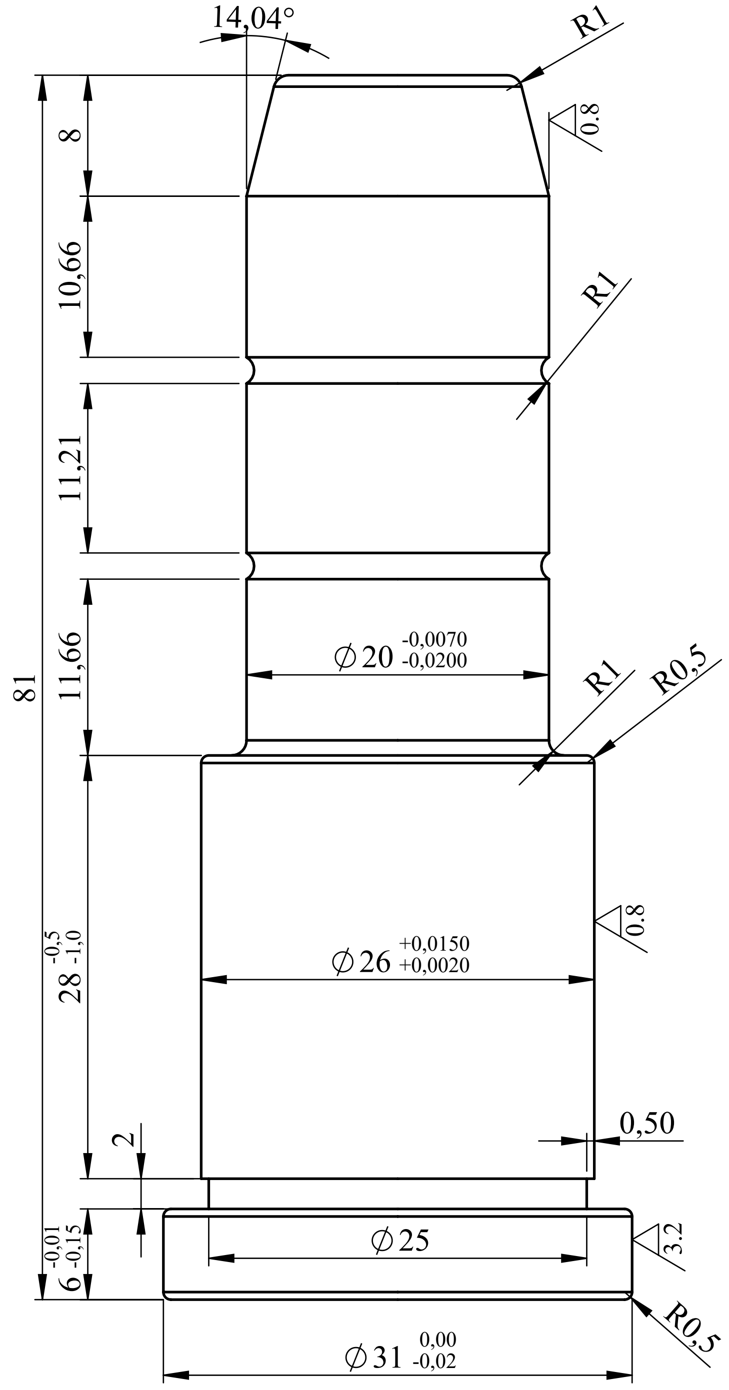


SECTION D-D



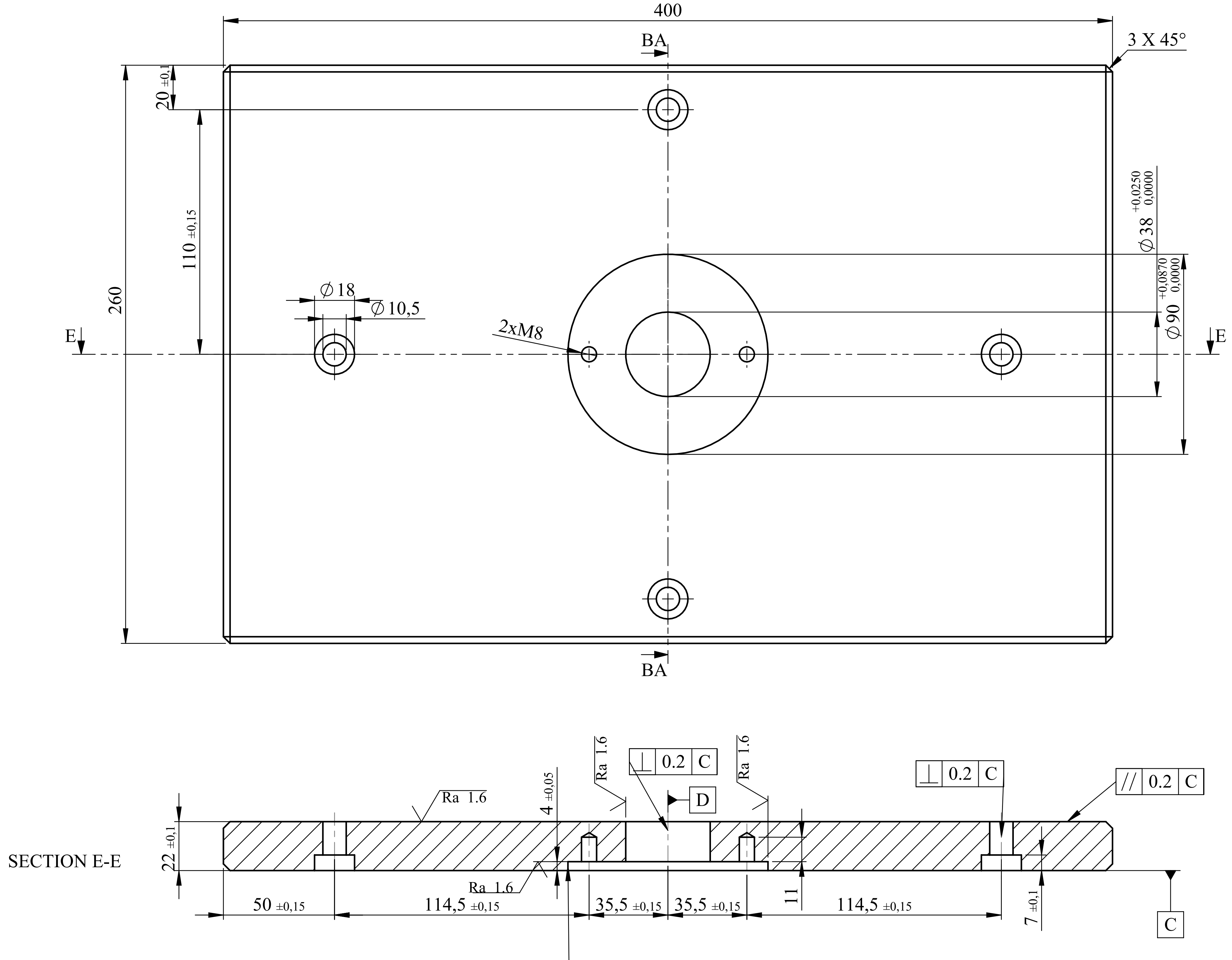
5	1	Sprue bushing	60MnSiCr4	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 1:1	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022



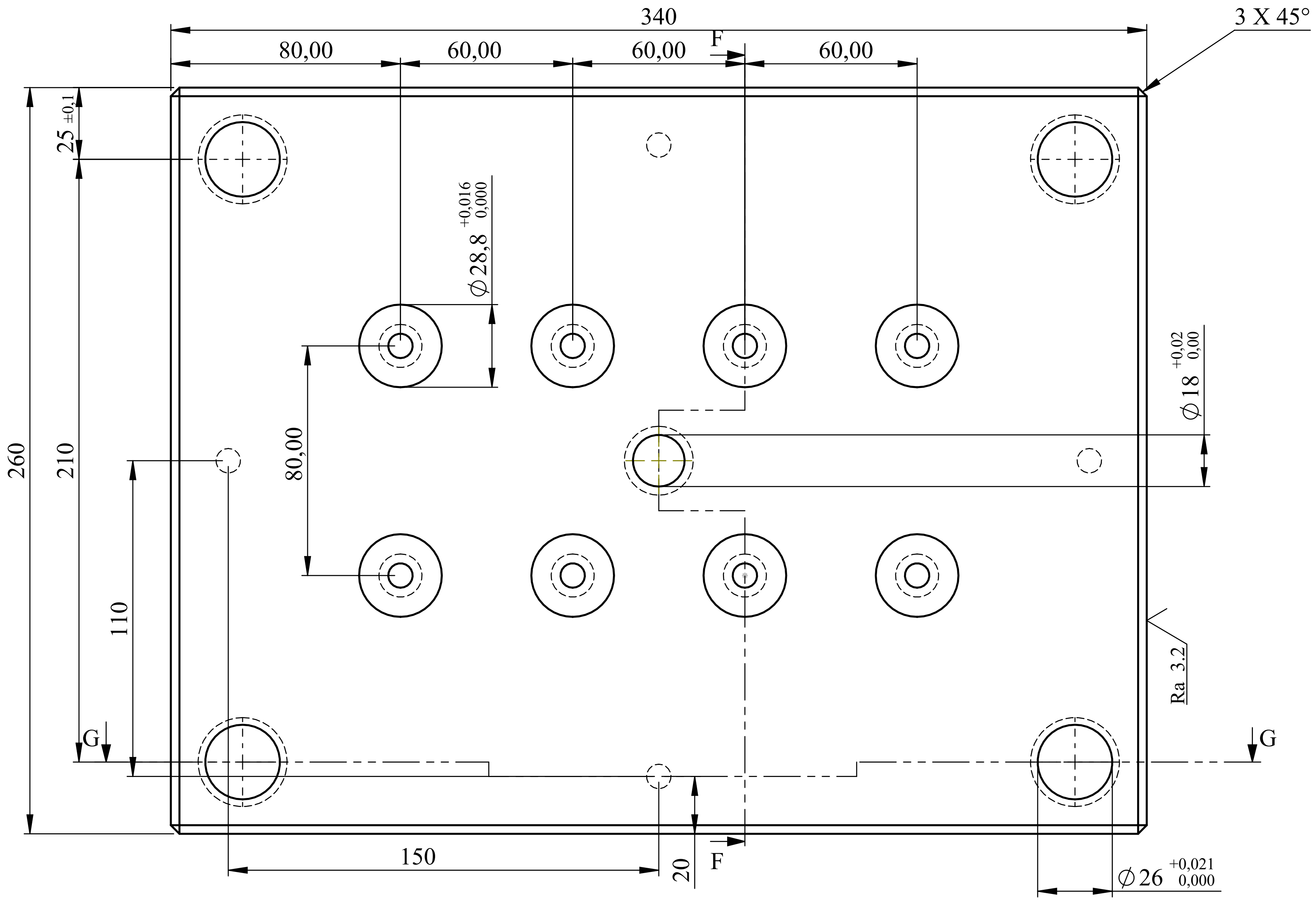
6	4	Leader pin	16MnCr5	
REP	NBR	TITLE	MATERIAL	OBSERVATION

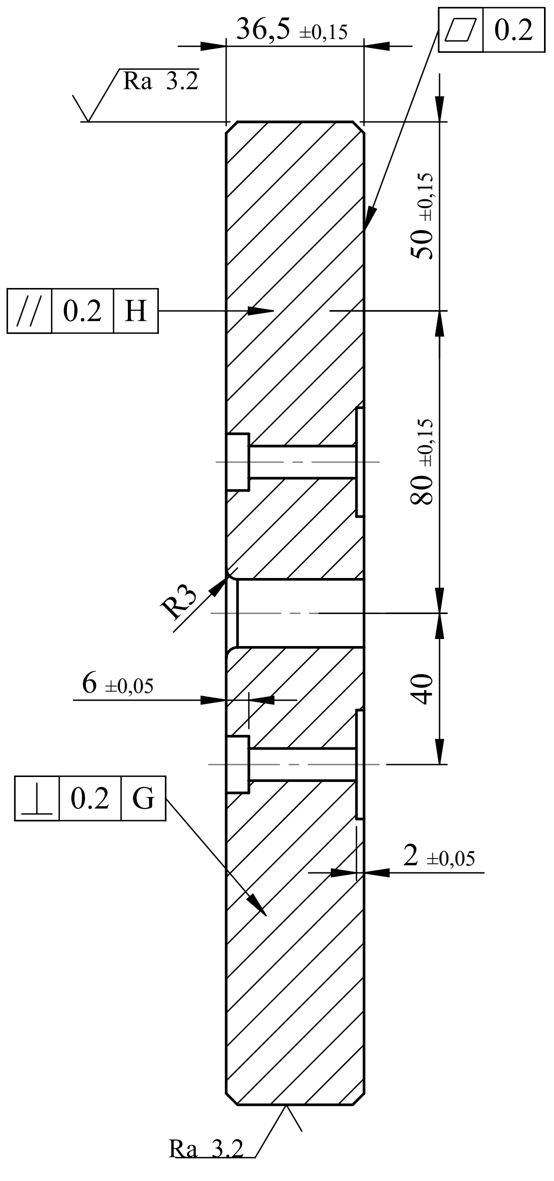
SCALE: 2:1	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022



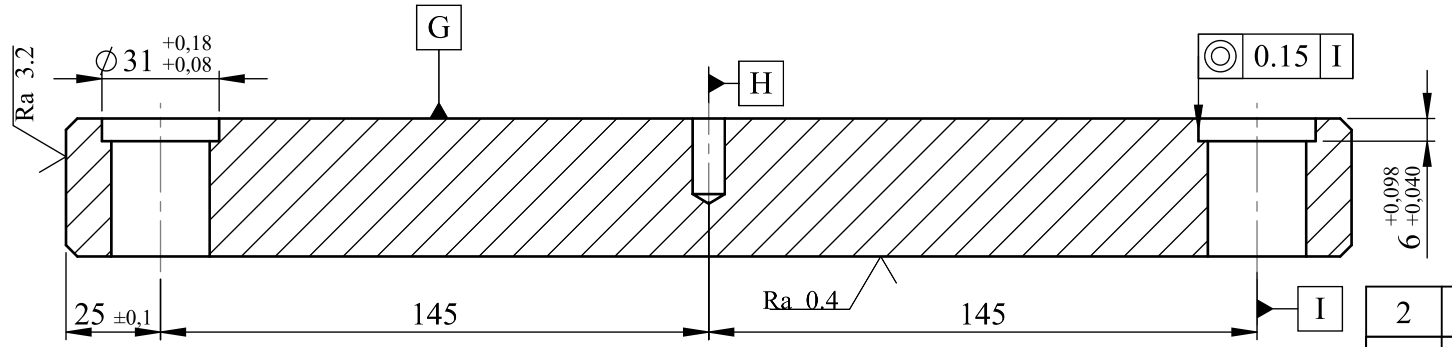
Tolerances IT=0.5 √Ra 3.2
Except what is mentioned

1	1	Top clamp plate	C30	
REP	NBR	TITLE	MATERIAL	OBSERVATION
SCALE: 1:2		**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**		**PRESENTED BY:** **DAKICHE MARWA**
				.../07/2022
A3		UMBB-FT-DGM-CMM		CLASS OF 2022





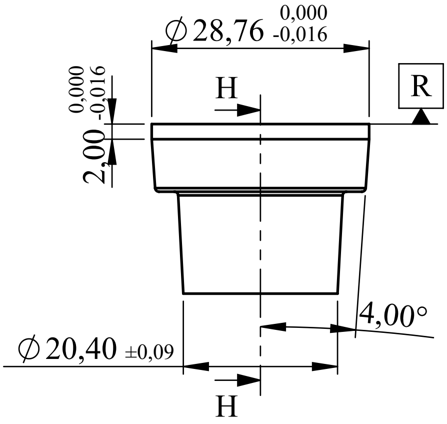
SECTION F-F

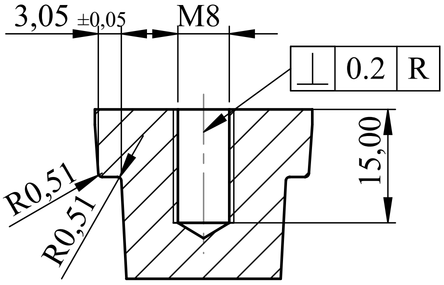


SECTION G-G

Tolerances IT=0.2 √Ra 1.6
Except what is mentioned

2	1	Core holder block	25CrMo4	
REP	NBR	TITLE	MATERIAL	OBSERVATION
SCALE: 1:2		**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**		**PRESENTED BY:** **DAKICHE MARWA**
⊏⊐ - ◎				.../07/2022
A3		UMBB-FT-DGM-CMM		CLASS OF 2022



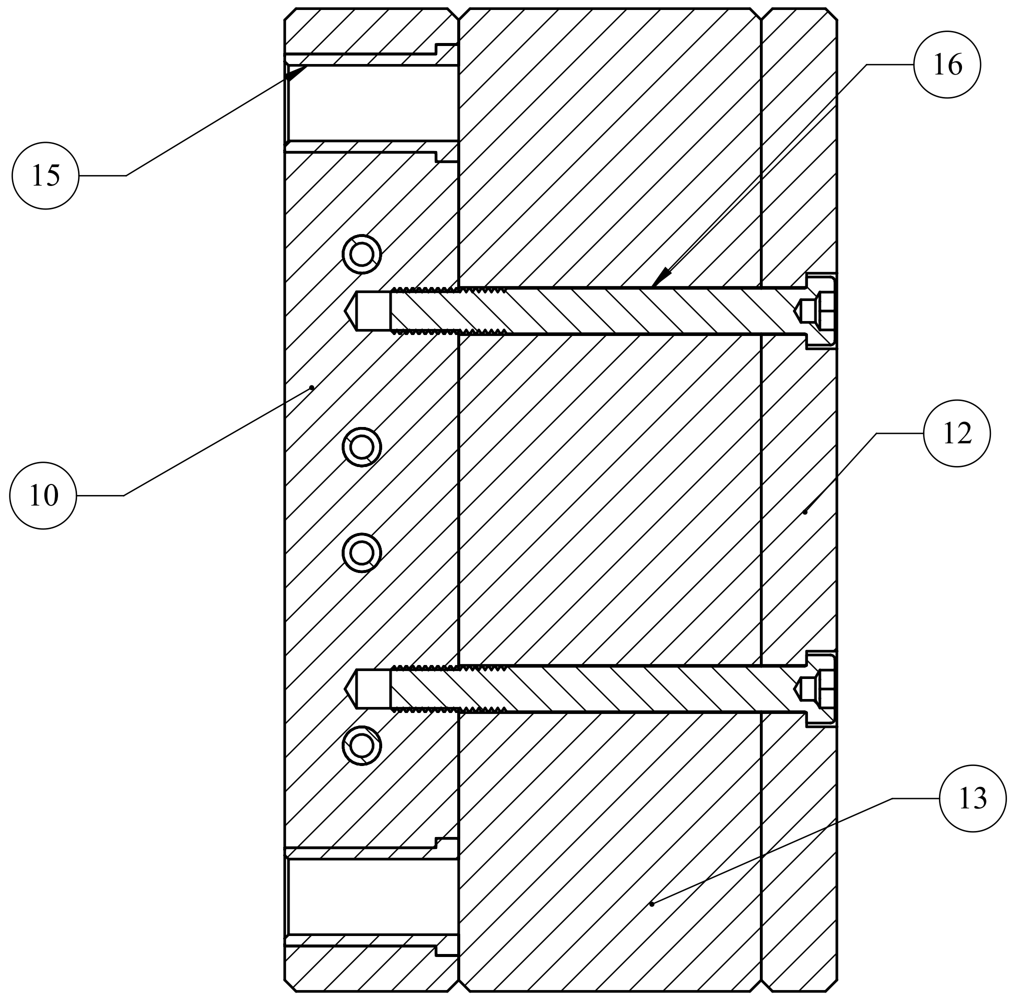


SECTION H-H

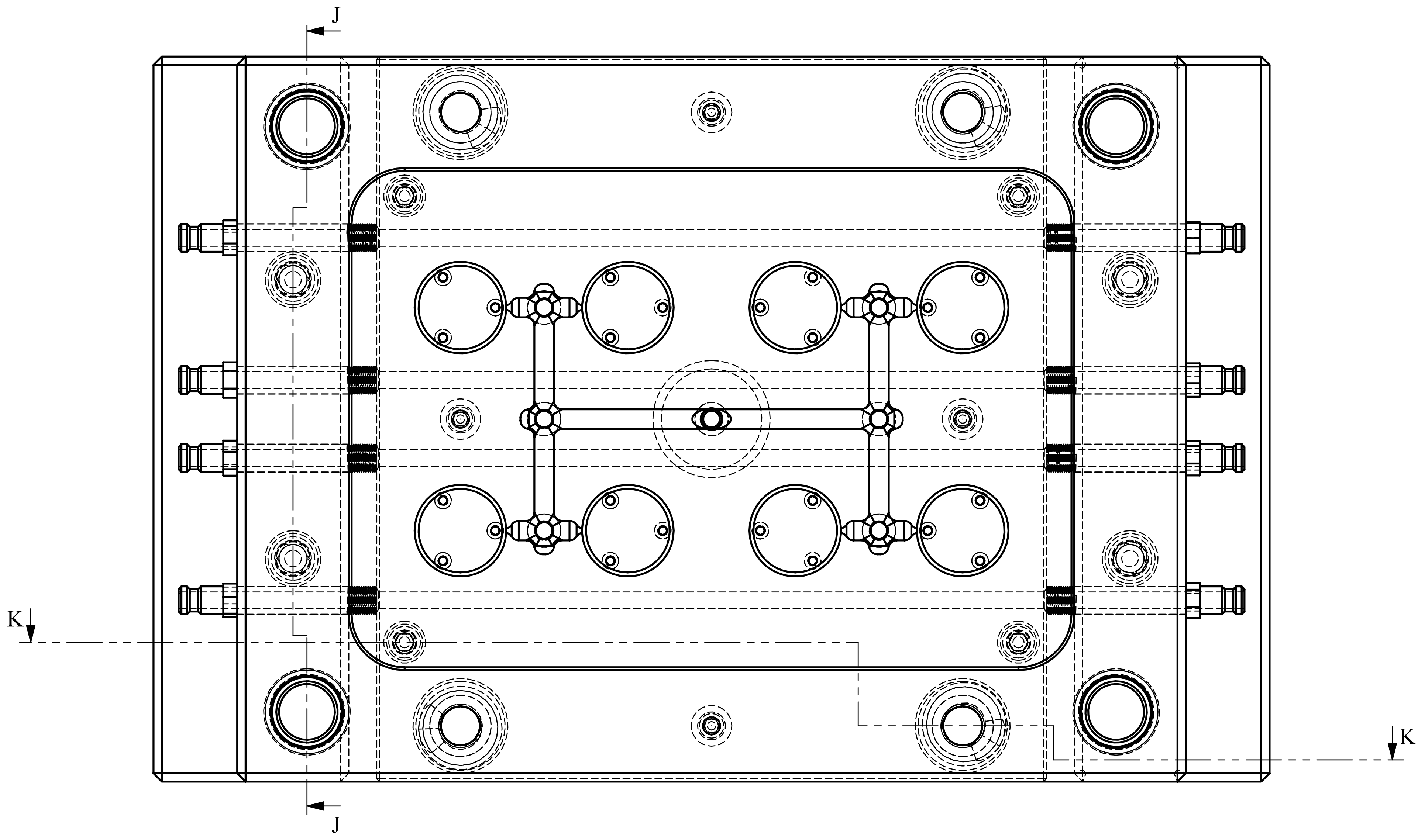
Tolerances IT=0.15 $\sqrt{\text{Ra } 1.6}$
Except what is mentioned

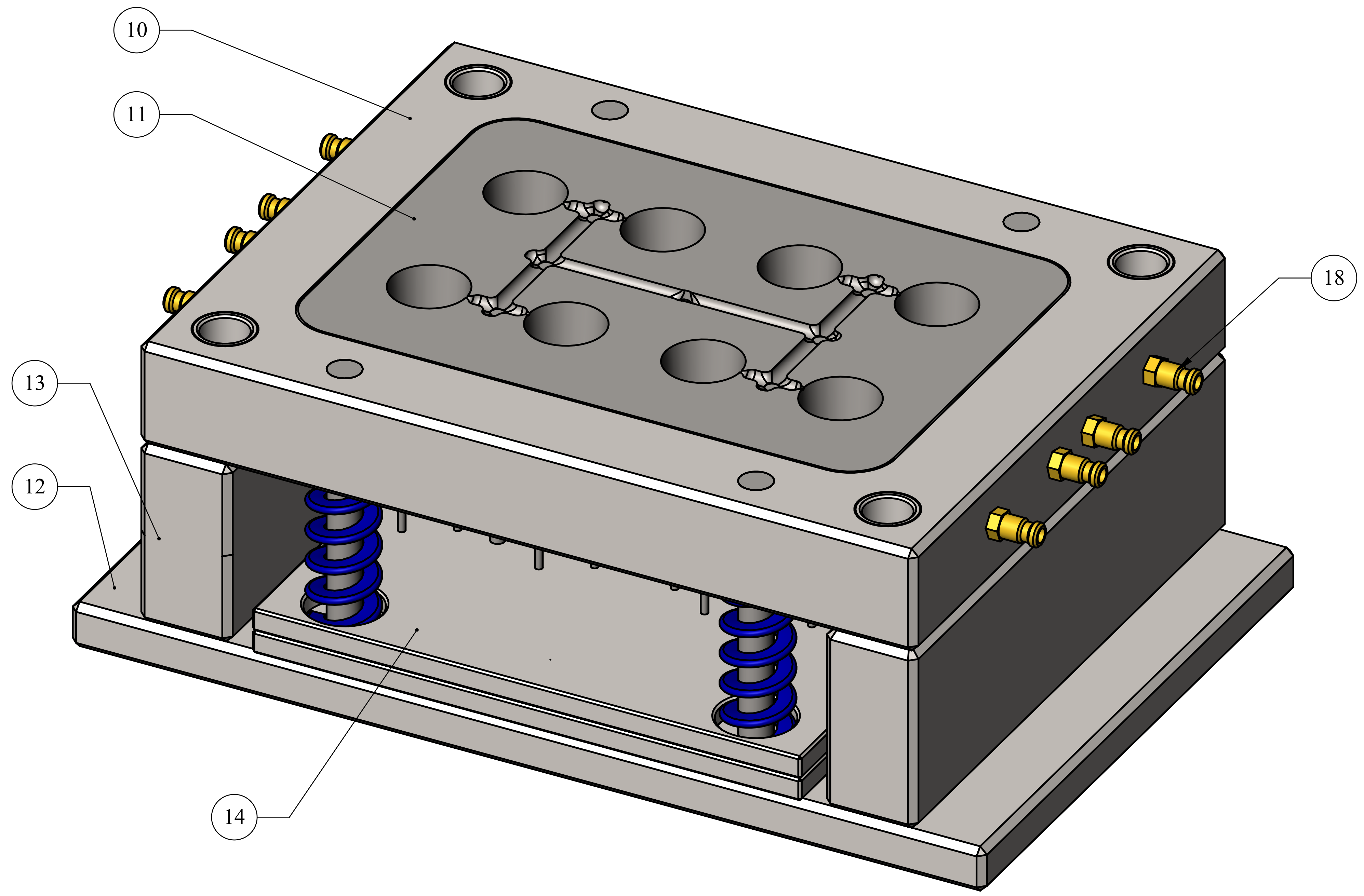
3	8	Core part	3Cr2Mo	
REP	NBR	TITLE	MATERIAL	OBSERVATION

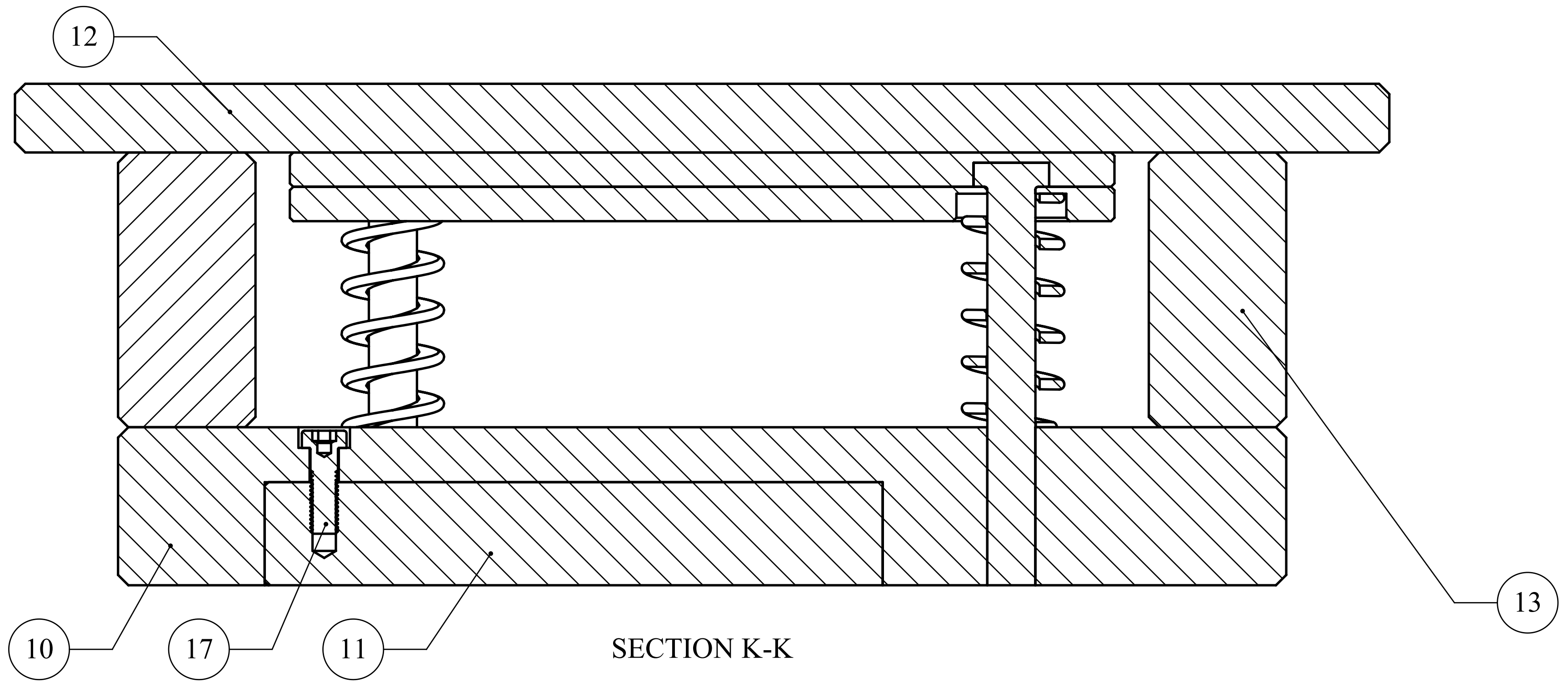
SCALE: 1:1	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022



SECTION J-J







SECTION K-K

Moving part sub assembly

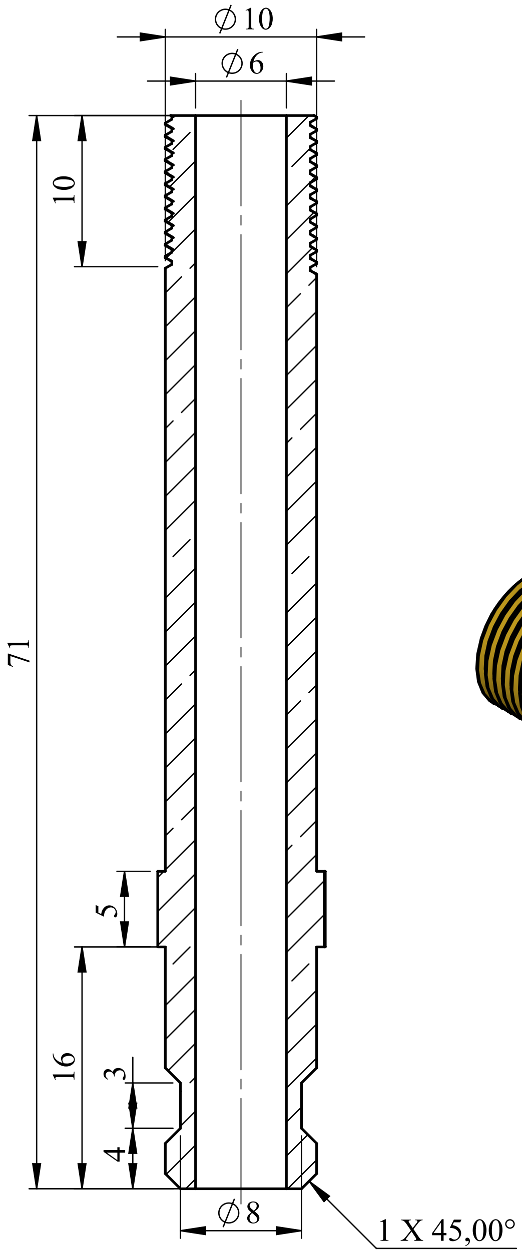
REP	NBR	TITLE	MATERIAL	OBSERVATION
18	8	Extension pipe	CuZn39Fb3	
17	4	HSH Screw M8x25	C35	
16	4	HSH screw M12x80	C35	
15	4	Leader pin bushing	16MnCr5	
14	1	Ejection system sub assembly		
13	2	Spacer plate	C30	
12	1	Bottom clamp plate	C30	
11	1	Cavity plate	3Cr2Mo	
10	1	Cavity holder block	25CrMo4	

SCALE: 1:2

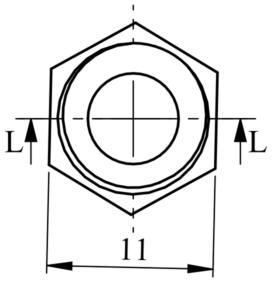
INJECTION MOLD
FOR A WHEELBARROW'S
WHEEL SPACER

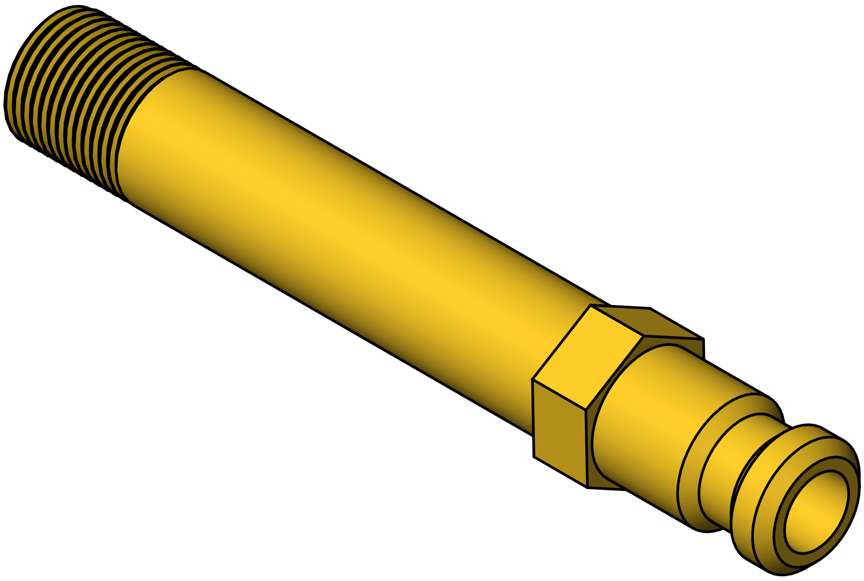
PRESENTED BY:
DAKICHE MARWA
.../07/2022

A2
UMBB-FT-DGM-CMM
CLASS OF 2022



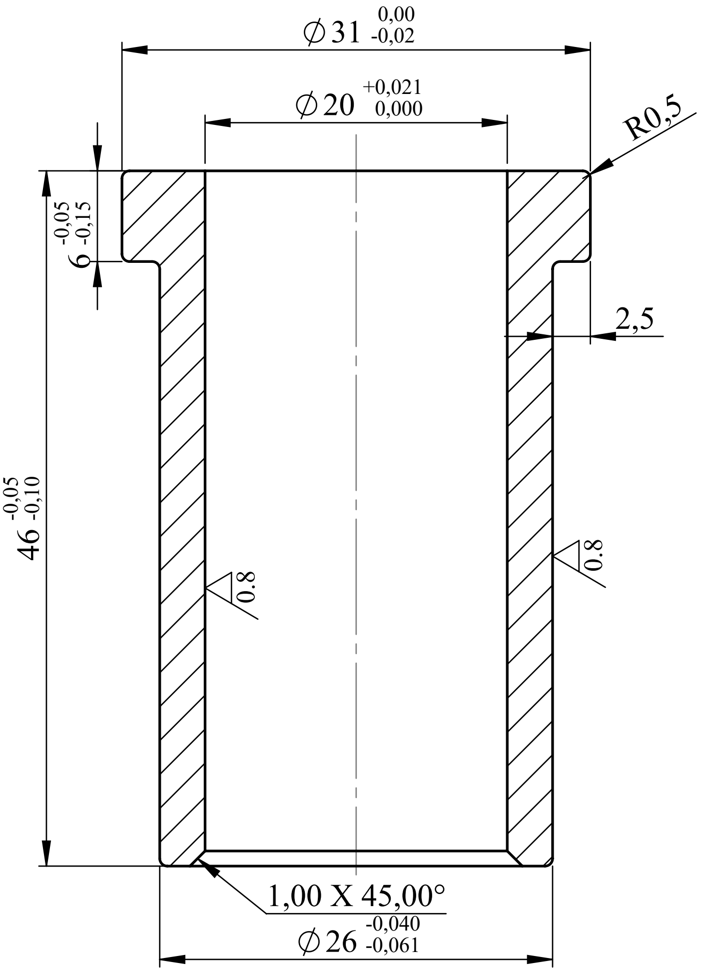
SECTION L-L



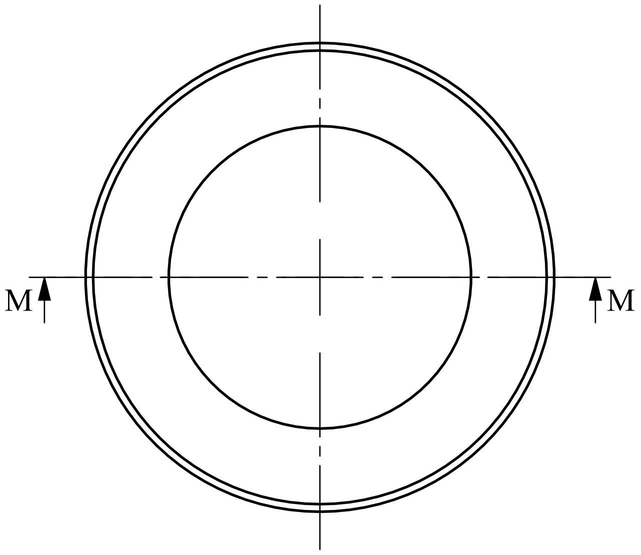


18	8	Extension pipe	CuZn39Pb3	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 2:1	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022



SECTION M-M



15	4	Leader pin bushing	16MnCr5	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 2:1

INJECTION MOLD
FOR A WHEELBARROW'S
WHEEL SPACER

PRESENTED BY:

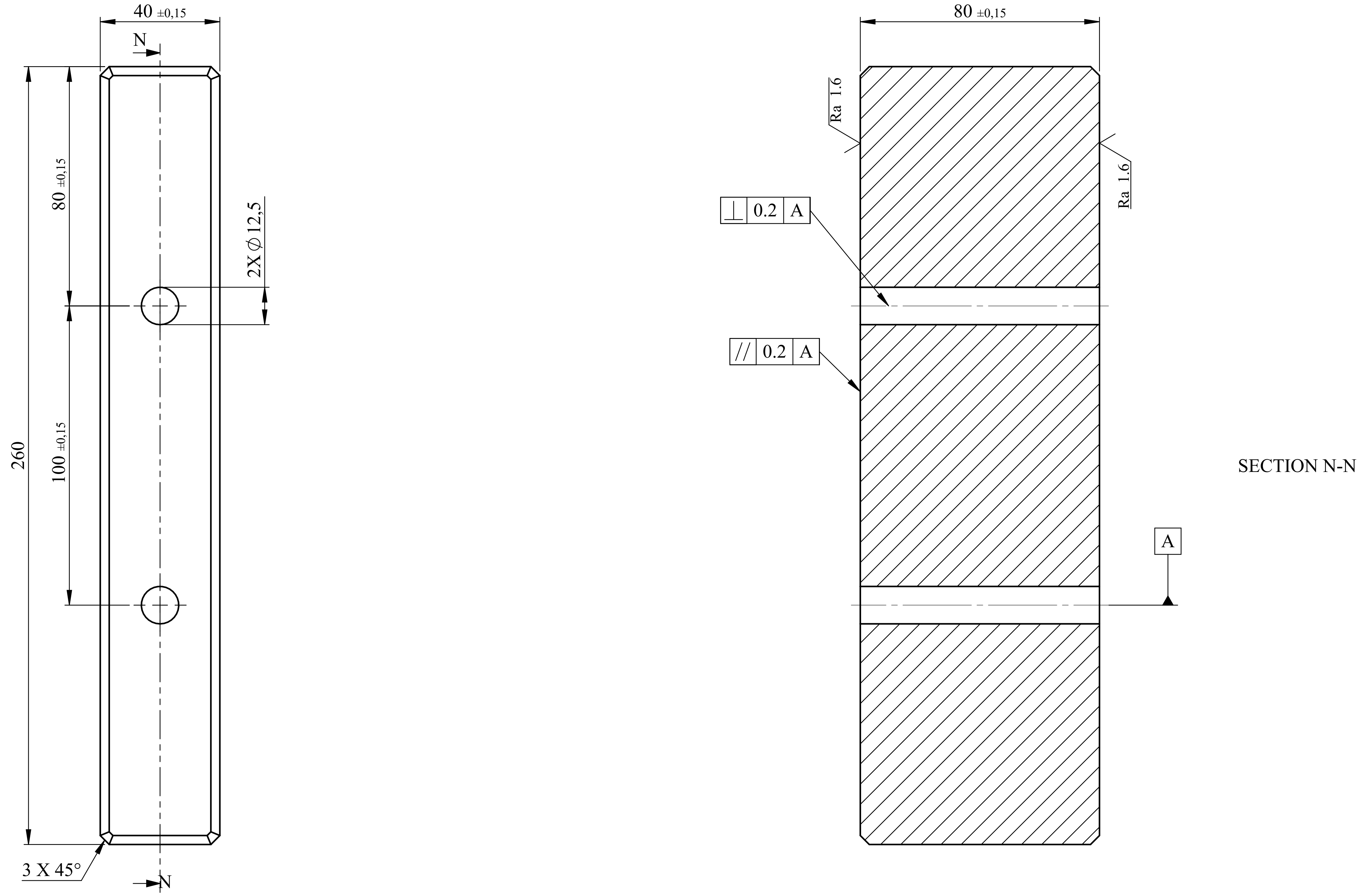
DAKICHE MARWA

.../07/2022

A4

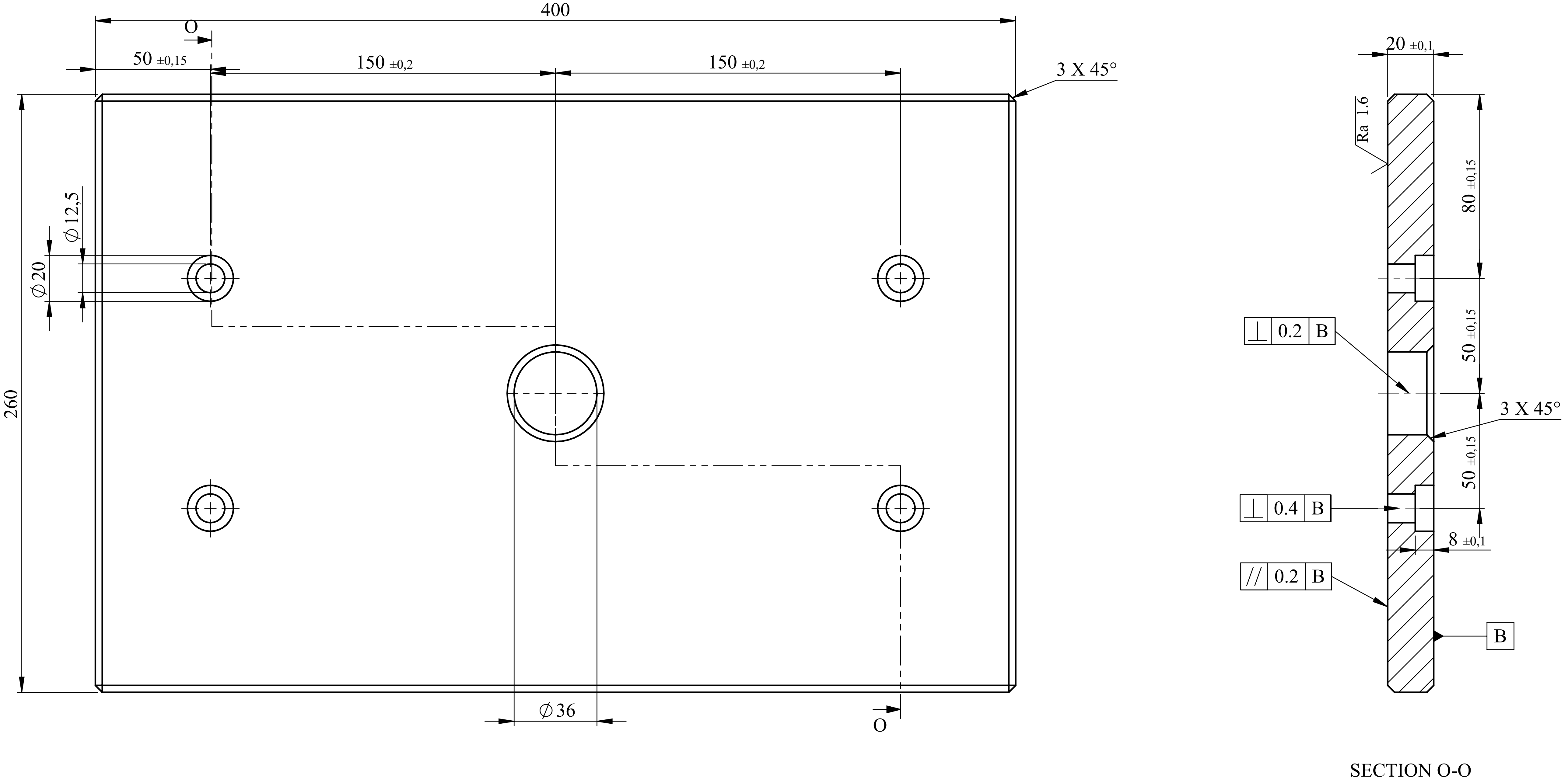
UMBB-FT-DGM-CMM

CLASS OF 2022



Tolerances IT=0.5 √Ra 3.2
Except what is mentioned

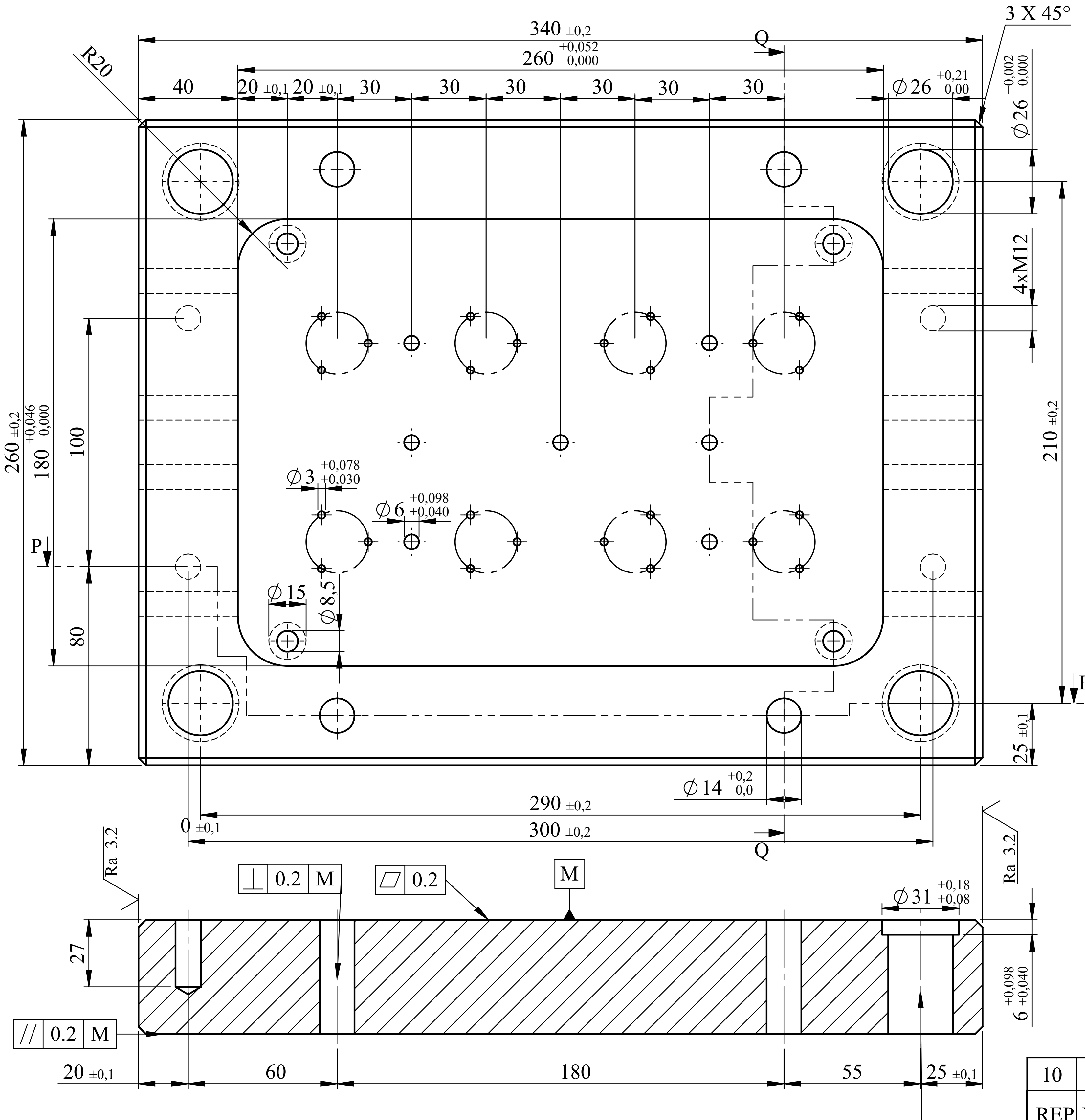
13	2	Spacer plate	C30	
REP	NBR	TITLE	MATERIAL	OBSERVATION
SCALE: 4:6		**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**		**PRESENTED BY:** **DAKICHE MARWA**
⊲ - ◎				.../07/2022
A3		UMBB-FT-DGM-CMM		CLASS OF 2022



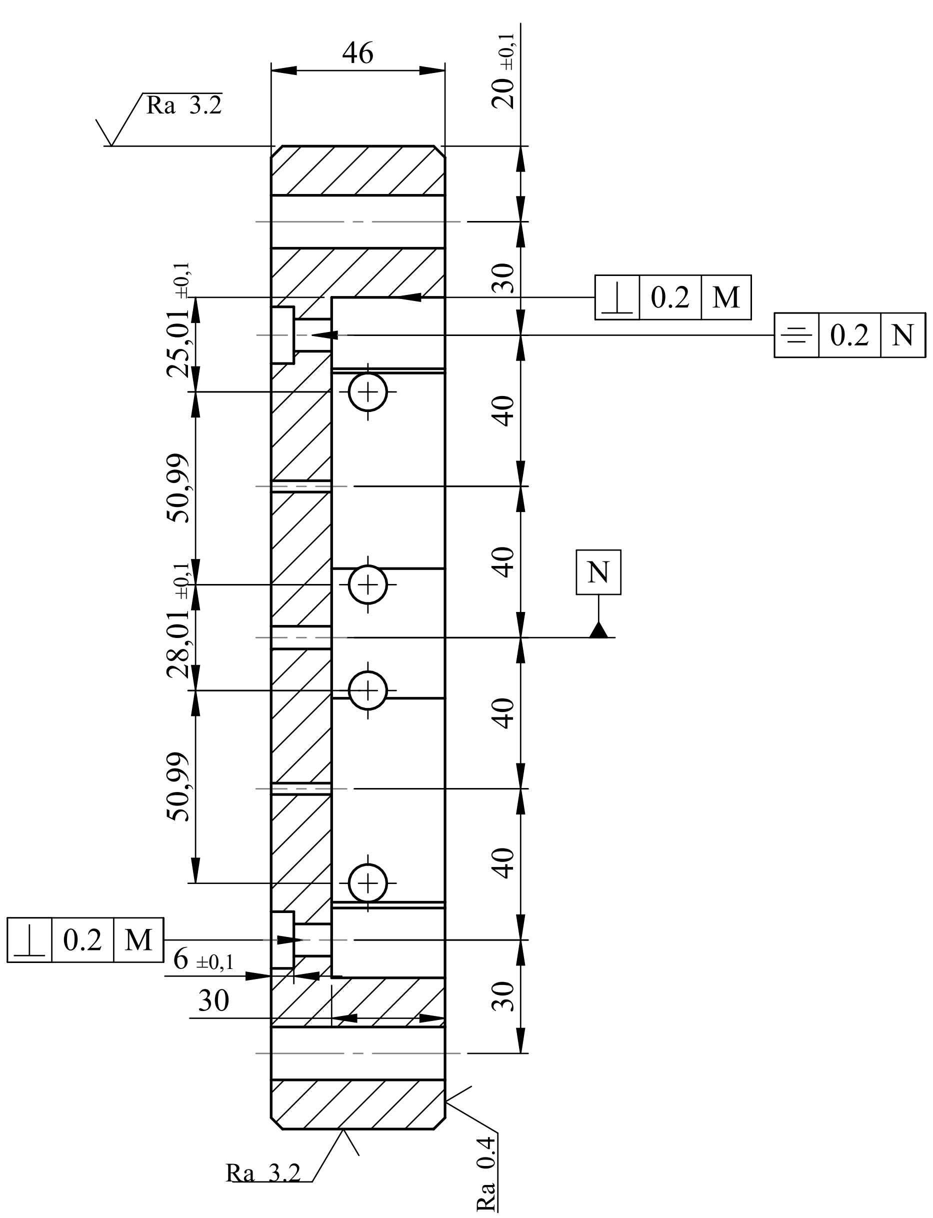
Tolerances IT=0.5 $\sqrt{\text{Ra } 3.2}$
Except what is mentioned

12	1	Bottom clamp plate	C30	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 1:2	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A3	UMBB-FT-DGM-CMM	CLASS OF 2022



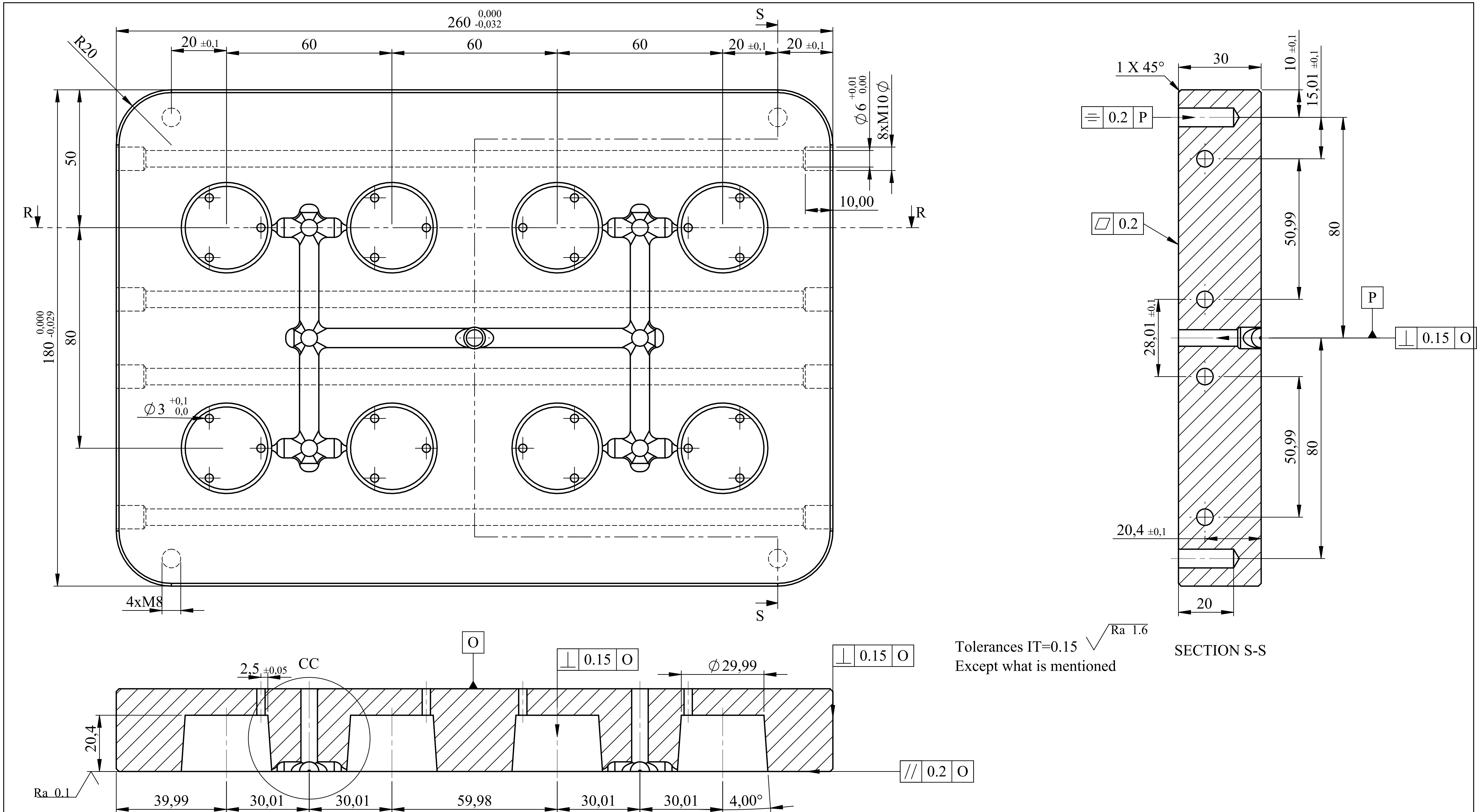
SECTION P-P



SECTION Q-Q

Tolerances IT=0.15 √Ra 1.6
Except what is mentioned

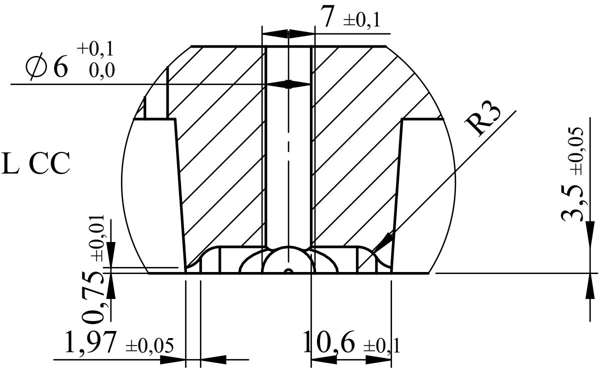
10	1	Cavity holder block	25CrMo4	
REP	NBR	TITLE	MATERIAL	OBSERVATION
SCALE: 1:2		**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**		**PRESENTED BY:** **DAKICHE MARWA**
				.../07/2022
A3		UMBB-FT-DGM-CMM		CLASS OF 2022



Tolerances IT=0.15
Except what is mentioned

SECTION S-S

SECTION R-R



DETAIL CC

11	1	Cavity plate	3Cr2Mo	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 3:4

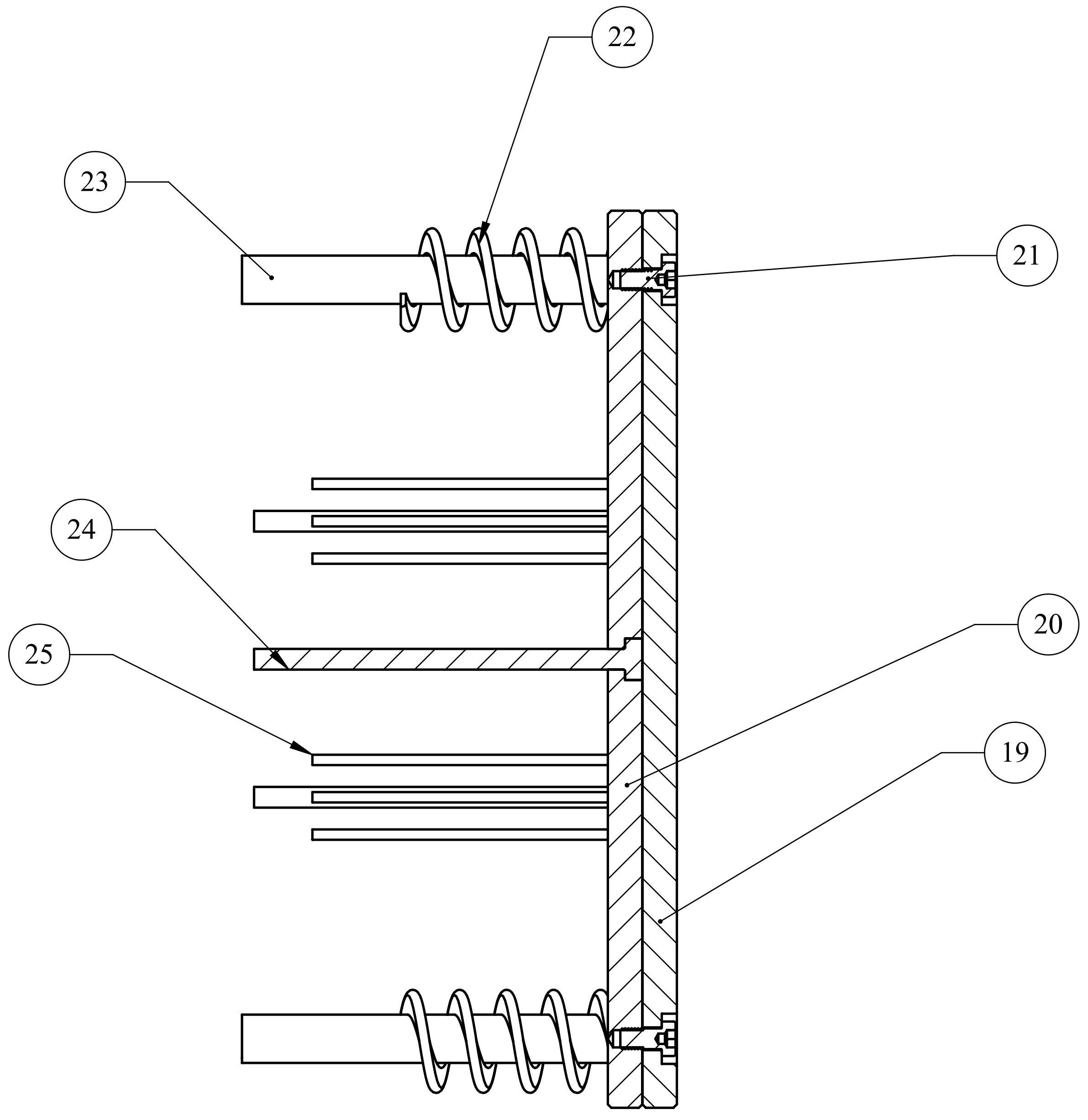
INJECTION MOLD
FOR A WHEELBARROW'S
WHEEL SPACER

PRESENTED BY:
DAKICHE MARWA
.../07/2022

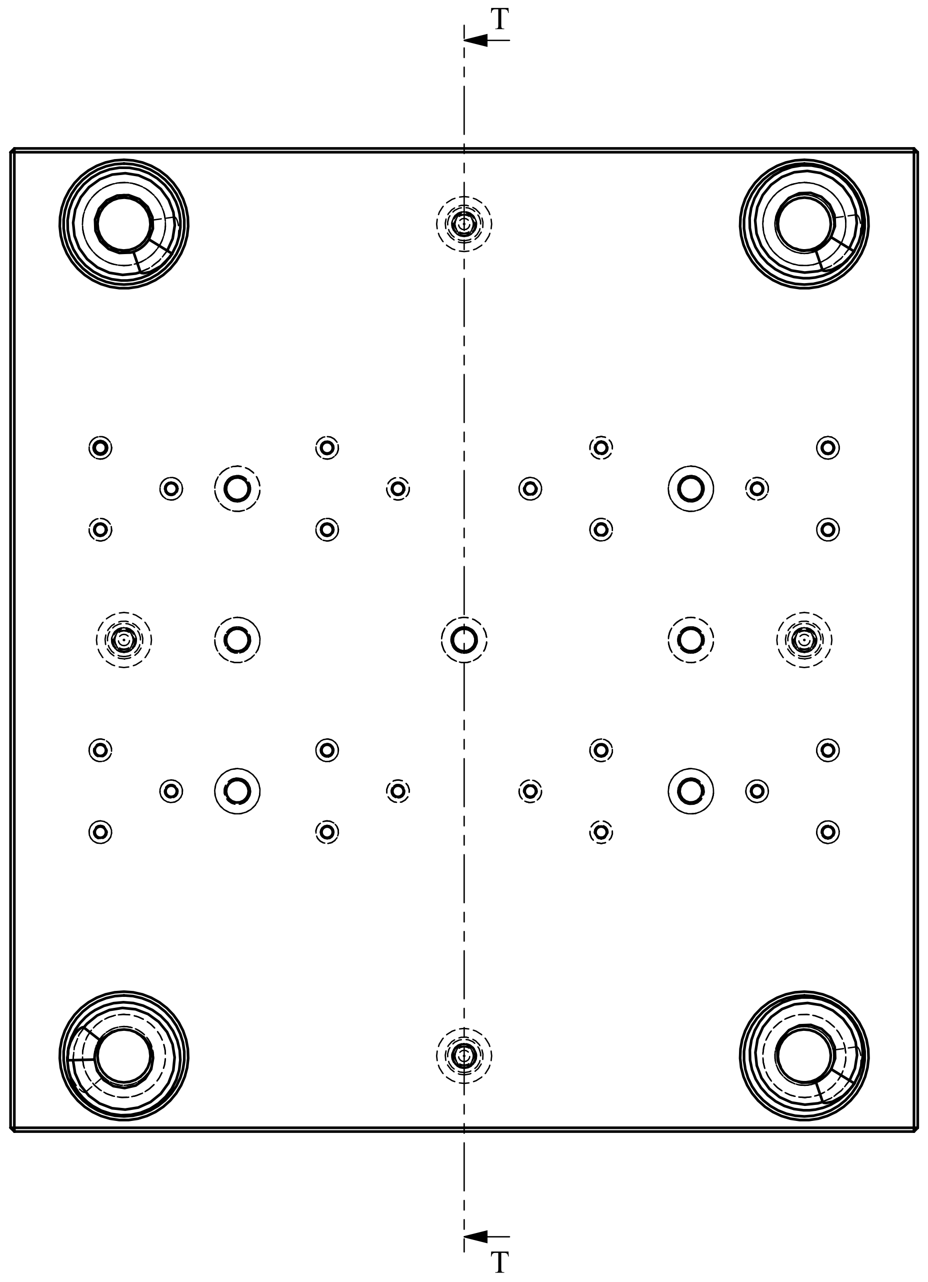
A3

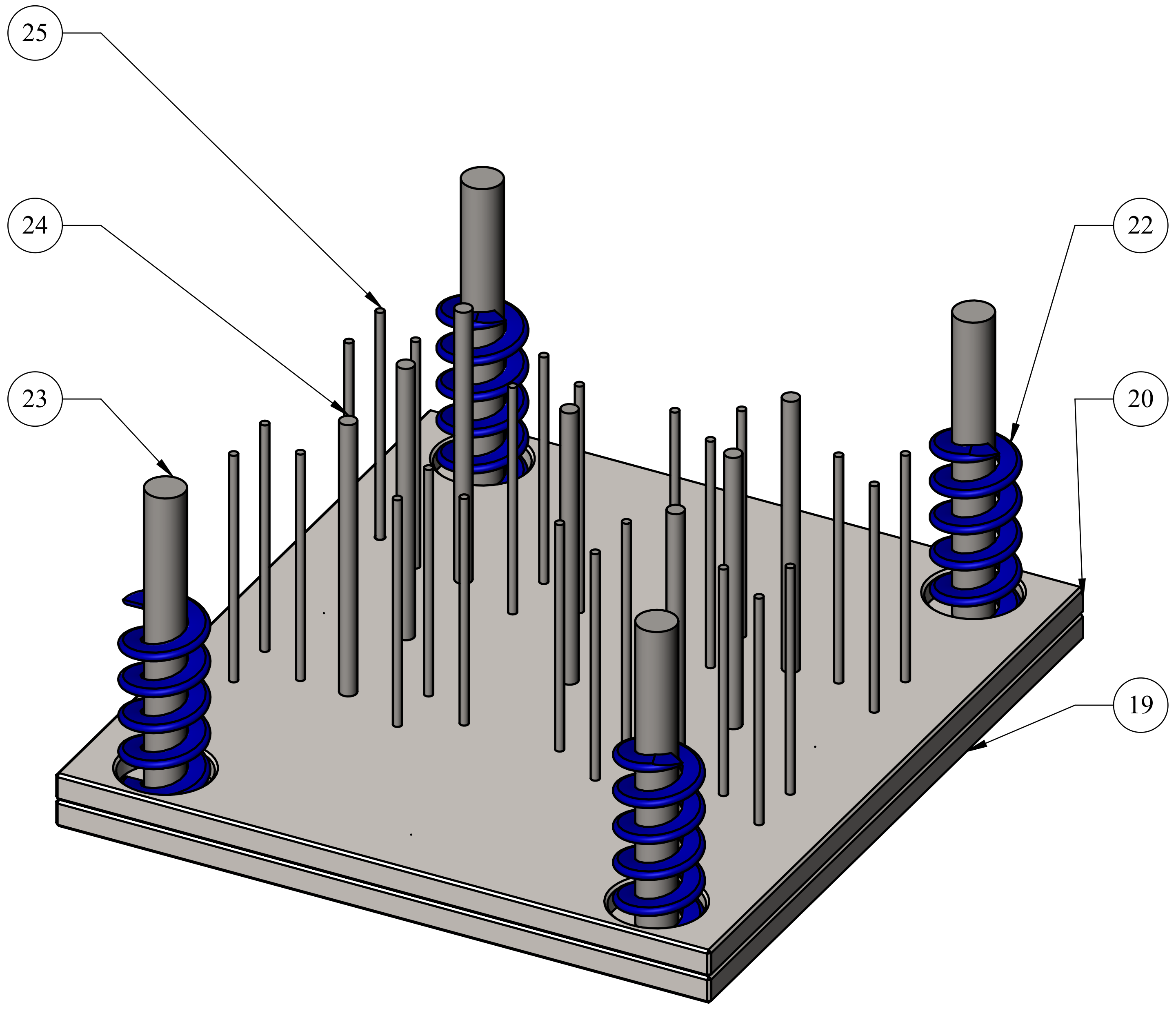
UMBB-FT-DGM-CMM

CLASS OF 2022



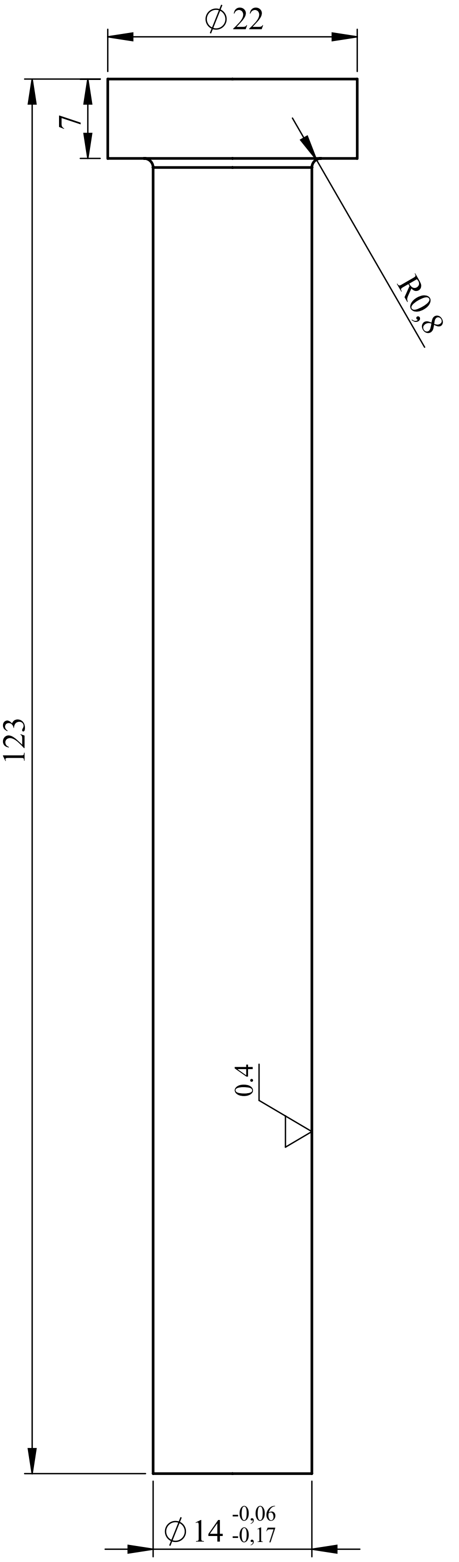
SECTION T-T





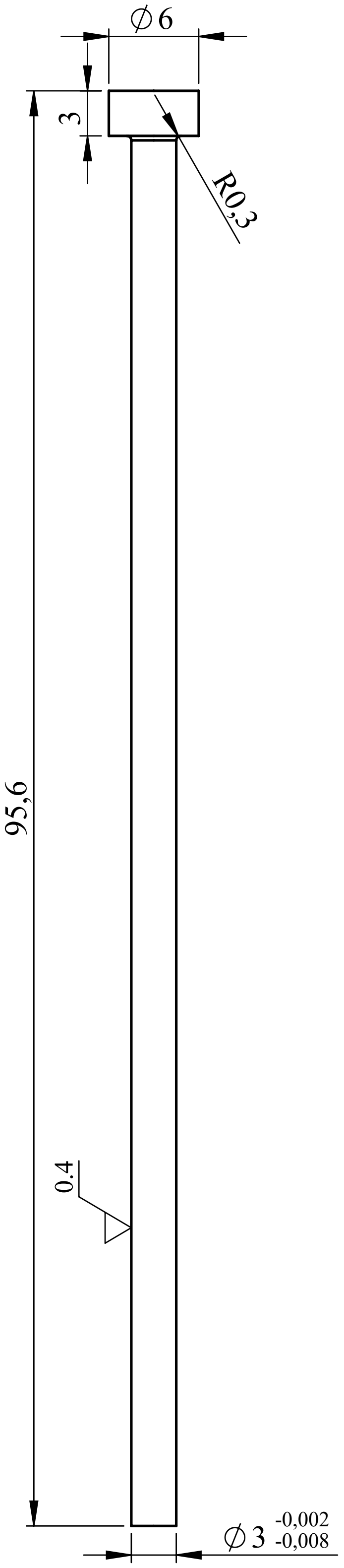
Ejection system sub assembly				
25	24	Cavity ejector pin	100Cr6	
24	7	Runners ejector pin	100Cr6	
23	4	Return pin	100Cr6	
22	4	Blue spring slat round wire		See catalog
21	4	HSH screw M6x12	C35	
20	1	Ejector plate	C22	
19	1	Ejector retainer plate	C22	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 1:2	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA** .../07/2022
A2	UMBB-FT-DGM-CMM	CLASS OF 2022



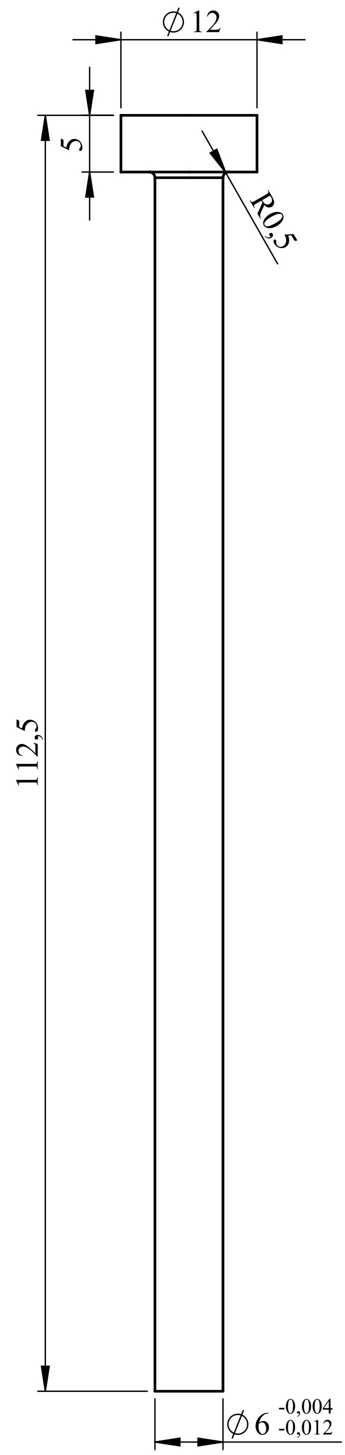
23	4	Return pin	100Cr6	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 3:2	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022



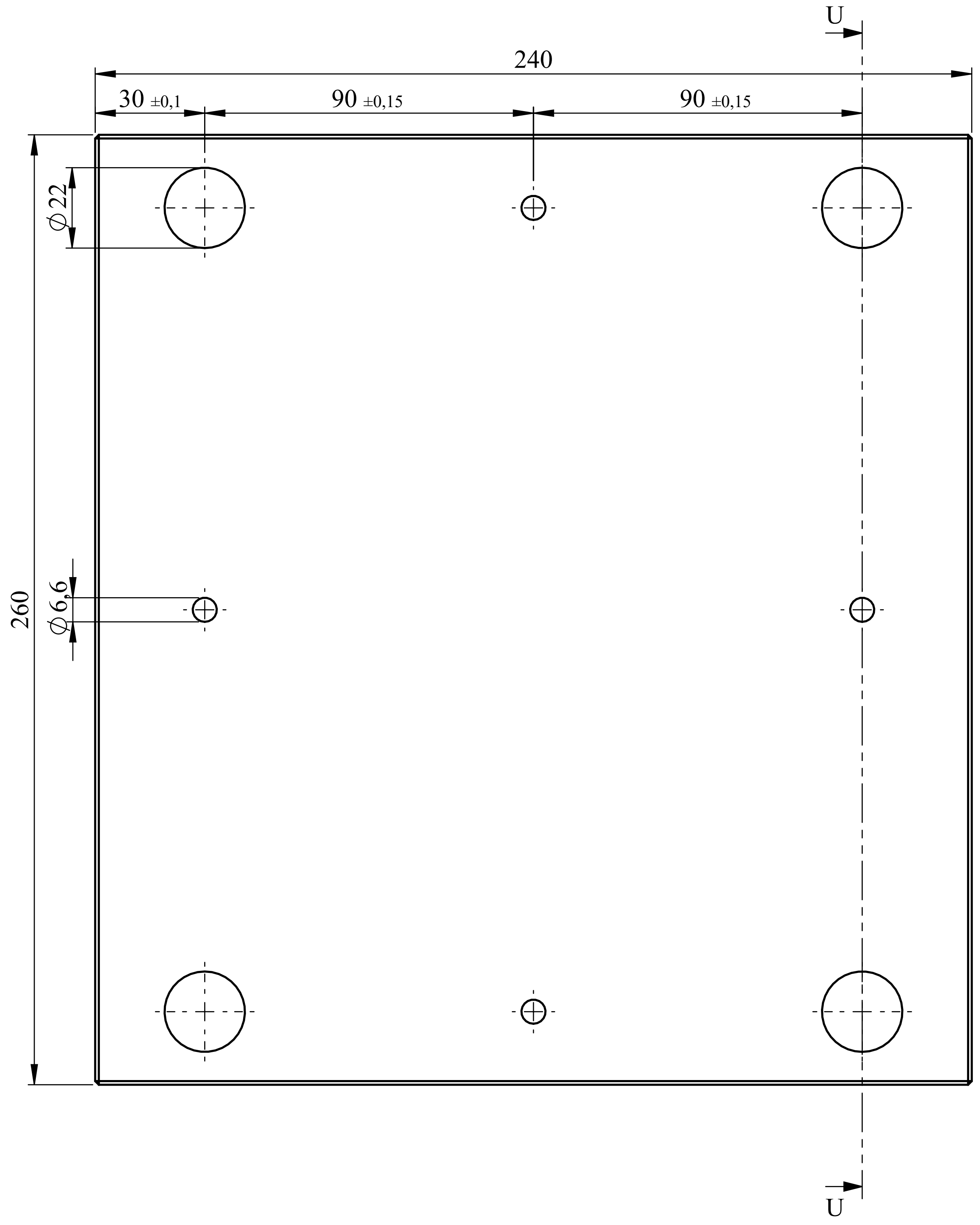
25	24	Cavity ejector pin	100Cr6	
REP	NBR	TITLE	MATERIAL	OBSERVATION

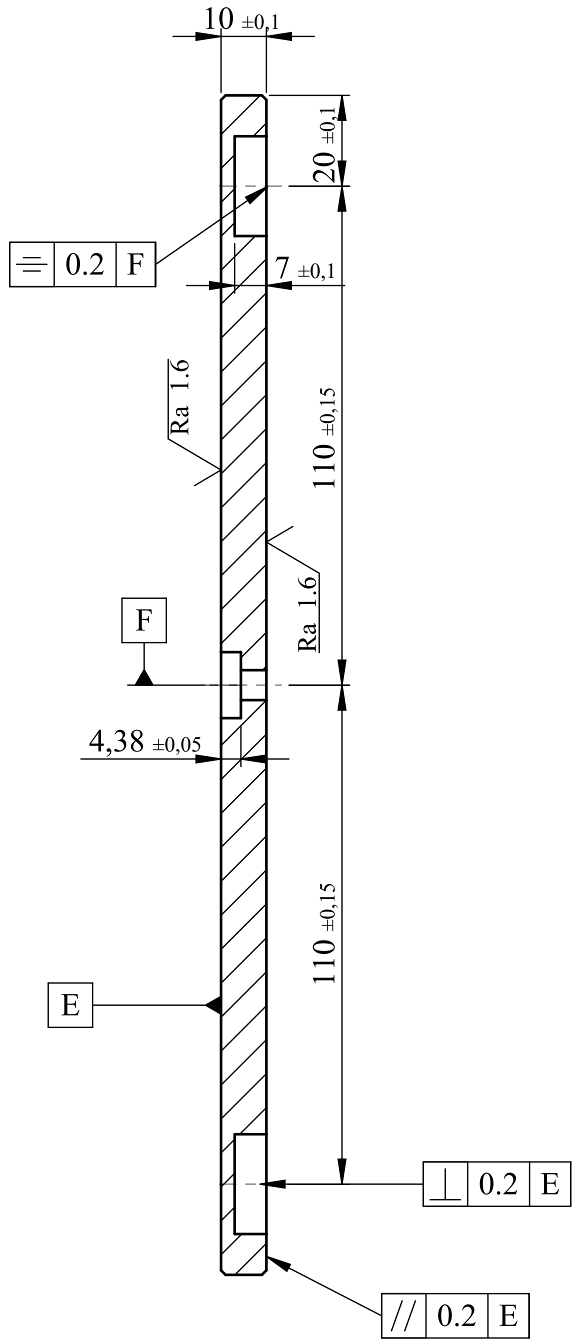
SCALE: 2:1	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022



24	7	Runner ejector pin	100Cr6	
REP	NBR	TITLE	MATERIAL	OBSERVATION

SCALE: 3:2	**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**	**PRESENTED BY:** **DAKICHE MARWA**
		.../07/2022
A4	UMBB-FT-DGM-CMM	CLASS OF 2022

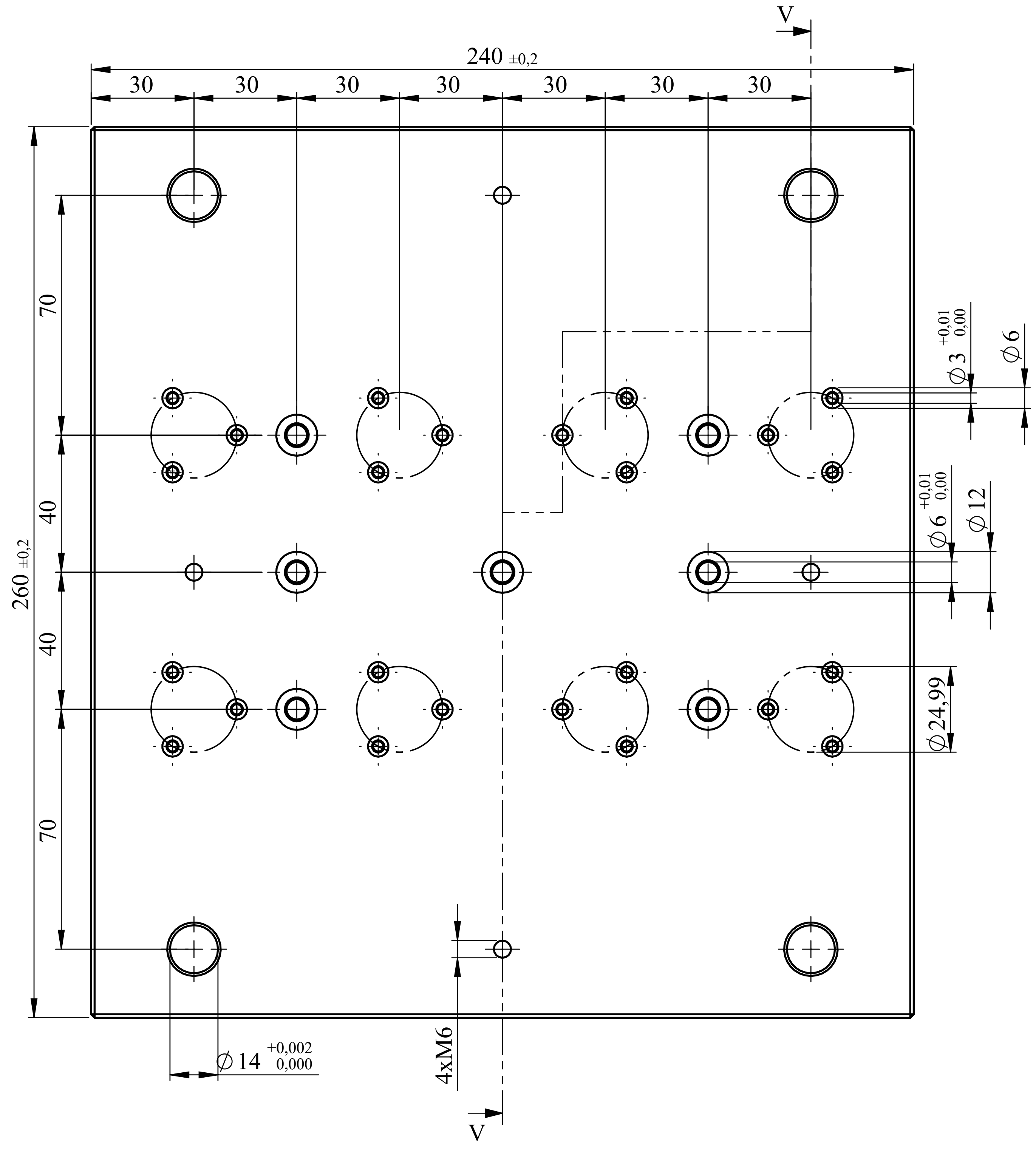


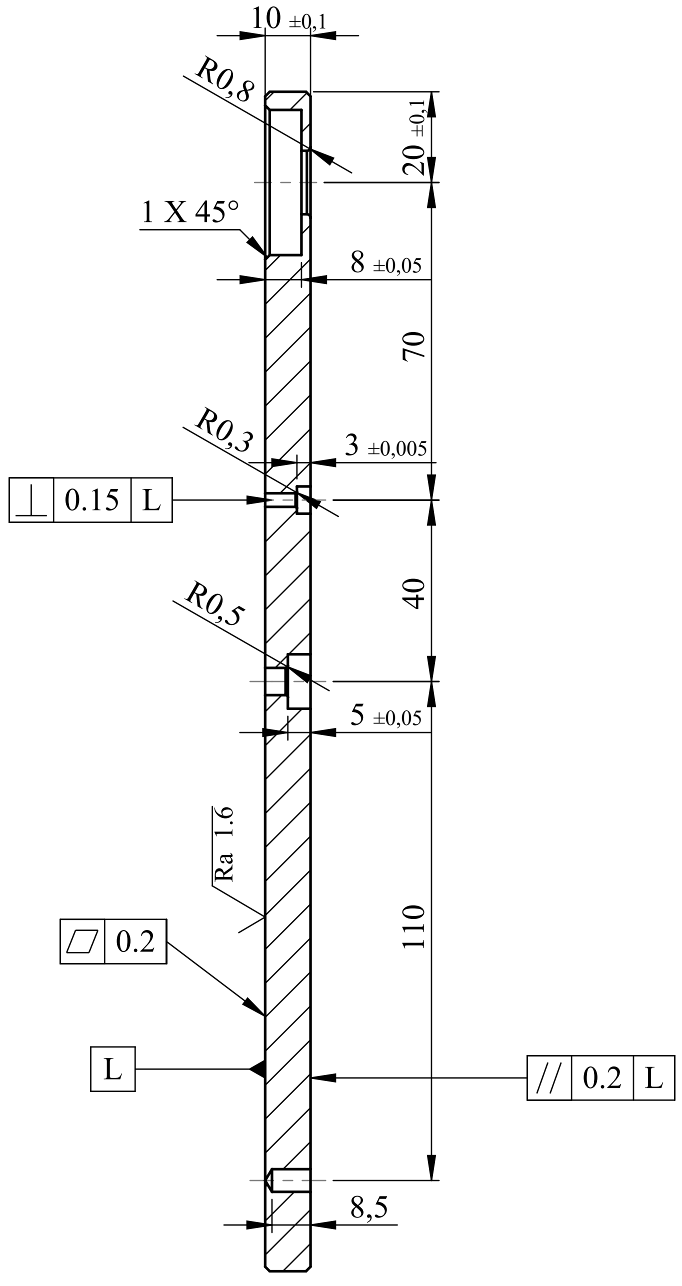


SECTION U-U

Tolerances IT=0.5 √Ra 3.2
Except what is mentioned

19	1	Ejector retainer plate	C22	
REP	NBR	TITLE	MATERIAL	OBSERVATION
SCALE: 3:5		**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**		**PRESENTED BY:** **DAKICHE MARWA**
				.../07/2022
A3		UMBB-FT-DGM-CMM		CLASS OF 2022





SECTION V-V

Tolerances IT=0.15 √Ra 3.2
Except what is mentioned

20	1	Ejector plate	C22	
REP	NBR	TITLE	MATERIAL	OBSERVATION
SCALE: 3:4		**INJECTION MOLD FOR A WHEELBARROW'S WHEEL SPACER**		**PRESENTED BY: DAKICHE MARWA**
				.../07/2022
A3		UMBB-FT-DGM-CMM		CLASS OF 2022