ARCADA

Degree Thesis:

Design, Simulation and Analysis for Injection Moulding of Manual Juice Maker

Plastics Technology, PTE

By: Kibrom Mesfin

ID: 10328

Date: Oct 07, 2014

DEGREE THESIS	
Arcada	
Degree programme	*Plastics Technology*
Identification Number	*10328*
Author	*Kibrom Mesfin*
Title	*Product Design, Simulation and Analysis for Injection Moulding of Manual Juice Maker*
Supervisor	*Mathew Vihtonen*

Abstract

As of the fact that most plastic products are manufactured using injection molding process now a days, this thesis project brings to the surface the study of the underlying principle and theories of mold designing basics and proceeds along the way with a designing and simulation of a plastic product manual juice maker showing theories, steps, methods, designing, simulation and analysis.

A detail study of product design carried on for an injection molding design and simulation. A basic study of plastics principle and materials of the part design studied in a basic level with a progressing hierarchical building of points of mould designing basics and functions. This thesis study aims on designing, simulation and analysis of an enhanced squeezing mechanism manual juice maker, the part is designed using the software SolidWorks in a successive two phases starting from the Dom then the containing jar with finally incorporating both as a single Manual Juice Maker design after the required criteria and features of the Manual Juice Makers separate components achieved. On top of that keeping its uniqueness and identities from the scratch level of research study on a basic sample fruit dimensions, geometry and way of squeezing focusing with manufacturing a Manual juice Maker with low coast plastic material and saving material coast, most of all meeting the objectives of this product design for enabling maximum squeezing capacity of the Dom with enhanced added designing features on it for the clockwise rotation of the fruit. In addition an easy, handy and elegant designing of the containing jar achieved that incorporated together.

Along with this thesis study, after the required product designed achieved using the software SolidWorks, then carried on different realistic parameters and assumptions of the part design studied, then applied for simulation analysis of the plastic melt using the software Autodesk Moldflow Synergy and here by, based on the analysis the melt flow is analyzed and optimum results are found. At the final stage these results compared, contrasted and forecasting this values and results for the real world injection molding production of the manual juice maker on the ground level.

Keywords	*Part Design, Mould Design, Injection Moulding, SolidWorks, Autodesk Moldflow Synergy, Melt Flow Simulation and Analysis, Dom, Manual Juice Maker*
Number of pages	*92*
Language	*English*
Date of acceptance	

Acknowledgment

For so much of thankfulness, sincere gratitude goes to my Supervisor Mathew Vihtonen and Examiner Rene Herrmann for the guidance, comment, support of the thesis and all along the course study year experience and help. Also I would like to thank Erland Nyroth for his assistant on the progress and manufacturability feasibility of the thesis product part design, it also worth mentioning Catia Toress for her advices and support during the study.

Secondly, a special thanks to my brother Ermias Mesfin for his intervention from early initial stage of the study and the thesis till I make big giant jump of finalizing it in all aspects of backing me up and braking the curtesy in creating for me the comfort zone for accomplishment.

On the jester of faithfulness, I would like to thank my mother Teacher Menen Shewandagne for the value she added on me, shaping me from childhood to a mind-set of achievement and her untouchable great love.

Finally, the collaboration, advice and support of the whole family and friends is so very much helpful being decisive, revitalizing my energy and boosting me up in making the study real to the final point. This thesis is dedicated to my family and in simple words, big time respect.

Table of Contents

2.2 Basics of Mould Design

3. Method and Results

List of Figures:

List of Tables:

Glossary:

PP: *Polypropylene*

PE: *Polyethylene*

LDPE: *Low density Polyethylene*

HDPE: *High density Polyethylene*

PVC: *Polyvinyl chloride*

PS: *Polystyrenen*

PC: *Polycarbonate*

PA: *Polyamide*

PPO: *Polyphenylene*

ABS: *Acrylonitrile-butadiene-Styrene*

PMMA: *Polymethylmetacrylate*

T g: *Glass transition Temperature*

ε: *Strain*

δ: *Stress*

IM: *Injection Moulding*

P |L: *Parting Line*

Re: *Reynolds Number*

V: *Velocity of Coolant*

ρ : *Density of Coolant*

µ: *Absolute Viscosity*

ν: *Kinematic Viscosity*

D: *Diameter of Channel*

Q: *Volume Flow Rate*

***V*:** *Screw Speed*

***r*:** *Screw Radius*

***A_p*:** *Projected Area*

***P_o*:** *Operating Injection Pressure*

***F_C*:** *Machine Clamp Force*

***F_{CC}*:** *Calculated Clamp Force*

EDM: *Electro Discharging Machine*

1. Introduction

1.1 Background

Currently, plastic products are taking vital part on our day to day life, in the past few years they been playing part in different applications of our daily life such as housewares, packaging, electronics, electrical equipment, cameras, films, automotive, farming and aircraft components, furniture, closing and housing.

For producing the various products of plastics for daily life, different method of productions are involved, most commonly these plastic products are manufactured using injection moulding, thermoforming, blowing and extruding. Considering the most common methods for the production of plastic products at percentage, most of them are done using injection moulding. Hence, injection moulding is the most common and economic method for the production of plastic products.

On the study of the injection moulding process the most important point that lies under is the mould. The mould, which is the most important component part, that gives the product the shape required and designing the part product associated with the mould, hence worth study.

Back in time before the involvement of product, mould designing and appearance of mould designers, artisans and die makers were taking part. After the second world war when plastic technologies was beginning this artisans were engaged on mould designing but as time pass by and there emerged a demand for increased verities of plastic products designing with different moulding parameters and high specialization seeks enabling the level of mould makers, mould and product designing on a specific profession.

After the emerging of product designers and mould designing as a specific field in the plastic technologies enabled the plastic field to the next level, covering the part in plastic product manufacturing for the injection moulding as the construction and function of the mould by making the part designing and assembly drawing of every mould part with detail and accurate dimensions and tolerances for the mould maker to manufacture the mould for final production.

In this era of technology and science it is almost impossible imagining daily life without plastics application that varying from common place domestic articles to sophisticated scientific and medical instruments. It makes peculiar and worth studding plastics and product designing due to that they have properties not available in any other materials, provided that they have some limitations they offer properties that interest and keep an eye for designers this days, as mainly speaking, they offer advantages such as lightness, resilience, resistance to corrosion, colour, fastness, transparency and easy of processing. Having all this facts at hand, these plastics are processed with different plastic formation methods such as injection moulding mainly in industries. Hence, before processing them it worth understanding of selecting the basic plastic property, material type and basics of mould designing, as of for injection moulding product design, all lies at the heart and state of the art of mould and product designing that this study focuses as main plastic production line.

This days mould and product designing involves the use of computer designing software's such as AutoCAD, Solid Edge and SolidWorks for the part designing and assembly software for the aggregate designing of the mould assembly making the mould manufacturing precise and accurate for the injection moulding manufacturing process of plastics and enabling wide varieties of tolerance with production varieties.

1.2 Objectives

As an objective of this thesis project, after covering the literature of basic plastic material analysis and basics of mould designing theories for injection moulding, the Manual Juice Maker product part design, simulation and analysis is done based on researches, functionality, consistency and general peculiarity keeping its own identities as a plastic product part design and the following objectives carried out:

- ✓ A plastic product of a Manual Juice Maker is designed using the software SolidWorks.

  The part designing incorporates the designing of the Dom which is the squeezing mechanism with its own peculiarity and identity for maximum squeezing techniques is covered.

  After the squeezing mechanism is achieved based on the research study of scratch sample fruit and its needed characteristics for maximum squeezing then the jar is designed along with the Dom based on its easy of designing features and innovative geometric, elegant shape appearance.

  At the final stage of the part design, after the desired characteristics and features of the Dom and the jar container achieved and tested for designing consistency and feasibility of manufacturing, they incorporated together as a single part design to achieve the final part designing of the Manual Juice Maker.

- ✓ The polymer melt flow simulation and analysis is carried out for the Manual Juice Maker after converting it in to simulation able design entities. Various melt flow trials is carried out on using the simulation and analysis software Autodesk Moldflow Synergy and at the final stage an optimum values for the fill time, pressure distribution and clamp force is achieved based on the initial analysis trials and melt flow parameter assumptions made at the initial stages.

- ✓ Optimum Simulation analysis are obtained, compared and contrasted with the different trials made and the analysis assumptions carried out initially. Finally, values obtained, optimized and forecasted for real life injection moulding manufacturing on the ground level of the Manual Juice Maker.

2.1 Literature Review

2.1.1 General Property and Formation of Plastics

2.1.1.1 Definition and Characteristics

At the very beginning, plastics are assumed to be evolved around 1860, on the trial for material substitution of natural ivory. On literally understanding them, the word plastics comes from the Greek word 'Plasticos', which means to be able to be shaped or moulded by heat. When looking deeper and understand them in a professional and technical way, plastics are generally defined as any material consisting of very large molecules that are characterized by light weight, high corrosion resistance, high strength to weight ratios and low melting point. [1]

Plastics are made from different organic materials such as fossil fuels, petroleum, natural gas, coal and some plant materials, depending on the purpose they are designed in different compositions and mainly categorized as thermoplastics and thermosetting plastics. Plastics are characterized by their ability as a material that can be reshaped and changed in to a desired product using different industrial plastic production methods, such as they can be remoulded, extruded, piped and drawn to extrusion by applying heat and pressure. These plastics are formed by polymerization process through industrial production process and consist of mainly synthetic resins, which is the polymer that constitute in their structure as chained monomers. [2]

2.1.1.2 Properties

Organic properties of the polymer generally define the properties of the plastic, since the polymer is the basic constituent ingredient in the plastic. Most plastic material show unique properties when compared to other materials due to this chemical structure of the polymer resulting from the bonded monomers. On understanding and further elaboration of the polymer structures, it is a composition of two different base structures as physical or chemical mixtures that are polymeric blends and block copolymers. This can be amorphous or semi crystalline as a result of the molecular structure given by the thermoplastic component. [3]

So, these structures that are given as elaboration and illustration of the plastic structural bonding underneath principle all together the additional substances bonded to the carbon atom of the monomer accounts for the general principles of the resulting plastic properties. As a result of the discussed factors, plastic in general exhibits different physical and chemical properties. [3]

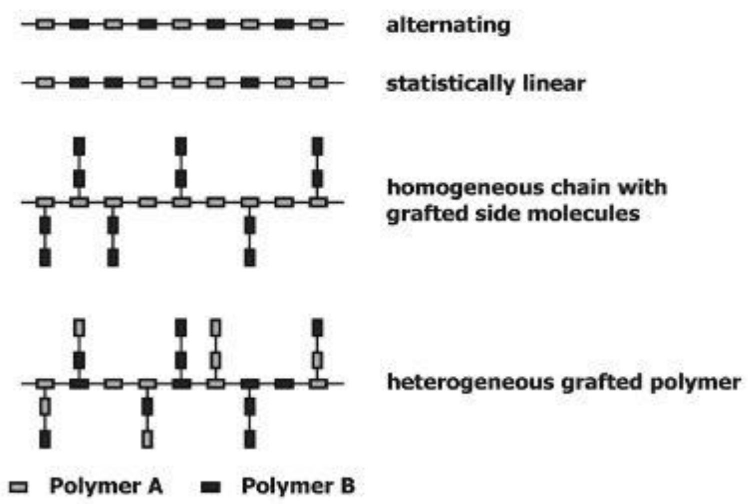


Figure 1: *Schematic builds up of copolymers [3]*

Some of the most common physical properties of plastics are mentioned below:

- Transparency
- Flexibility
- elasticity
- permeability
- water resistivity
- electrical resistance
- specific gravity
- softness when hot

Whereas, some of the chemical properties of plastics include:

- Solubility
- Chemical resistance
- Thermal stability
- Reactivity with water
- Flammability
- Heat of combustion [3]

2.1.1.3 Formation:

In considering formation and innovation of plastics, the main constituent is polymers and hence plastics are most part of the polymer. Polymers are basic ingredient of plant and animals with their chemical structure consist of very long chain like molecules. In turn the polymer consists of a group of monomers; the monomers are small molecules consisting of carbon and other substances and are found mainly in plants, oils and natural gas. On mentioning some of the natural materials with this structure includes silk, shellac, bitumen, rubber and cellulose. Before the innovation of plastics, a polymeric material called parkesine was first developed about the 19^{th} century. After this an important breakthrough in the development was successfully made with developing celluloid. During the development of synthetic materials, early as 20^{th} century a big jump taken on developing phenol formaldehyde and then after by the second world war materials such as nylon, polyethylene and acrylic appeared paving the way for plastic innovations. [4]

Before coming to the point of formation of plastics, the basics lies on the polymeric material and on elaborating it, the formation consists of by joining together thousands of small molecular units known as monomers by the process called polymerization. As mentioning some examples the polymer polypropylene is formed from the monomer propylene and as forth for the other polymers. Hence, when coming to the point of plastics it is related to polymers bearing in mind that the polymer is the pure material. Due to the fact that the pure polymer material is not applicable in its own the innovation of plastics came along with adding chemicals to the polymer called additives that give different features for application and hence plastics are formed in this form. Elaborating this formation, plastics are produced by joining two or more monomers to produce a polymer, during this chemical reaction addition or condensation process carried on for the formation of these plastics. [1] [5]

Then after the formation of these plastics to be applicable, there are many chemicals called additives that are added to give the plastic specific chemical and physical properties for application that gives advantages over the pure polymer. For example the additives, antistatic agent have a special feature of attracting moisture from the air to the plastic surface that enables for improved surface conductivity of plastics, and likewise, there are many additives that are added to plastics such as coupling agents, Fillers, Flame Retardants, Lubricants, Pigments, Plasticizers, Reinforcements, and Stabilizers. And there application and advantage for adding physical and chemical properties for the plastics is revised on the table below. [5]

Table 1: *Additives and their improved properties of plastics [5]*

Additives	**Purpose of the main additives in plastics**
Antistatic agent	Surface conductivity of plastics
Coupling Agent	Bonding of plastics to inorganic materials
Fillers	Mechanical properties of plastics
Flame Retardants	Reduces combustion
Lubricants	Reduces viscosity and improves forming characteristics
Pigments	Colors in plastics
Plasticizers	Properties and forming characteristics of plastics
Reinforcement	Strength and stiffness
Stabilizers	Prevents deterioration of plastics due to environmental factor and ultra-violet radiation

2.1.2 Plastic Classifications

Based on the macromolecular structure and temperature dependent, physical structure plastics are classified under wide categories as thermoplastics, thermosetting and elastomers. This categorization is also based on the chemical structure and reaction for formation and the properties of the resulting polymer constituent molecule.

2.1.2.1 Elastomers

This type of plastics is characterized by molecular chains that are wide chained cross-linked. This characteristics enables the material to have high stability but still elastically malleable and cannot be reshaped after melting. In addition their chemical structures enables them that by applying load the chains disentangled and upon removal of load restores and relax to original positions. [6]

2.1.2.2 Thermoplastics

Thermoplastics are the type of plastics formed by a reaction that resulted long chain like molecules in a randomly distributed long strand. Unlike elastomers and thermosetting, this type of plastics have molecular chains that are not cross-linked. Due to this and the fact that their molecular chains are held by weak van der Waals force they are thermo formable, meaning that they are weld able and mainly shaped in to different forms by applying heat.

Having said all this about their molecular structure, they are easy to process and applicable for plastic production since upon heating the intermolecular force weakened resulting soft and flexible form for processing and at high temperature they get to a viscous melt besides upon cooling it solidifies again in a repeated manner, therefore hear lies the important point for processing methods.

Regarding the molecular structure of thermoplastics that plays a role in the formation and processing methods have further categories due to the nature of the chain structure, hence the structure could be a more regular and closely packed crystalline or loosely oriented amorphous structure. It is wise to bear in mind that complete crystalline structure is impossible due to the fact of thermal history however some plastics exhibit higher degree of crystalline to a considerable limit. Hence, the crystalline structure is subdivide in to partially and semi- crystalline structure. To sum up, these wide categories of crystalline and amorphous structures have their own physical and chemical nature. Based on this a designer can chose which type of plastics properties is desired for the product beforehand. The table below revises the nature and property of these categories and listed below. [5]

Table 2: *Amorphous and crystalline categories of thermoplastics [5]*

Amorphes Structure	**Crystalline Structure**
Broad softening Range : due to weak secondary bond results broad rate	Sharp melting point: due to regular closed pack structure bonds broken down at the same time
Transparent: looser structure transmits light	Opaque/ transparent: due to difference in refractive indices between phases
Low shrinkage : on solidification the random arrangement of molecules produce little volume change	High shrinkage: on solidification the polymer take up closely packed high aligned structure, significant volume change
Low chemical resistance : open and random chemical structure enables deep penetration of chemicals and braking of secondary bonds	High chemical resistance : the highly packed and arranged chemical structures prevents external chemical reaction within the bonds
Poor fatigue and wear resistance: the random structure results on poor qualities	Good fatigue and wear resistance: due to the closed packing and uniform resistance enables good qualities

As been summarized on the above table, the thermoplastics have their own categories of amorphous and crystalline with the physical and chemical properties mentioned. Based on this it is to own designers demand to specify the type of plastic nature as of the demand for production line. Moreover these broad categories encompass different types of plastics most commonly using for plastic processing. Hence, it is worthwhile mentioning these types of plastics on their respective categories and they are summarized by the table below on their respective categories example. [5]

Table 3: *Examples of amorphous and crystalline thermoplastic* [5]

Amorphous	**Crystalline**
Polyvinyl Chloride PVC	Polyethylene PE
Polystyrene PS	Polypropylene PP
Polycarbonate PC	Polyamide PA
Acrylic PMMA	Acetal POM
Acrylonitrile-butadiene-Styrene ABS	Polyester PETP,PBTP
Polyphenylene PPO	Fluorocarbons PTFE,PFA,FEP ,ETFE

2.1.2.3 Thermosetting Plastics

Thermosetting are plastics that exhibit a different chemical bounding structure than that of thermoplastics. Mainly their long chained molecular structure undergoes a secondary reaction that could be up on heating and /or pressure so that their structure exhibits crosslinking with interlinking to form a strong bond. Due to the manifestation of this strong bond the molecular structure cannot be softened back again upon cooling hence with exposure of excessive heat and pressure the structure deteriorates than giving the original form. [5]

2.1.3 Selection of Plastic Materials and Properties

2.1.3.1 Selection of Plastic Materials:

On the virtue of the point of mould designing, there is a point a plastic designer must consider before starting any mould and product design, meaning, first and most valuable point is that to decide which plastic material is best for the particular application of the product needed, a complete understanding and knowledge of plastic material properties, behaviour, flow properties is needed.

As of the fact that plastics are essential in our day to day life and they are substituting most materials due to the fact that they have good thermal, mechanical and electrical properties, there selection should be studied with proper material selection.

Material selection is the science where we lay the ground for choosing the type of plastic material for the product we needed to manufacture. It requires and applies the knowledge of the general behaviour of plastics with detail investigation of special characteristics of type of plastic product required.

After generally decide and define the purpose and function of the plastic product we need, the next steep takes selecting the material for this product that meets the required characteristics from the range of plastic material available. Hence, the main point here in material selection lies on understanding the most important characteristics of the engineering plastic. Based on this perspective, this part deals with the underground point of material selection, which understands the material characteristics and properties. There are most important characteristics and properties need considering during material selection and among them the mechanical properties of plastic is the most important in terms of mould and product designing, discussed as follow. [7]

2.1.3.2 *Mechanical Properties:*

Mechanical properties of plastic mainly includes the properties such as, strength, stiffness, specific strength and stiffness fatigue, toughness and deals with analysing the effect of temperatures on these properties.

Stiffness and strength are the main mechanical properties of plastics, and this mechanical property indicates characteristics both for viscous liquid and elastic solids. This means that for viscoelastic materials, that are thermoplastics. Since thermoplastics are viscoelastic, their properties of the viscoelastic materials are dependent on some specific characteristics that we study here in material selection part mainly:

- Time
- Temperature and
- Strain Rate

On describing this mechanical properties, most commonly a stress – strain test is used which will provide the characteristics and properties for initial designing selection. These properties observed from the resulting stress– strain graph and the properties are analysed by observing the graph. Here, also need to bear in mind that there is effect of temperature on the strain –stress relationship and both are shown below in graph and elaborated.

On observing the stress–strain for plastics graph, which are quite similar to a metal, there is an elastic region at low– strain limit and a linear relationship with the stress but when the strain gets higher there gets a region of permanent deformation and the strain has nonlinear relationship with the stress at a higher limit it riches the tensile yield point.

On designing a material and selecting the appropriate characteristics, it is usually considered a strain limit of 1% and a lower values of about 0.5% for more brittle thermoplastics, and on considering thermosets plastics a strain limit of 0.2 - 0.3 % would be appropriate. [5]

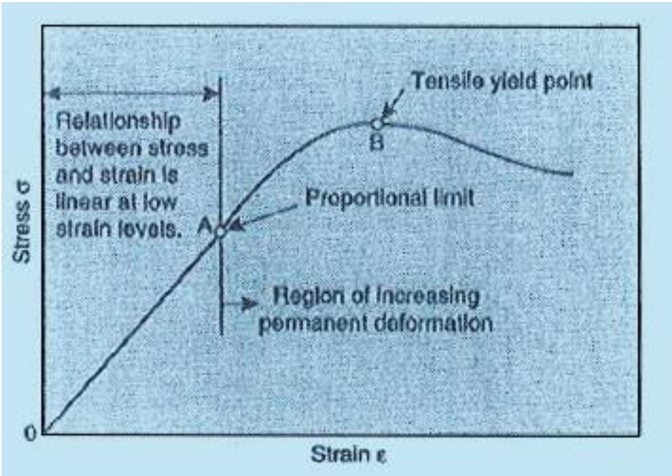


Figure 2: *Stress -Strain graph for plastics [5]*

The other important factor need to be considered is the effect of temperature on the stress strain relationship for the material. As the temperature increased within the given stress the material deforms more and the material become more flexible. [5]

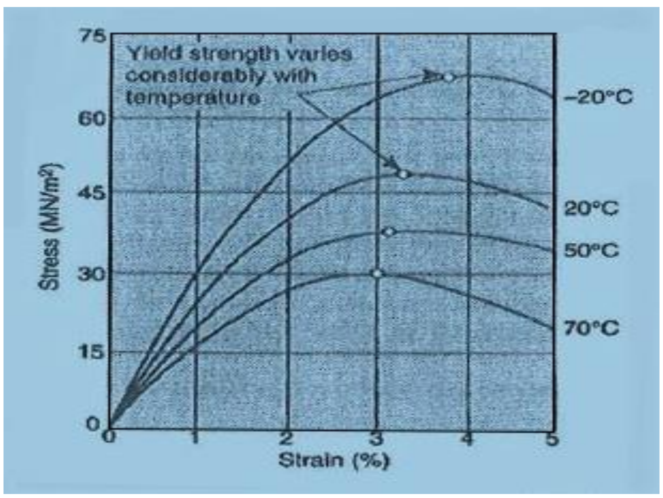


Figure 3: *Effect of material temperature on Stress-Strain behaviour of plastics [5]*

And finalizing the factors that affect the mechanical behaviour of plastics, as mentioned above, there is one point worth mentioning that has an effect on the stress- strain relationship that is the strain rate. It works in a relationship that if a thermoplastic is subjected to a rapid strain rate it influences in such a way that the material become stiffer than the same material where the strain is equal but applied on a lower rate. Moreover with the attached reference PDF file here acquaints the topic more with the properties and performance of the material under different parameters stated about design criteria like: stress, stiffens, strain, tensile strength and so on. [8]

On the other point of view of designing the material and selecting its characteristics, selecting the stiffness and the strength lays the foundation for any selection criteria. The Young's Module of elasticity determines the stiffness and it is the main criteria that can be observed from the slop of the stress –strain curve. And also on material selection for strength, their strength desirability factor that can be calculated from structural efficiency data of range of materials. [5]

2.1.3.3 Thermal Properties

Plastic materials need to be selected regarding their respective material application, above all that described above regarding their properties and selection purposes, there are some major special material properties that they possess and worth discussing. In considering some of the special properties of plastic such as thermal properties, optical properties and electrical properties are among the main but thermal and mechanical properties are mainly related to plastic product designing and mould designing, discussed as below.

Thermal properties are the main material properties of plastics that have effect on the mechanical property of the plastic. As a result of their molecular structure plastics is temperature dependent, hence their molecular structure results in plastics molecule in more molecular motion up on heating that causes them to be more flexible and on cooling process they have restricted molecular motion resulting them to be ridged and brittle.

Hence, temperature has been a major factor on designing any plastic material and determines properties. On general there is a temperature at which all plastic materials exhibit variation in their physical and mechanical properties, that is glass transition temperature T_g. This temperature serves as a border line between this properties shifting, most plastics above there glass temperatures are flexible and soft but when they cooled below their glass transition temperature they become stiffer and brittle. Also, it is important to remember that some crystalline plastic materials exhibit properties of rigidity and stiffens above their glass transition temperature due to strong packing molecular force they possess.

Other important worth mentioning material properties of plastic that is temperature dependent is their thermal conductivity and thermal coefficient of expansion. On general speaking plastic materials have low thermal conductivity and high coefficient of expansion. [9]

2.1.4 Polypropylene

The plastic material for this product design developed in the thesis is chosen as polypropylene. The material is chosen considering various properties and advantage of polypropylene for the application of the part design. Peculiarly, polypropylene exhibits great properties suitable and desired for the part design production like light weight, strong, high heat resistance and stiffness. Moreover polypropylene can be fabricated in various production methods and mostly using injection moulding process for this part design production. [10]

- ***Formation of Polypropylene***

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

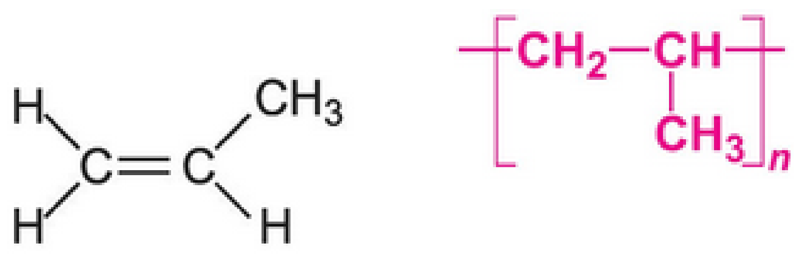


Figure 4: *Chemical structure of propene and the polymer* [11]

Propene is largely produced from gas oil, ethane, naphtha and propane. Hence, upon the proper chemical reaction of addition polymerization, propene forms poly propylene .In basic terms there are two industrially employed manufacturing of polypropylene using a well-known catalyst called Ziegler Natta Catalyst which is a mixture of aluminium chloride and aluminium alkyl**,** they are the Bulk Process and the Gas phase process. [11]

✓ *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. [11]

➢ *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

➢ *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 [10]

➢ ***Properties of polypropylene***:

Most common typical properties of the plastic material polypropylene are revised on the table below, based on the reference documentation.

Table 4: *Typical properties of Polypropylene* [12]

TYPICAL PROPERTIES OF POLYPROPYLENE (PP)

ISO or UL 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	**Water absorption** (%)	0.01-0.03	0.01-0.09	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 break** (%)	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	**Flexural modulus** (MPa)	117.2-172.3	144.8-689.5	89.6-137.9	144.8-661.9
ISO180	**Notched Izod impact strength** (J/m)	21-75	32-641	59-747	32-214
ASTM D785	**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 expansion** (10^{-4} m/m-°C)	1.4-1.8	0.27-0.90	1.08-1.80	0.36-1.08
ISO75	**Deflection temperature** (°C)				
	At 1.80 MPa	49-60	54-166	49-60	47-138
	At 0.45 MPa	107-121	104-149	85-104	77-152
UL 94	**UL flammability rating*** **	HB	HB	HB	HB

*Homopolymer polypropylene. **Copolymer polypropylene. ***V-2, V-1, V-0, V-5 grades available.

2.2 Basics of Mould Design

As of this thesis focuses on injection moulding design and simulation of a plastic product and on this chapter it focuses on the basics mould designing for injection moulding. To begin with, there lies a basic concept what the mould is itself. The mould is the common term in injection moulding process; it is an aggregate of different components in consistence and mainly consists of two primary components called the injector mould and the ejection mould. Basically, a mould is designed in such a way that the moulded part remains on the ejector side of the mould plate when it opens and hence, the sprue and runner together with the part product taken from the injection side plate.

When see what really goes in the mould and its principle of operation during the injection process is that, the molten plastic called reins in an amount of a shot, meaning the proper dosage to fill the sprue runner and cavity of the mould, enters the mould through from the nozzle by the connected sprue and running through the runners in to the gate. Hence, after mentioning the basic principles underlying in the mould as introduction, here is covered the basics of the mould components for further elaboration. [13]

2.2.1 Cavity and Core

The cavity and core are the most important part of the mould. The mould cavity is usually the concave hollow shape part that forms the outer part of the product to be moulded and hence it is the negative part of the shape of the desired product. Usually for large products it is built for one cavity space only but for conditions for economy and small product parts, the cavity can be built as many desire numbers of pairs with multiple cavities to increase production. But at the same time with increasing the cavity the filling time increases as it will need more time to fill all the cavities than a single cavity, on the other hand it also requires a larger machine with more clamping capacity. The cavities are usually placed on the injection half of the mould which has direct contact to the injection point. On considering the manufacturing of cavities different methods can be employed. This shape of cavities usually created applying chip removing machine tools and most commonly using modern machining techniques such as electric discharge machining called EDM and in a modern ways CNC machine can be employed. Since the cavity side is in contact to the injection point of the machine, there is high injection pressure involved at the point and hence the cavity mould has to resist, due to this and other factors the cavity is made of durable and high resistive materials, most commonly steel, aluminium or metal alloys. [14]

On the other hand the cavity has a matching which has a shape of convex called the core. The core is mounted on the other half of the mould, which is connected to moving plate of the machine for further ejection mechanism. [14]

2.2.2 Parting Line

In any mould there are two separate halves, specifically the cavity in one side and the core on the other side. In practical sense there is a separation between these plates and there exist a separation plane that is called the parting line, PL. This is the basic part of the mould when the mould haves closed ensures that it shuts off tightly when the mould is clamped and prevents from flashing out of resin.

Manufacturing is easy for a mould with parting line on one plane and as a principle the parting line is always at the widest circumference of the product that is an important feature that makes the ejection of the product from the mould possible. In some mould designs the parting line not necessarily is in a single line, in some cases it might be offset or to be inclined at an angle. [15]

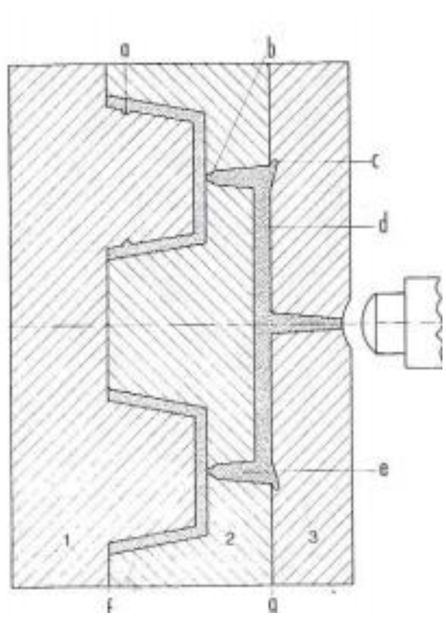


Figure 5: *Demonstration of core, cavity and parting line. [16]*

Labelling: 1. *Core, movable mould half*

2. Cavity, stationary mould half

3. Stationery Mould plate

a. Under cut in core   b. Gate c. Under cut   d. Runner

e. Gate sprue   f. Parting line, separating core and cavity

2.2.3 Sprue

There is a mechanism by which the reins that comes from the barrel through the nozzle transfers to the runners, at the contact point of the nozzle there is a sprue mechanism that enables passage to the runners and getting in to the gates that finally fills the cavity. [15]

Generally speaking, the melt enters the mould via the sprue bushing system which sealed off the barrel and where the sprue is machined, this system enables a leak proof passage from the injection nozzle to the runner. Since, during injection there is high pressure when the melt enters the bushing system and hence, the bushing system must have a mechanical load bearing capacity, usually in most applications the sprue bush system which is applicable for this particular product design is sprue gate system which is replaceable and must have the following properties for a better application. [16]

Properties

- ✓ It is made of hardened steel for the purpose of bearing high mechanical load, hence able it to have wear resistance.
- ✓ Must have a flange and rounded edge for flexural fatigue strength.
- ✓ As small as possible diameter as possible due to the reason that the sprue leaves a mark on the mould.
- ✓ Diameter of the nozzle must be 1.5 mm smaller than that of the sprue bushing and the nozzle must be aligned with the bushing system.
- ✓ At the point of contact surface between the nozzle tip and the bushing system in the horizontal plane, the diameter of the bushing system tip contact must be 1 mm greater than that of the nozzle and the sprue is tapered at angle between 1 and 4 degrees. [16]

2.2.4 Runners

Runners are channels through which the plastic melt that passed through the sprue during injection passes to the gates and fills the cavity. In a nut shell they connect the sprue via the gate to the cavities. As a basic principle, runners should have the capability for the plastic melt in the same condition and the same filling pressure with uniform filling time of all the cavities. Runners can be designed for two cavities or more but in all cases the above condition must be preserved for proper moulding system.

The underlying process going on the runners is that, when the plastic injected, the plasticized material enters the runners with a high velocity and then heat is rapidly removed from the material by the walls of the runner through heat transfer and forms a layer of skin on the surfaces of the runner. Hence, resulting the hot core in the centre to flow and serves as insulating layer for the plastic hot core melt passage, finally this hot fluid core fills the cavities. This hot core fluid is maintained in the runner until the part product in mould gets completely solidifies and accounts for compensating the volume shrinkage during solidification.

On designing a runner, the basic point need consideration is maximum material saving and minimum pressure loss. The next step is on designing the diminution of a runner and optimizing them, which is depend on according to the plastic material type being processed and the thickness of the moulded part material. As a general principle the cross-section of a runner is usually 1 mm larger than the maximum thickness of the moulded material in the cavity. But larger cross-section is advantageous for optimum filling and maintain adequate holding pressure, on the other hand this might have some side effect on material usage saving and increase cooling time. Runners play a great role for the moulding process; they have many functions and objectives regardless of their type and classifications. Some of the functions and objective of runners are listed below. [16]

2.2.4.1 Objective and Functions

- ✓ With minimum of heat and pressure loss they drive the plastic melt rapidly and in unrestricted way in to the cavity via the gate in shorter ways.
- ✓ Fills cavities via the gate at the same pressure and temperature
- ✓ Though larger cross sections are advantageous for optimum cavity filling and adequate holding pressure, since with large cross section there is a disadvantage of material wastage and larger cooling time, hence runners should be designed with small cross sections.
- ✓ Surface volume ratio must be kept as small as possible and applicable with the mould process.

On top of this, runners must be designed and optimized well to achieve economical and efficient mould process. There implication in the mould design is much more important and are affected and associated with many factors.

2.2.4.2 Designing Factors

- ➢ **Mould part geometry, process and plastic characteristics**

- ✓ Part volume
- ✓ Wall thickness
- ✓ Plastic material
- ✓ Length of flow path
- ✓ Resistance to flow
- ✓ Surface volume ratio

- ➢ **Mould process and material characteristics**

- ✓ Heat losses
- ✓ Losses from friction
- ✓ Cooling time
- ✓ Amount of scrap
- ✓ Cost of manufacturing
- ✓ Mould type [16]

2.2.4.3 Classification of runners

Once the above mentioned characteristics and objectives for a particular use of the runners are understood, the type of runners can be classified and choses after wards among the available types. On designing phase it is important to choose the runners for economical and optimum quality output of the mould process and part, once this is known the design can proceed. Based on the temperature control there are three types of runners available and each has their quality, advantage and disadvantage.

- Standard Runner System
- Hot Runner System
- Cold runner system

2.2.4.3.1 Standard Runner System

This type of runner system is rarely used and applicable for thermoplastics and thermosets plastics. It has a special feature and character that it is directly machined in to the mould plates, along the parting line and the temperature of the standard runner is the same as the temperature of the mould. During injection, the melt passes through the sprue and gets in the runner, and it freezes after filling the cavities accomplished. Hence, the plastic freezes in this type of runner system and need to be demoded along the moulded part after ejection. Usually the material in the runner can be recycled if it is a thermoplastic and limitation in recycling in case of thermosetting plastics and associated wastage of material. [17]

2.2.4.3.2 Hot Runner System

Hot runner system is one of the most sophisticated components of an injection mould; it consists of a sprue bush system, a runner and a gate or nozzle. This system employs hot runners and a great technology for the injection mould process resulting in precise, quality and cost profitability. The hot runners in this system has a complicated system, they are built in such a way that they maintain their temperature hot for the melt to flow and the melt remains hot inside them while the mould gets cooled and solidifies. Hence, a heat barrier isolates it from the mould to achieve this quality, also enabling the temperature of the hot runner lies in the melting range of the thermoplastic melt. As a result the outside boundaries of the runner solidify and cooled while the melt flows only in the proximity of the runners and filling the cavities. [18]

The hot runner system, as shown the one in the figure, includes a heat fold and nozzles along the runner which has application for the system to distribute the plastic melt entering the mould to various nozzles that meters precisely to the injection points in the cavities.

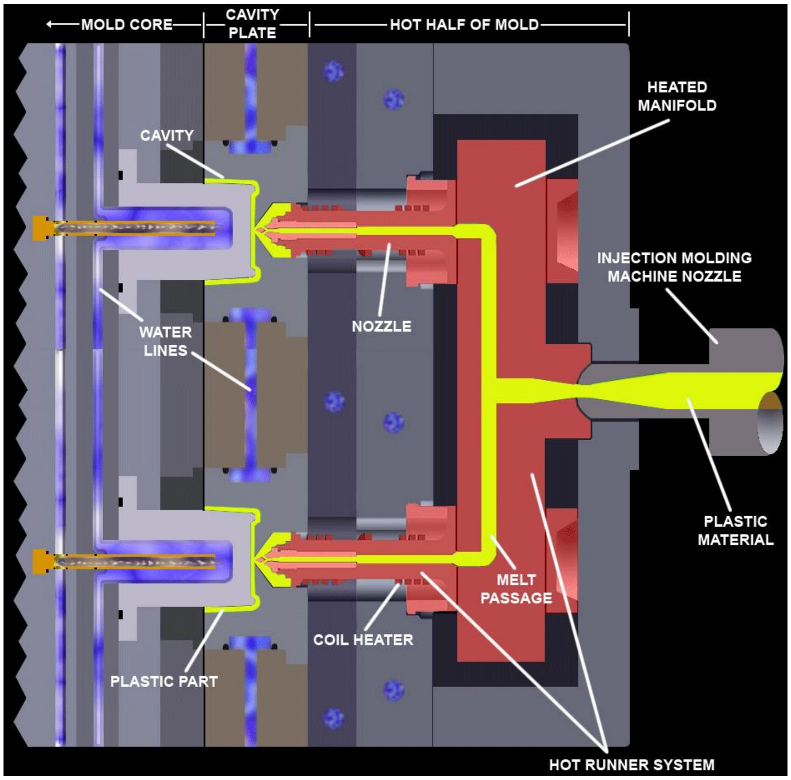


Figure 6: *Hot runner system \ nozzle, sprue bush system runner and gate*

Heated Manifold barrier [18]

There are two types hot runner systems, externally heated hot runner system and internally heated hot runner system. On the former one, the melt flows in the solid manifold and nozzles whereas on the later one the melt directly flows inside the runner with an internal built up heaters. Due to the fact that they offer more efficient moulding methods, the use for hot runners are increasing this days in the injection moulding process. Regarding the sophistication of today's injection moulding, they are a break through and almost impossible achieving a detail and precise product moulding. They exhibit several advantages over the other runner systems; some of the advantages are listed below. [18]

Advantages of hot runner system

- ✓ Uniform wall thickness
- ✓ No loss of melt, less energy and work input
- ✓ Easier and fully automatic operation
- ✓ Since, melt transfers to the cavities at the optimum sites it results in great qualities
- ✓ Reduction or elimination of nit line
- ✓ Applicable for high cavitation and large moulding
- ✓ Applicable for micro moulding

To sum up, in addition to the above mentioned advantages, hot runners greatly help and makes advantageous their system in a way that there is less wastage of melt material and due to the fact that the melt do not freezes in the runner, they reduce the cooling time and hence, less cycle time. However they are more expensive to build in the mould system and have the disadvantage of high coast. Moreover there is a risk of decomposition and stoppage of plastic materials with low thermal resistance and it is usually difficulty with thermal isolation from the hot runner manifold. [19]

2.2.4.3.3 Cold Runner Systems

Cold runner systems are channels for the mould for the flow of the melt plastic from the injection nozzle through the runners at a lower temperature to the cavities via the gate. As mainly hot runners are for thermoplastic injection moulding, cold runners are used for thermosetting and elastomers that are vary reactive plastic materials. In cold runner system the runners are kept at lower temperature about 80-120 degree Celsius for the purpose that the material not to react prematurely in the runner. In this type of runner systems, every time the mould material ejects, the material in the runners removed also, hence, resulting a material wastage and increased cycle time due to the reason that the material takes time to freeze in the runner before removal.

Cold runner systems are applicable mainly for elastomers and rubbers, due to different aspects they have drawn backs and limitations as listed below.

Limitations of cold runner systems

- ✓ They have very high pressure consumptions, and as a result it makes the design very expensive.
- ✓ Slightest temperature difference causes very large viscosity difference and hence, non-uniform plastic melt.
- ✓ Difficult filling the cavities at the same time and same condition
- ✓ Larger cycle time and irregular fill time. [19]

2.2.5 Gates

Before the plastic freezes and cools so that it cannot flow the plastic after plasticized must fill each cavities with equal pressure and fill time, this small openings and usually the thinnest point of the whole system that connects the runners to the cavities and allows the melt to enter are called gates. [15]

Gates are the transition zone between the runner system and the cavities, they must ensure filling of the cavities with uniform flow and they must avoid plastics jet in to the cavities. On the virtue of this characteristic, gates are best connected to the runners in two main forms as centric gate and eccentric gate. The former one has semi-circular and rectangular cross sections with defined characteristics of centric positioning and small surface to volume ratio for circular positioning that reduce heat loss and friction, but separation of this gate is more difficult and may promote jetting. Whereas the later one, eccentric gating has circular or rectangular cross section, with eccentric positioning that facilitate removal of the mould easy. Generally, the gates have circular, semi-circular and rectangular cross section. Whereas the most common one is the rectangular gate and the semi-circular one ensures easy removal of mould from the mould parts. Hence, in all kind of gate system a careful selection, location and positioning of gates to the moulded part and to the runner system results on net finished of the mould part and optimizing of the position is required before selecting the gate position. [16]

2.2.5.1 Designing of Gates

Gate design is very important and lays the foundation on mould designing besides the runner and the bushing system, the location of the gate should be designed carefully and must permit the flow of the melt evenly, same fill time with all the cavities. On designing a gate the following general points should be considered and applied:

- ✓ The gate must locate at the thickest section
- ✓ Modifying the gate position, location and type to avoid melt jetting in the cavities
- ✓ It must be as small as possible that the material is heated but not damaged by shear
- ✓ Uniform and even filling by balancing the gate and optimizing
- ✓ Place for the easy of designing and de moulding from the mould part
- ✓ Minimize entrapped air to avoid void burn marks
- ✓ Avoid and consider areas subjected to mechanical stress. [20]

The above mentioned points are preliminary, general perspectives worth considering on designing of a gate system. Furthermore a detail analysis, characteristics and considering various issue cover ups is needed for a more professional mould designing finished result. [20]

2.2.5.1.1 Gate Location on Flow Behaviour of Melt

For proper melt flow, the location of the gate should be considered from three main point of view aspects of pressure, orientation and weld line, etc. These are some of the main factors considered for the proper melt flow right positioning of the gate:

- Pressure
- Orientation
- Weld line

Pressure:

It is needed adequate pressure for complete filling of the part; with enough pressure packing would be perfect and a better mould result. Enough pressure will also prevents sink marks, avoid voids forming and ensures proper melt flow. Hence, the proper positioning of the gate to ensure this quality from pressure point of view will be at the place where located with thickest section of the part

Orientation:

The gate should be located in a way to minimize molecular orientation. Molecular orientation is a significant effect especially in thin sections, where the molecules align themselves in one direction which is the flow of the melt. This high degree of orientation results in uniaxial strength, meaning one direction resistance of load and stress. To avoid this gate should be located in a way such that the melt once entered the cavity diverted by the walls of the cavity or other surfaces such as ejector pins. Hence, it exhibits proper melt flow avoiding only uniaxial loading and results better load bearing capacity.

Weld Line:

When considering the weld line, the gets should be located in a position close to the weld line, by doing so the design is for granted to maintain high melt temperature that is desired to a strong weld line. Moreover it is wise to locate the gates in order to minimize the number and length of weld line; all this ensures the proper function and appearance of the mould part perfection. [21]

2.2.5.1.2 Size and Location of Gates

The gate can be designed in various size and configurations, in all cases the gate is the narrowest point in gate system. It should be noted that when the melt flows in the gate system it encounters a considerable resistance to flow, and as a result the injection pressure drops to a considerable rate and the melt temperature rose. This phenomenon is the melt characteristics in any gate system. As a result, basic parameters for proper melt flow and uniformities are fulfilled and they are:

- ✓ The melt entering the cavity become more fluid and forms the cavity part better way
- ✓ The gate opens longer for holding up pressure and the surrounding metal is heated up for better melt flow

On top of this, after the basic assumptions that the above parameters are fulfilled and based on, we can consider designing the gate size and determining the gate locations. Hence, the designing factor depends on the characteristics of the mould part, the moulding material and other general factors which are listed in detail bellow: [16]

2.2.5.1.3 Factors Affecting Gate Design and Locations:

Moulding part:

- Geometry and wall thickness
- Direction of mechanical loading
- Quality demand in dimensions, mechanics
- Flow length to wall thickness ratio

Moulding Material:

- Viscosity and temperature
- Flow characteristics
- Fillers
- Shrinkage

General Factors:

- Distortion
- Knit Line
- Ease of mould removal
- Separation from moulding
- Coast [16]

2.2.5.2 Classification of Gates

To its significance, the gate is categorized in to different category types based on various variants, properties and characteristics with its application specifications. Mainly characteristics of the mould part to be produced like the size and diameter with the thickness wall impact the type of gate to use. Also, the filling parameters where the gate meets the part, weather laterally, in the middle or edge filling classifies the gate type and the shape with the size of the gate and method to vary accordingly. Moreover the filling and solidification process of the melt in the cavities also differs in different types of gate, finally as main factor, the shape of the moulded part as being circular, cylindrical or planar gives a way to choose the type of gate and the gate removal post processing method like manually or automatically classifies the gate type.

Based on the above mentioned properties and characteristics gates are classified. The main type of gates is: sprue gate, edge gate, fan gate, disk gate, ring gate and tunnel gate or submarine gate. [20]

2.2.5.2.1 The Sprue Gate

This kind of gating system is common and recommended for parts with symmetrical filling of thick section single cavity. It is the simplest and old kind of gates with circular cross section and tapered, adjusted with the larger diameter towards the thick part of the mould part.

This type of gating is more effective for thick section parts because holding pressure is more effective and enabling rapid mould filling with low pressure loss. Moreover the adjustment of the sprue gate system should be placed in the thick section part of the mould so that holding pressure can remain effective during the entire time the mould part solidifies, also by this arrangement the volume contraction during cooling is compensated by additional material forced in to the cavity. This process also accounts to avoid the formation of voids or sink marks on the part.

When we consider dimensioning of the sprue gate, it is controlled by the diminution of the thickness of the mould part at the contact point being tapered towards the part and the diameter of the nozzle at the exit point. Hence, the sprue at the contact point has a diameter of 1 mm to twice the thickness of the part and at the nozzle it has a diameter of 1.5 mm more than that of the nozzle. Too easy of mould removal and optimum filling the gate is tapered towards the part at an angle of 1 to 4 degree being 2.5 degree the standard and small radius of curvature at the contact point of the part. [16]

To its disadvantage, the sprue gate leaves visible spot mark after post processing, to avoid this the sprue gate can be provided with turn around called overlap gate that reaches the moulded part from the inside at a point not noticeable, the sprue gate also has another variant called the curved sprue that permits lateral gating of the part to accomplish balanced position of moulded part which is commonly applicable for thermoplastic elastomers. [16]

2.2.5.2.2 The Edge Gate

This type of gate is used for multi-cavity two plate moulds, applicable for moulding parts with large surface and thin walls. The gate is located on the parting line and fills from top or bottom, this type of gate has additional distributing profile channel that the melt after the sprue enters this extended distributer channel which connects the cavity through a narrow land with the runner system. Thus the channel is filled with the melt before the material enters the cavity through the land. The edge gate due to its structure and arrangement with the way of filling has its own peculiarity and advantages for a specific mould part, they are:

- ✓ Parallel orientation across the whole width.
- ✓ Uniform shrinkage in the direction of flow and transverse.
- ✓ No inconvenient gate mark.

Regarding the diminution and setting of the edge gate, it is primarily dependent on the part thickness.

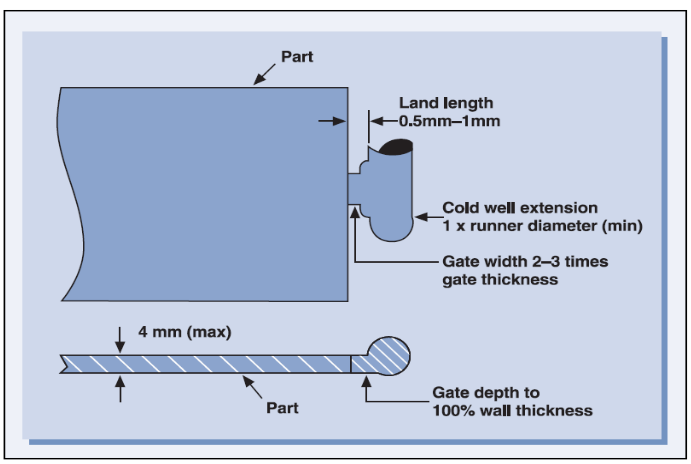


Figure 7: *The edge gate* [21]

Hence, a typical edge gate is 80% to 100% of the part thickness up to 4 or 3.5 mm and has a width of 1 to 12 mm and there is a final requirement that the gate land should not be more than 1 mm being 0.5 mm the optimum. [21]

2.2.5.2.3 Fan Gate

This type of gate is similar with edge gate; it is practically a wider edge gate with specialty and peculiarity for the application of mould parts for low stress moulding and advantages in detail perfection for dimensional details and stability. These characteristics are acquired due to the property and mode of operation that it enables slow injection without freeze off.

When overviewing its diminution and structure, almost similar to edge gate but with additional tapers in both width and thickness to secure constant cross sectional area, as a result of this structural features, this gate has the following advantage and characteristics:

- ✓ Constant melt Velocity.
- ✓ Same pressure across the entire width.
- ✓ Entire width is applicable and used for the flow. [16]

2.2.5.2.4 The Disk Gate

The disk gate is a disk like shape plane of circular or cone usually with 90 Degree taper, this gate has a special feature and application for uniform filling of cross section of cylindrical like moulding. It operates in such a way that distributes the melt uniformly onto larger diameter of the moulded part.

Due to its structure and mode of operation, the disk gate has the following advantages:

- ✓ Permits the moulding of cylindrical parts with undercuts in a simple mould without slides or split cavities.
- ✓ This type of gating enables that knit lines are eliminated.

Disk gate can be arranged in two different forms as conical disk gate and umbrella disk gate. The former has additional opening for core support and the latter can be arranged in two different systems as either directly or with land connected to the core.

Finally, the disk gate dimensioning is calibrated and designed in such a way that, the ratio between the length of the core and its diameter is smaller than 5/1. [22]

2.2.5.2.5 The Ring Gate

The ring gate is a system with special feature that is applicable when the core is needed to be supported at both ends, and employed for cylindrical parts for multi cavity mould.

Its mode of operation is that the melt passes through first into annular channel which is connected to the part by a land; first the channel gets filled with the melt and passes to the cavity with a land.

Its structure and operation has the following advantages:

- ✓ Feasibility of supporting the core at both ends.
- ✓ Utilized for cylindrical parts in multi cavity mould.

Regarding the dimensions and set up of the ring gate depends on the dimension of the moulded part, weight and type of plastic to be moulded and flow length. Generally, applicable for parts with equal wall thickness and the ratio of length over diameter of the core greater than 5/1 value.

To sum up, a design variation of ring gate can be found as internal and external ring gate, where the internal ring gate has a feature of two weld line, more expensive to machine and core support at both ends. Whereas the external ring gate material enters from one side forming weld line in opposite side of the runner where weld line is not transferred to the part. [16]

2.2.5.2.6 The Tunnel Gate (Submarine Gate)

This type of gating system is self-operating which can be operated automatically; it can be used for multi-cavity moulds of two plate for the manufacturing of small parts. In this case an angled tapered tunnel is machined connecting the end of the runner and the cavity below the parting line. Hence, a tunnel like hole is milled in to the cavity hole in an oblique angle.

The arrangement in the mould is in such a way that the part and the runner are placed in the movable mould half part where as the tunnel can be located either the fixed or movable mould half part where the gate is detached easily .

Depending on the shape of the tunnel hole, the tunnel gate is classified mainly in to two types; the first one is with a tip of pointed ends and works in such a way that the melt transfers to the cavity mould part in punctate. [21]

The second one is shaped like a truncated cone with elliptical cone, which is the most advantageous and the melt enters in the mould in the cavity in slower fashion with elliptical end, due to this has a feature that the melt freezes slowly and permits longer holding pressure.

Finally the arrangement of the tunnel gate is shown in the fig below and the diminution varies depending on the type of plastic melt with a standard and average gate size of 0.8 mm to 1.5 mm. [21]

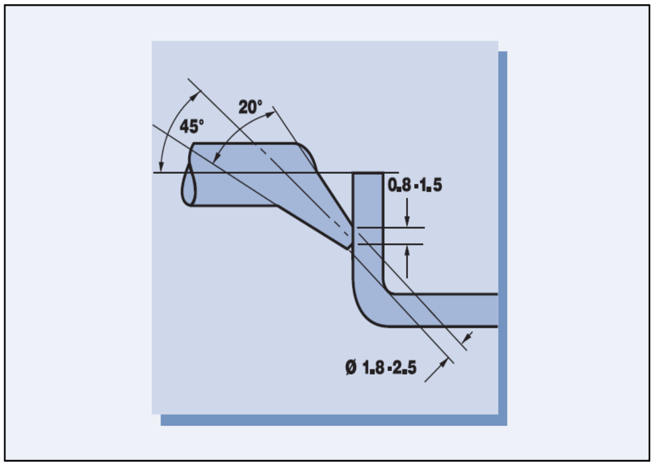


Figure 8: *The tunnel gate [21]*

2.2.6 Venting

This subject matter concerns with venting, it is needed for most mould designing procedure. The point lies on the injection moulding phenomena of mould cavity filling. When the melts fill the cavity they happen pushing the air ahead of them to the periphery of the wall of the cavity, the air compressed hence the surrounding pressure, temperature raises and might burn the polymer melt nearby or creates a resistance that the air trapped portions at the end walls not completely fill. On the other hand when two gated system cavities or where the melt flows in a way, it splits in to two and where the melt meets form air gap that can result as a hole in the part. Putting this underlined, this happening need to be considered in any mould designing procedure and this air need to escape to avoid the above mentioned points, this process of letting out the air in a mould cavity is venting. [16]

2.2.6.1 Designing of Venting

Venting is designed on the principle that allows air to escape from the cavity preventing out the flashing of the melt plastic material. Based on the driving factor venting classifies in to two as passive venting and active venting. Active venting is where a pressure and pressure gradient different is artificially created to let the air out where as passive venting is the common designing method that the pressure of the melt itself drives out the air from the cavity.

Designing of venting is an optional part in mould designing, meaning; normally as the mould closes the trapped air may escape through the parting line and the ejector pin spacing, but regarding designing of a venting when needed or if there is no way for the air to escape, should be done on the parting line. The dimensions of the venting are directly related to the rheology of the melt, i.e. the type of plastic and its viscosity, the higher its viscosity the larger the dimension of vent. It should be noted that the size must not be too small that prevents expel of air and too large to cause the melt flash out. On the figure below common standard dimension of a particular venting system is demonstrated. [20]

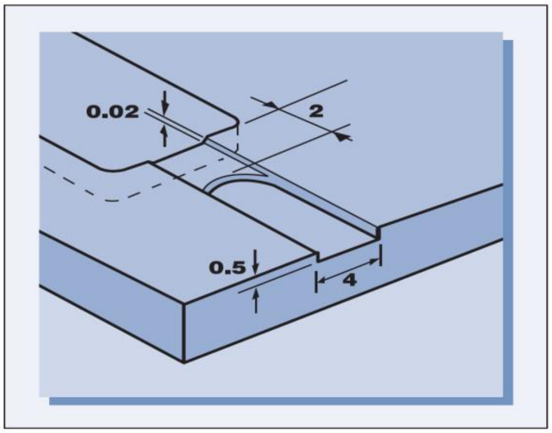


Figure 9: *Construction dimension of venting channel [21]*

When we consider positioning the venting system in a mould, they should be located in a parting line anywhere appropriate for air outlet on average spacing of about 25 mm. Basically during filling the portion of the mould which is thicker section is filled and the thinner section part is fill later that air is trapped and may causes in incomplete filling. Hence, venting should be positioned on areas that will fill finally.

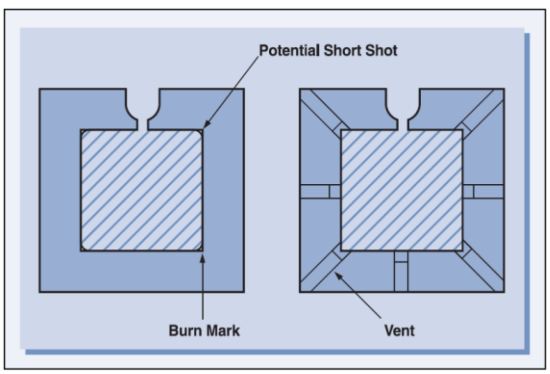


Figure 10: *positioning a vent in a mould [21]*

Another extreme method of venting where the above mentioned procedure is not applicable or further enhancement is needed is to use sintered metal inserts. This procedure is used as last resort on non-visible surfaces and enables the escaping of gases to pass through them without clogging up with polymer. [21]

2.2.6.2 Advantages of Venting

Finally, has been seen in the above points venting should be applicable in most mould designing parts. It enhances the quality of the mould part in many ways and result in a better surface finish, quality and dimensional perfection with consistency.

Overall, venting has the following advantages for a mould quality and avoids:

- ✓ Burn spots
- ✓ Incomplete filling
- ✓ Weak and visible weld lines
- ✓ Poor surface finish
- ✓ Poor mechanical property
- ✓ Local corrosion of the mould cavity surface
- ✓ Irregular dimensioning [16]

2.2.7 Heat Exchange system

2.2.7.1 Overview

To begin with, the term heat exchange applies to mould designing in a significant way, the injected plastic melt material is heated during curing and after injected in the cavity, it is in a constant heat exchange with the mould. This is a decisive factor for any injection mould production and quality. Hence, depending on the temperature of the melt, specific heat of the plastic material and in a relation to the temperature of the outside surface and that of the environment, the mould is treated as a heat exchanger. As it is desired to maintain constant temperature in the mould and cooling for solidification of the product part, the mould is treated with a cooling system for optimal cooling time, efficiency and quality of production. [23]

2.2.7.2 Mould Cooling

After the plastic melt heated by the rotation of the screw in injection mould, it has considerable heat content and after injections in the cavities this heat must be dissipate out from the mould, hence cooling the mould part to become ridged and ready for ejection, this dissipation of heat is mould cooling. It ensures adequate mould temperature control and must be uniform for consistent moulding. This mould temperature must be optimum and regulated carefully since the right properties of engineering plastics achieve at this temperature that has a substantial result on mechanical, shrinkage, flow length and cycle time. On the underground point, efficient cooling system must be employed to maintain this optimum temperature with uniformity and dissipation flow of heat.

Mould cooling is the most significant process in moulding; it takes about 2/3 of the product cycle time, as a result must be designed with a convenient cooling channel efficiently to reduce the cycle time and increase production rate and quality. It has far reaching consequences in the quality of the part as avoiding hot spot, sink marks, shrinkage, and thermal residual stress. [20]

2.2.7.3 Cooling Time

Cooling time is the duration of time that the part in the mould cools and solidifies after injection of the melt in to the cavities. It is usually 2/3 of the total cycle time required for the whole process. In practical application it is related to the heaviest wall thickness of the part and doubling the wall thickness quadruples the cooling time, it is also inversely related with thermal diffusivity of the melt. Thermal diffusivity of the melt is characterized by thermal conductivity, density and specific heat of the melt. Table below revises the cooling time practical relation applications and formulas as below. [21]

Table 5: *Cooling time equation [21]*

$$\text{Cooling time} = \frac{(\text{Heaviest wall thickness})^{2}}{\text{Thermal diffusivity of polymer melt}}$$

$$\text{Cooling time} = \frac{(\text{Largest runner diameter})^{1.6}}{\text{Thermal diffusivity of polymer melt}}$$

where the thermal diffusivity of polymer melt is defined as

$$\text{Thermal diffusivity} = \frac{(\text{Thermal conductivity})^{1.6}}{(\text{Density})(\text{Specific heat})}$$

2.2.7.4 Cooling System Designing

Cooling system designing is vital for a mould design in a way that it ensures quality and productivity by directly altering the cycle time; however it is not necessarily true for all moulds, in case of small production scale the heat from the plastic melt simply dissipates in to the mould and its surrounding environment without a cooling system, but cooling system is strictly important for large scale production where quality and detail precision is an issue. Regarding this designing, it can be done directly by computer programs to determine the cooling pattern for a mould, in this case the computer program is applicable after the mould is designed and takes in to consideration the shape of the product and the mould design itself with other parameters such as the temperature of the melt, coolant and mould material characteristics, but for the need of designing a cooling system before the final stage of the mould, designing of a coolant system should be carried on in alternative practical procedures of designing and understanding the cooling layout and functions. [23]

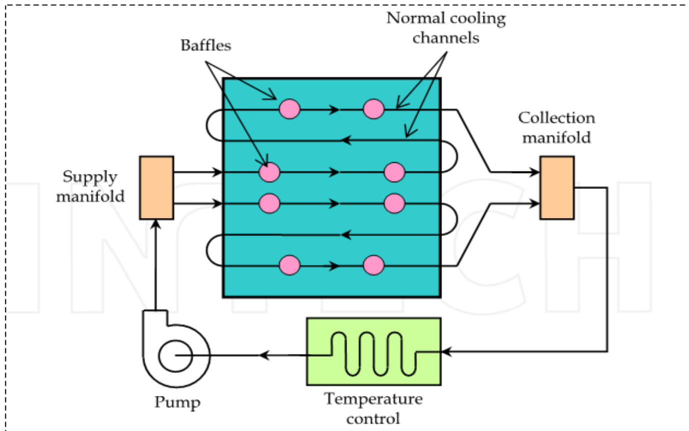


Figure 11: *A typical cooling system layout [23]*

In this typical cooling layout as shown in the figure above, the mould can be considered as a heat exchanger and the coolant circulates in a designed pattern to take the heat that comes from the polymer melt injected in the cavities. Considering the whole cooling system aggregate, it consists of temperature controlling unit, pump, houses, supplying and collection manifold and cooling channels, after understanding the general designing procedures and cooling layout, this part of the study elaborates designing technique and guides for a functional cooling system. [23]

2.2.7.4.1 Designing Principles and Guides

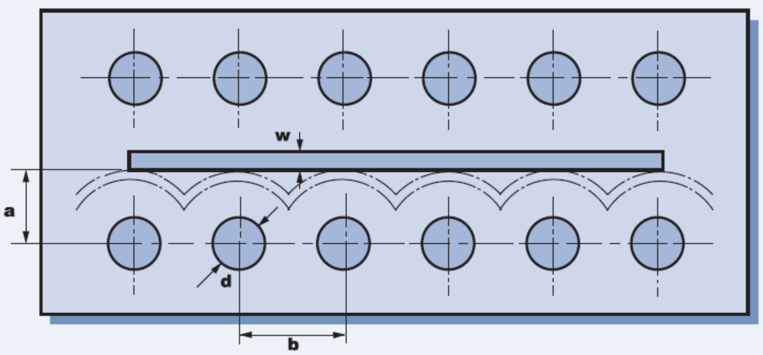
In this subject matter, designing principles of a cooling system and guidelines are covered. Putting it in a nutshell, cooling system designing is carried out by milling or drilling a coolant channel over the surface of the mould closer to the cavities and core. The basic principle is that a moving coolant in the channel circulated through by moving heat content from the high temperature to the lower.

Designing principles

- ✓ The first deigning principle, has been said, lies on the concept that the cooling channels are place on the mould surface closer to the cavities and the core, in even distance uniformly between them. This channels must have a cross sectional diameter appropriate for effective cooling depending on the mould temperature and the melt material temperature with proportion according to the supply manifold underlying or surrounding channels, besides their distance from the corresponding cavities and core should be appropriately chosen. This dimensioning and positioning of the coolant channels are fully summarized in the table below. [21]

Table 6: *Designing and dimensioning of a cooling channel [21]*

"w" wall thickness of the product mm (in)	"d" diameter of the cooling channels mm (in)	"a" center distance with respect to mold cavity	"b" center distances between cooling channels
2 (0.08)	8-10 (0.31-0.40)		
2-4 (0.08-0.16)	10-12 (0.40-0.47)	1.5-2d	2-3d
4-6 (0.16-0.24)	12-14 (0.47-0.55)		



After lying the basic dimensioning and positioning of the channels, It must be designed in such a way that the coolant flow should be in a turbulence flow pattern than laminar, this ensures that the coolant swirl around bringing fresh cool liquids to the coolant surface. This characteristics flow is defined by Reynolds number, Re and a value greater than 4000 implies turbulent flow. Re is defined as:

$$Re = (V * D) \div v \qquad , \quad v = \mu \div \rho$$

Where V is velocity of the coolant in m/s, D is diameter of the channel in m; v is kinematic viscosity in m/s ^{2}, μ absolute viscosity, and ρ density of the coolant. Hence, based on the above analysis the turbulence characteristics can be achieved for a value more than 4000. In most cases for a fast and effective coolant system, Re value calibrated between 10000 and 20000. [21]

- ✓ Considering the coolant temperature with respect to the mould is also a point worth mentioning, for the coolant temperature lower than the core and the cavity in a considerable amount so that the flow takes heat in an efficient way. In regarding multi-cavity moulds, the coolant temperature change as it flows from cavity to cavities must not be large to ensure efficiency and quality achievement, usually change of temperature between 1 and 5 degrees Celsius expected. But there are also cases for greater change of temperature in the coolant flow for some moulds for the need of using less coolant and inexpensive cooling system desire.[15]

Designing guides:

The following are the most important optimum temperature control cooling system guides:

- Symmetrical cooling channels around the mould cavities.
- Short cooling channels to ensure the in and out temperature difference must not exceed 5 degree Celsius.
- Avoid dead spots or air bubbles in the cooling channel.
- Difference in flow resistance of cooling channels cause by diameter changes should be avoided.
- Heat exchange between the mould and the machine should be minimized. [21]

2.2.7.5 Types of Cooling System

When we see close to the flow patterns of the coolant channel, which lays the foundation for the classification of the cooling system, it can flow in a serious pattern from cavity to cavity or core to core, or in some cases the flow splits so that it can have a parallel pattern. In many mould cooling arrangements this pattern can be arranged in combination so that to have a serious parallel pattern, what so ever the case may be, as long as the basic designing principle and guides are fulfilled any of the cooling arrangement can be effective for the cooling system of a single mould or multi-cavity system. Hence, based on this pattern cooling system can be classified as:

2.2.7.5.1 Serial Cooling Channels

This is the most common type of cooling channel arrangement where the flow of the coolant is in a single loop connected from the in and out let manifolds as shown in the figure. In this kind of arrangement, mostly practiced in the mould cooling system by maintaining uniform cross sectional diameter, channel size and allowing turbulent flow for efficient and uniform mould cooling. There can be arranged a serious of serial cooling channels where there is a need for maintaining uniform cooling for large moulds. [23]

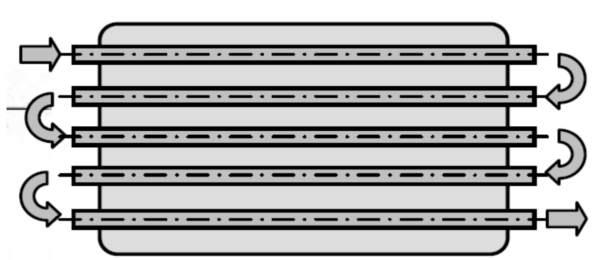


Figure 12: *Serial Cooling Channels [23]*

2.2.7.5.2 Parallel Cooling Channels

In this type of cooling channels, the flow of a coolant from the supply manifold to the collection manifolds are in a split way flows along parallel arranged channels, the arrangement is as shown below in the figure. As characteristics of this type of cooling channel arrangement, it has a feature of varying flow rate due to the arrangement which in turn results the cooling capability of the system to be non-uniform resulting in non - uniform mould cooling.

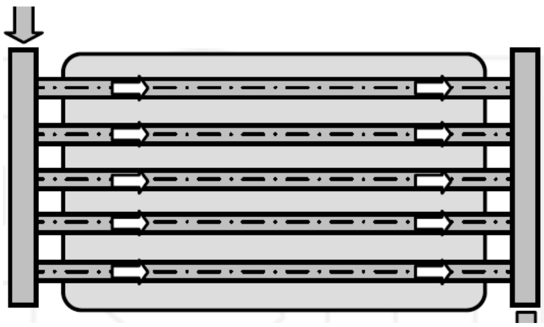


Figure 13: *Parallel Cooling Channels [23]*

2.2.7.5.3 Serious Parallel Cooling channels

Mould cooling system can be arranged in serious parallel arranged channels. The arrangement can be in serious for the flow from cavity to cavity or core to core supplied by parallel arrangement from the supply inlet manifold and finally after the serial pattern flow it can be collected in a split parallel way to the collection outlet manifold. [23]

2.2.8 Shrinkage

2.2.8.1 Introduction

Underneath all the basics of mould designing principle and methods, there is a vague part usually uncovered and controversial; it is so vital that has by far consequences in part precision and general overall productivity quality. Shrinkage, literally speaking refers to the actual volume change with respect to the original. Regarding mould shrinkage of plastics materials, it encompasses the high tendency compressibility characteristics of plastics than any other material up on load, that is due to the injection temperature expansion and the compression due to the respective pressure, Hence, though in many studies it is difficult to achieve an exact shrinkage rate due to this counteracting facts in the mould process, in general plastic materials exhibit a mould shrinkage of about 2%. [15]

2.2.8.2 Mould and Material Shrinkage

Mould shrinkage is a vital issue in the designing process and should be considered in relation to the temperature dynamics, which mainly affect shrinkage as the melt injected with high temperature and cools in the mould, also the injection pressure contributes to a considerable rate. As of this fact different plastic materials exhibit shrinkage properties depending on their inherent material characteristics, meaning weather they are semi/crystalline or amorphous. On further elaboration, amorphous materials such as polycarbonate, polystyrene and ABS have a homogenous molecular structure, so that during cooling and after withdrawal of the injection pressure, they exhibit a uniform all part shrinkage whereas semi-crystalline, such as the material used in this thesis study, polypropylene, shrinks at a different rate that the crystalline part shrinks more than its amorphous part as it cools resulting in a non-uniform shrinkage properties. Usually it is difficult arriving at exact shrinkage factors but for mould designing application on practical case, a range of shrinkage factors are available by a material suppliers between the range 0 to 5 % and some common plastic material mould shrinkage value is listed.

Table 7: Material Mould Shrinkage values [24]

COMPARATIVE MOLD SHRINKAGE VALUES

Material	Type	Shrinkage/in/in
Polypropylene	Semi-Crystalline	0.010 – 0.025
Polyethylene	Semi-Crystalline	0.015 – 0.040
Nylon (6-6)	Semi-Crystalline	0.007 – 0.018
Acetal	Semi-Crystalline	0.018 – 0.025
ABS	Amorphous	0.004 – 0.009
Polycarbonate	Amorphous	0.005 – 0.007
Polystyrene	Amorphous	0.004 – 0.007
PPO	Amorphous	0.005 – 0.008

Key: in/in, inch per inch shrinkage

On top of this, mould shrinkage is a complex topic in mould designing and the relative plastic materials exhibit different shrinkage characteristic based on their inherent molecular and chemical characteristics. Mould shrinkage is mainly affected by the temperature dynamics and the injection pressure but also there are many variables directly related with the plastic material, such as mainly, the type of melt, melt temperature, the injection pressure, the ejection pressure, part geometry and the thickness of the part design are among the main.

To sum up, shrinkage should be considered in most designing that needs part cooling inside the cavity mould system, and consider the part shrinkage during ejection to have a proper and accurate size and production quality. Shrinkage should be studied along with the filling and injection system so as to get a best result for professional mould designing procedure. [24]

2.2.9 Alignment

Mould designing gets its part design precision and accuracy up on the feature of alignment mostly apart from the other mould designing guides stated, as this thesis literature covers the theory of the methodology part all along the way, which is the mould designing of the part product, alignment at its best is employed. Alignment is needed in a way that the part design assembles precisely with different features considered after the part design is done. Hence, points like holes tolerance for load bearing, plate and face mates of the mould, pins and bushing system diameter and length must much chosen correctly from the commercial standards for accurate part design product result.

Beside The alignment feature application in the mould assembly part of the design, its application is further related together with the Injection moulding machine. As looking in the technical application of the mould aggregate after assembly, it feet in the clamping unit of the machine, hence, alignment is necessary along the palates of the machine. Basically speaking, when the clamping units of the machine closes it guides the mould aligned in some ways but that is not enough since during injection high pressure results shifting of the core, cavities and also miss much of parts of the mould aggregate. As a result of this the mould must align with plates of the machine and with its own internal alignment features must be kept aligned as classified and stated below. [16]

2.2.9.1 Types of Alignment system

- Alignment with the axis of the injection unit
- Internal alignment and Interlocking
- Taper Lock between cores and cavities

2.2.9.1.1 Alignment with the Axis of the Injection Unit

Alignment with the axis of the injection unit, which is from the tip of the nozzle and the mould part of the sprue bushing system, is mandatory for proper injection of the melt to pass to the cavities, for this very purpose alignment is made between plates of the machine and the mould plate by a flange like locating ring holes and pin attachment, this alignment is designed in such a way to make the nozzle tip concentric with the sprue bushing unit match and avoiding shifting during Injection. The locating ring is the key for this alignment type and it has a tapered inner and flange outer surfaces that pass from the machine plate and fasten up a little bit to the mould plate, its diminution diameter and length vary according to the type of the injection machine plate part with available commercial standard dimensions. [25]

2.2.9.1.2 Internal Alignment and Interlocking

Though the tie bars plate of the machine guides the mould during closing and opening of the clamping unit that is not enough for the mould to be aligned in a perfect way, as a matter of fact the mould need to be aligned internally by alignment interlocking mechanism which is mainly done by the leader pin and bushing system. The leader pin connects the core of the mould haves to the cavity haves and slide to feet in and out during mould closing and opening so that guides and aligned the mould perfectly, also on its match mould haves it has a bushing system and at the end of it a locating sleeve for perfect alignment. The diameter and the length of the leader pin depend on the wall thickness of the part product and the depth of the cavity with proper dimensioning and tolerances of available commercial standard. The most common leader pin is the shoulder type and its alignment in a mould with its bushing system and locating sleeve as shown in the figure, also available commercial standard in the reference file. [25]

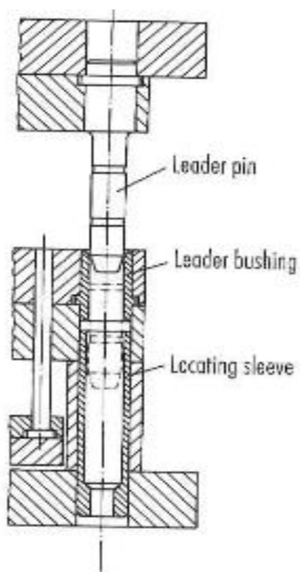


Figure 14: *Shoulder Type leader Pin and assembly [16]*

2.2.9.1.3 Taper Locks between Cores and Cavities

In addition to the tie bar plate locating bushing system and leader pin bushing alignment system, there is a need for taper alignment of the core and the cavity for proper dismantling to avoid locking, this is done by taper and before the mould closes with the clamping force the core and the cavity haves have a spacing called pre-load which matches each other during clamping. Tapering angle is as desired by standard between 5 to 20 degrees for perfect much. [15]

2.2.10 Ejection System

2.2.10.1 Overview

In this part of the theory literature, ejection system, deals with the final part of the injection moulding basic process where the plastic part product is dismantled, as we have been through and hence, at the final stage after the melt solidified in the mould core part, it cools down and solidified so that it is ready for removal, even if some moulds do not have ejection system for the removal of these plastic parts, most common moulds incorporate a provision for ejection system.

As this subject matter deals with this ejection mechanisms under covers that the ejection system associated with the movable part of the mould which is the core and aggregated with this is ejector plates where the ejector pin and sleeve originates. There are several ejection mechanisms like air valve, stripers but the most common and applicable is the ejector pin sleeve method. [26]

2.2.10.2 Ejector Pin and Sleeve Method

During mould designing, the proper layout, positioning and designing of ejector system is mandatory for the visual appearance of the part design; most importantly the proper functioning of the mould and production improvement is almost impossible without.

On a design base consideration of an ejection system, it is mounted on the moving half part of the mould core as of mainly using the ejector pin and sleeve, here by, it projects from the ejector plate just above the core and extend via the core to the part surface, hence, during the back ward movement of the core, the part remain inside after cooled and in contact with these pins to push it out, and then before the beginning of the second cycle of the injection machine the retaining pins retract to make ready the system for the second cycle. The ejector pins are available with different diameters from supplier catalogue but there diminution is chosen mainly based on the size of the part thickness and also the ejection temperature determines the size and type of the pins and sleeve to use. Documentation on types, size and diminution is given along with the reference file. [26]

3. Methods and Results

3.1 Overview

As a general overview of the methodology and the result part, it is structured and explains in such a way that, software application for designing the part design and the resulting analysis and simulation result part are compiled together in a coherent and flow pattern, it incorporates the general product designing procedures in distinct phase of designing, simulation, analysis and obtaining the desired optimum result.

This part of the thesis progresses through designing the part component, which is the Dom in innovative and identity wise perspective and incorporating it with the jar content design and finally achieving the unique Manual Juice Maker over the available one on its special property and advantages as mentioned in the part design section. Then after several melt flow analysis are done to meet the required optimum values of operating in Autodesk Moldflow Synergy, finally apart from obtaining the part product design, the final optimum melt flow analysis is carried on and discussed. Hence it covers as designing criteria the following parts as listed below:

- ➢ Part design of the manual Juice Maker
- ➢ Melt flow simulation and analysis
- ➢ Analysing, comparing and finding optimum values and forecasting to IM

3.2 Computer Aided Designing

On meeting the required level of perfection, accuracy and proper product finishes now a days computer aided designing is vital and crucial. Especially when we talk about plastic product designing, the tie goes beyond apparent level of notice, simply explaining, it is almost impossible without the aid of this applications. As briefly mentioning some of the advantages:

- ✓ Creating a visual appearance of the part design in a professional discreet manner before production.
- ✓ Taking the Limit of dimensional consistency and accuracy almost to perfection level and creating the channel for editing, enhancing the part design before production.
- ✓ Achieving a design in a minimized time span with different editable and enhanced parameters.

There are broad categories of computer aided designing and simulation software, depending on the required result and feature of the product, they can be chosen and applied on computer based designing procedure accordingly.

In these thesis product designing the software SolidWorks is studied from the scratch and applied for designing the part product as mentioned and elaborated on the next section and melt flow simulation and analysis is done in Autodesk Moldflow Synergy but on general designing product and simulation perspective there are other software also applicable and capable of achieving a part designing, simulation and prototyping some of them are mention as follow:

- ***AutoCAD, Solid Edge, and SolidWorks:*** Applicable for 2D and 3D designing of a required part design with each a distinct parameters, Interface and further can be employed for assembling of the part design for a mould aggregate compilation.

- ***Autodesk Moldflow Synergy:*** Simulation and analysis of a melt flow by setting injection point and optimizing it for optimal phase and values of filling the cavity, clamping force and also resulting with optimal analysis of filling time, under varying mould designing parameters such as clamp force, temperature and pressure it analyses the optimal phase of injection parameters.

- ***Nastran N x:*** Employs analysis of FEM and buckling with much more other static options for stability of the part design and obtaining the maximum retaining stress and buckling load.

- **Mastercam:** Applicable Creating a milling simulations of a mould aggregate part, such as core and cavity plate visually with analysis of appropriate and optimal machine time for production and mostly for manufacturing of the mould in digital connection with CNC machine and prototyping.

3.2.1 SolidWorks

Currently, in the present world designing industry, SolidWorks took the biggest share for visual designing creation and mostly applicable in designing industries for product part and mould assembly of plastic companies with a clear foresee boosting influence on a designing world, as of the need for meeting and being competent over the designing spectrum along with this thesis I have done detail studies of the software from the scratch level, which actually took most of the time in the thesis work, here by the part design is done purely starting from the scratch of Dom to the jar container in a unique identity and innovative way and final achieving of the aggregation of the product Manual Juice Maker part design.

SolidWorks 2012 is covered and applied in detail level to the extent of a mm level sketch line and curves accuracy on the part design, on generally describing the application software at this stage, it has enhanced and flexible advantageous feature to create a two dimensional sketch with a user friendly interactive sketch tab interface, as a peculiarity observation of the sketches can be back edited in edited mode and other component sketches can be created by the side as a layer appearance on the sketch layer as another distinct sketches which lies here the designing foundation of the software main part. Most interestingly, it is quite simple and gives the option of creating relations within sketch components of lines, arcs, splines, polygons which gives leverage for manipulation of a drawing at its best for elegant finish and dimensional accuracy with a vital parametric relations and aspects of each component.

Up on further observation and practice over the software further, it enables modelling 3D objects with its feature tab pan components feature such as extrude, revolve and loft command. Actually the Loft command is much more applied in the part design of the thesis work and its detail application and step is described in the procedure and method part, also more complex features can be achieved using the sweep and loft command. On the other hand 3D objects can be refined easily using the cut, loft and revolve cut commands.

SolidWorks at its best has automated wizards which easies modelling that are built in within the software, such as holes and cuts with uniform size diminutions.

Finally, it has an interactive and diverse option features for building assembly structure from parts and correlate the parts in the drawing with the assembly. On top of this, gives an option of auto mood or manual labelling of parts and arranging a bill of materials.

3.3 Product Design

Product design is a vast topic and section in the production industry, this thesis topic concerns and studies product design of a plastic product, simulating and analysing for IM, which is a Manual Juice Maker. As can be known, currently there are available Manual Juice Makers already but most of them are made up of coasty metallic material and some made up of expensive and uneasy plastics design. The need for designing this new innovative idea Manual Juice Maker and this thesis idea of making it comes from the perspective of easing designing style of the product with a better appearance and handy, substituting it with less coasty, strong and flexible plastic material easy for using and most importantly, a special study carried on for attaining the maximum efficiency of the device by enhancing the squeezing mechanism which used to be already in the market and adding a special technical and geometry features for the new device.

The product design of the thesis topic, the Manual Juice Maker, incorporates a detail product designing, simulation and analysis of the part design with its unique and special features as mentioned above, the melt flow simulation and analysis of the plastic material polypropylene carried on several times. Then after compared with different analysis trials, based on the calculated and assumptions values. Finally, the final feasible and optimum analysis is compiled, forecasted and customize for injection moulding manufacturing applications.

At this preceding section, the thesis project studies the part design. The part product is designed and studied in two main components that are the Dom, where enhanced squeezing mechanism and designing features applied as described in detail in its part and the aggregate, the Manual Juice Maker where the Dom and a containing jar combined to result the desired final finish of the plastic product.

The methods for designing based on the starting ideas, assumptions of designing and the study made in the project with outcome results of software designs, compiled as a method and result are presented on the respective orders and chapters.

3.3.1 The Dom

The Dom is the squeezing mechanism part of the Manual Juice Maker, it is designed in a unique way as of the seek for efficiency with its peculiarity as this project studies and incorporated with a containing jar of its own to result the Manual Juice Maker of this thesis project with its specialty.

The Dom, regarding designing study of this thesis project, unique appearance and enhanced features for maximum squeezing techniques with a physical on the ground realistic study is employed. It starts with the scratch point of view that a fruit to be squeezed, considering a typical orange or most fruit will have an average diameter of about 80 mm after sliced horizontally and ready for squeezing.

Based on this and the next study before going to the designing part is “How can we get a maximum squeezing result” analysed, it is assumed in a special way and researched that the squeezing efficiency and capability lies in its peripheral outside shape and the groves along its longitudinal direction. Hence, up on several trials made on designing software and discussions, finally achieved in such a way that the Dom has base and top circles with a curvature spline connecting them resulting a curvature outside profile.

The base and the top circles profiles connected with grooves pattern of rectangle for squeezing mechanism connected along the longitudinal profile of the Dom in a specially enhanced unique peculiarity that they are shifted and twisted with in an angle of 45 degree on each plane so that an angle of 22.5 degree tilting between the top end and the bottom plane groves and when connected results a longitudinal grove curvature as desired as shown in the figures and covered in the designing method procedure with a special feature of acting as curvature bend counter clockwise from bottom to top acting as a blade shaft for maximum squeezing ability efficiency for the clockwise rotation of the fruit along the curvature of the Dom surface when fitted inside and rotated.

Designing Method:

- Based on the average diameter of the fruit and the vertical diminutions, appropriate physically realistic diminutions are set for the basic diminutions of the Dom:

  Bottom Base Circle Diameter: 120 mm
  Top end Circle Diameter: 20 mm
  Vertical Length: 150 mm

- 80 mm reference circle in a horizontal plane is calibrated along the vertical reference plane of the Dom for the average diameter of the fruit to fit at a chosen vertical plane from the squeezing point at a distance of 50 mm from top.

- A grooving rectangular profile is added at the edge of the bottom and top base circles profile of diminutions 5 mm * 5 mm and patterned along this circles 45 degree on each plane and shifted from bottom to top within an alignment of 45 degree range and between them 22.5 degree spacing

- On the first designing procedure on SolidWorks, the base bottom circle and the top circle are sketched with a vertical distance of 150 mm on two distinct planes and with all diminutions as stated above.

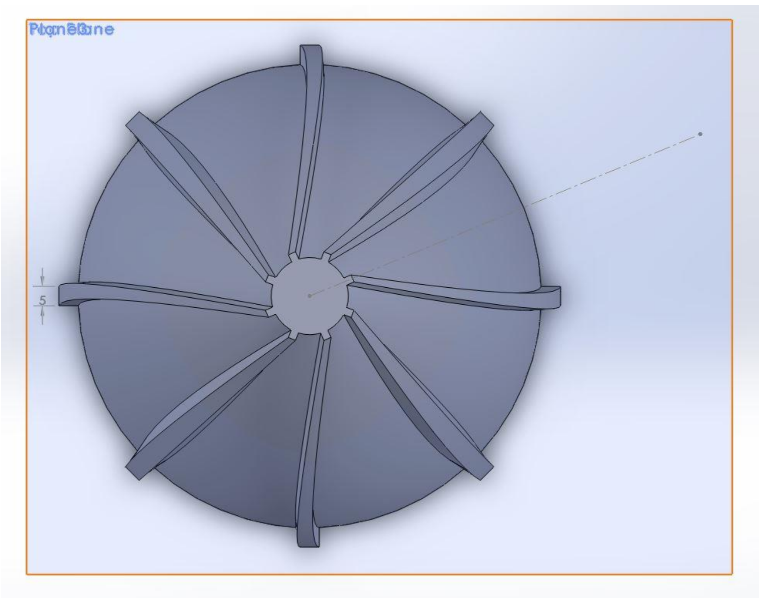


Figure 15: *Top view result of the Dom*

- A connecting spline guide curve is sketched extending from the bottom circle to top one in the 3D sketch mode with a guideline of distance of 80 mm diameter circle calibrated at 100 mm from the bottom.

- The two base circles with a connecting profile spline as a guideline are created as the Dom polygon with the feature command of solid works loft command and the connecting spline as a connecting guide.

- The grove rectangles are patterned 8 times using the patter feature of the software shifted at an angle of 22.5 degrees between the two planes after aligning with the following rectangular pattern grove an angle of 45 degree on each plane.

- After launching the software SolidWorks, going to the sketch mode view the two circles are drawn on top view with diameter 120 mm and 20 mm on different planes that are separated vertically 150 mm

- On 3D Sketch mode of the software SolidWorks, taking the 80 mm diameter horizontal profile as a guide line splines are drawn connecting the bottom and the top circle and as stated above line with all the patterns of the Dom outer curvature in a uniform way with equal curvature spacing.

- Using again The loft command of SolidWorks feature the bottom and the top grove patterns are lofted and connected using the spline as a guide curve and a 3D object of the Dom with the groves achieved as shown

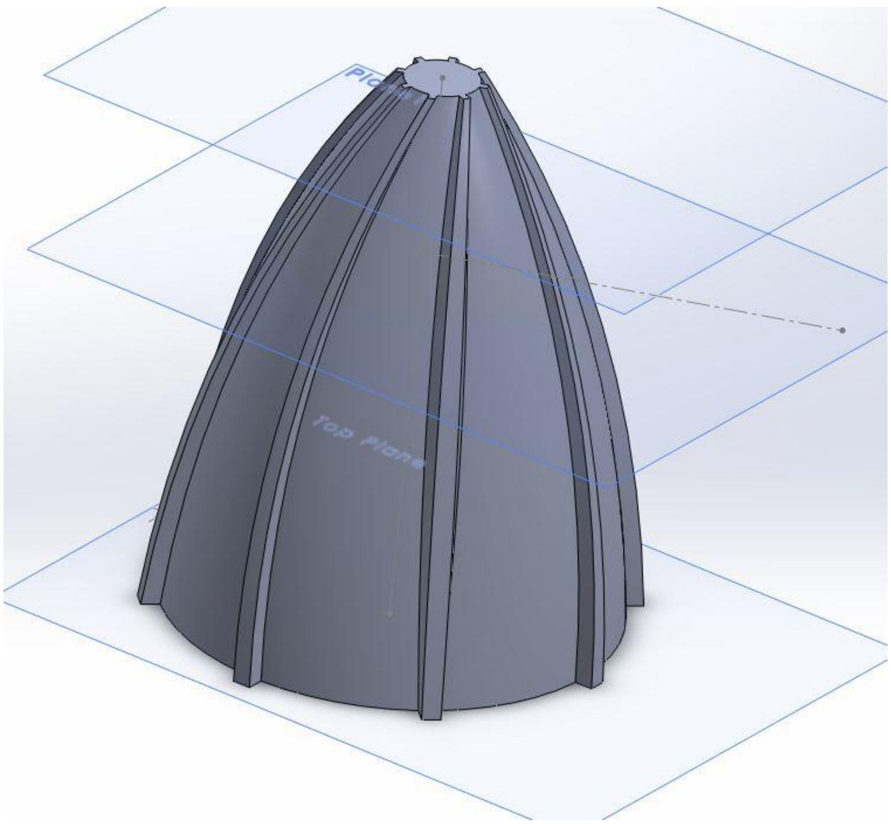


Figure 16: *Perspective view result of the Dom*

- The final part in designing the Dom is making it hollow profile to save material in the production, hence a 3 mm thickness is assumed for realistic practical applications. Using the loft cut command applied between two inside top and bottom circles and a connecting spline curve as a guide to drill out resulting a profile of 3 mm thick geometry shells hollow body.

To sum up, the figure bellow shows the Dom which is finally incorporated with the jar to result the Manual Juice Maker, as a final product design result. The Dom is peculiar in its design for an enhanced squeezing mechanism and material saving made with plastic material, it has a hollow inside with its thickness of 3 mm. Here the top, bottom and section view of the Dom with the detail illustration of the grove and the extending grove longitudinal profile is illustrated.

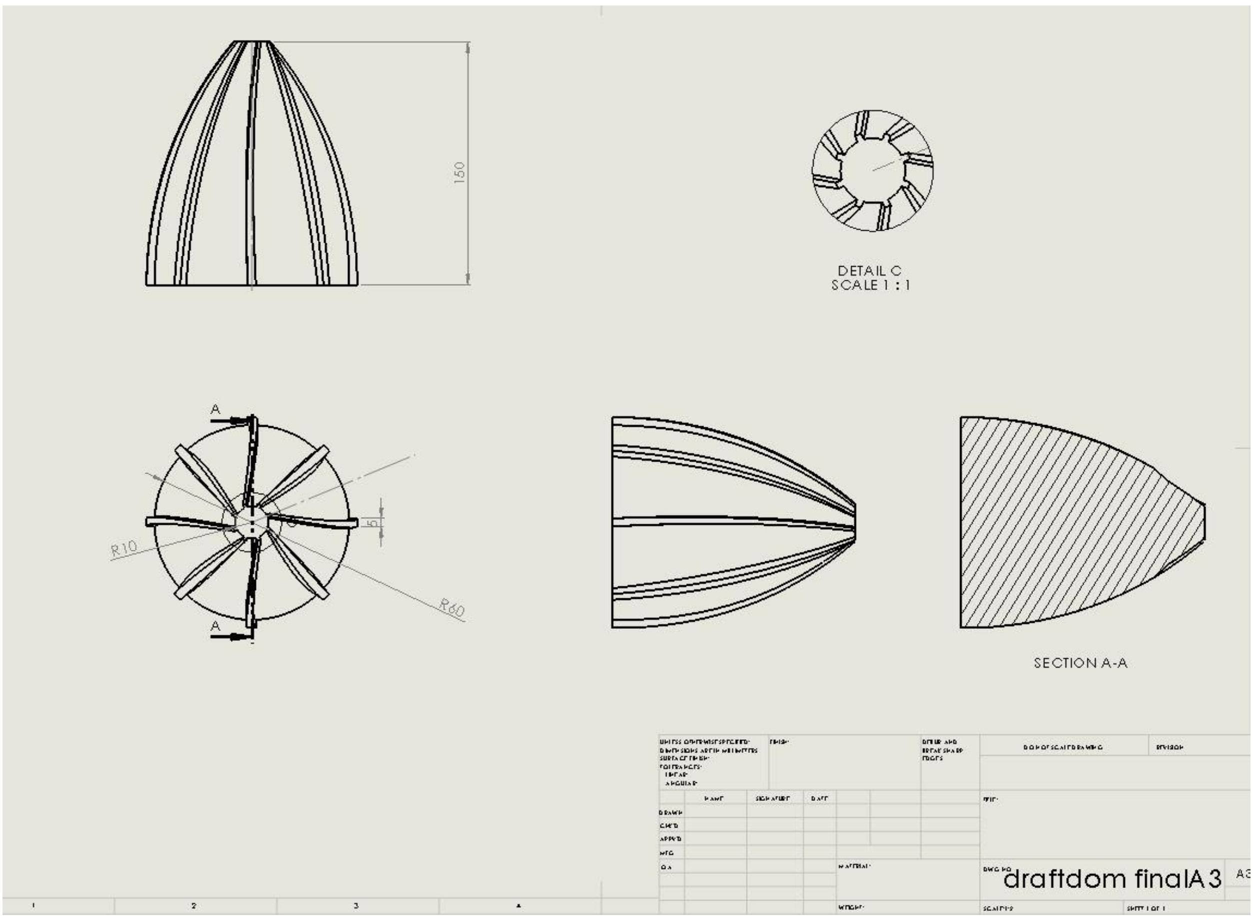


***Figure 17**: Draft detail view result of the Dom*

3.3.2 The Manual Juice Maker

The Manual juice maker is an aggregate composition of the Dom over the containing jar design mid-plane on vertically up ward geometric build up.

Describing it in a plane geometric descriptive manner, the Manual Juice Maker is just a merge up of the Dom over the containing jar. First, the Dom is designed and tested separately as in detail described in the previous section, after achieving the desired Dom design then the containing jar figured out for proper diminutions and styles, finally the containing jar is designed and the Dom is built up on the mid bottom plane upward to obtain the Manual Juice Maker design.

Furthermore, before coming to the final version setting of the diminution, shape, style and attributes of the Manual Juice Maker, some studies also carried on its manufacturability and appropriate physical diminutions on the ground level like the Dom has been discussed in the previous section. Again, before coming to the final version of the Manual Juice Maker design a couple of designs has been done several times and tested, arriving at this design finally which can be manufactured in standard mould diminutions in a physical mould and compatible with a feasible melt simulation and analysis application with a potential for real physical world production and manufacturability in injection moulding.

Designing Methods:

- Based on the above mentioned points, proper designing aspect and easy appearance with physical observations, the jar contain is set to have diminutions
  Base circular ellipse of Diameter: 200 mm*160 mm
  Length of containing jar: 100 mm
  Wall Thickness: 3 mm

- Launching the software SolidWorks, and going to the sketch mode after choosing the top plane for sketch a circular ellipse drawn for the containing jar of diminutions 200 mm * 160 mm for the base container
- This feature is extruded using the extrude boss command a length of 100 mm
- The resulting geometric part is refined with the feature tab of Shell command a thickness of 3 mm
- Then the resulting shape of the container is refined again from the feature tab of SolidWorks with draft command selecting the base surface as reference to an angle of 3 degrees, which is desired for proper mould opening procedure

- At the next step, the handler is built extruded out from the rear end of the jar with the horizontal centre line alignment, having a length of 30 mm *30 mm with a semi-circular arc joining end of radius 20 mm.

- The final designing part of the containing jar is the tip where the liquid purse, it is the most complicated part of the design next to the Dom grooves, here the tip have to be slanted horizontally and vertically also to get the exact shape an inclined planes are chosen as a reference in the referencing geometry option.

- Tip of the Jar is achieved by drawing in sketch mode a triangle on the top plane and a reference guide lines in the 3D sketch mode on the inclined plane connecting the triangle and a reference point on the surface of the jar and on the inclined plane. By alternately applying the Loft command and next the loft cut command at the outer and inner triangle respectively the shape achieved properly after a few trials.

- The top view of the containing jar with the Dom build up at the centre is illustrated below in a shaded edge view mode.

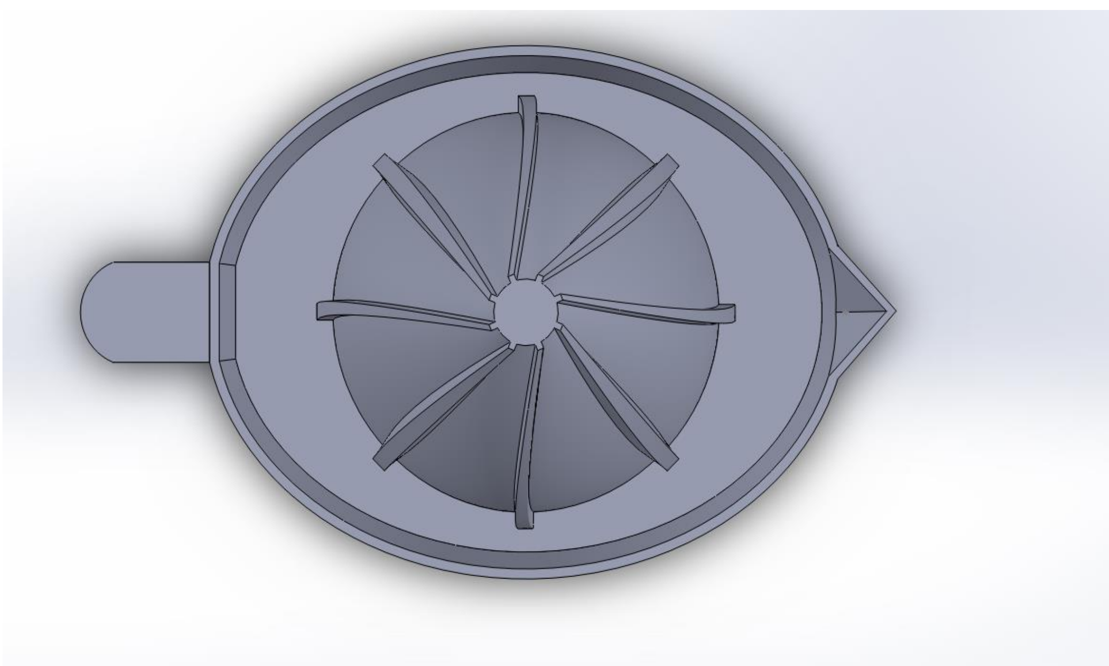


Figure 18: *Top view result of the manual juice maker / containing Jar in the outer*

After the desired containing jar design is achieved, The Dom is constructed inside the jar again likewise done separately and discussed in the Dom methodology part and finally, resulted the Manual juice Maker design, the desire to do the Dom separately at the beginning was it has a lot of complicated profiles which makes the design inconsistence and complicated constructing it altogether and for the sake of achieving maximum squeezing potential with the detail study of manufacturability of the Dom separately needed.

In addition with the identity and its characteristics it has need to be studied as the main part of the thesis project separately and tested for dimensional stability and parametric consistency and also simplicity for mould manufacturability.

- Once the containing Jar is done designing, proper positioning, alignment and planes are chosen over the base of the jar and the Dom constructed, which is mentioned in the Dom methodology part repeated with perfection (after Dom trial alone achieved) starting from the mid- plane and centre line vertically
- As shown in the figure bellow, the Manual Juice Maker design is achieved and the part design is finalized in a net dimensional consistent, proportion and mould production standard size with consideration of many aspects that is mentioned in the thesis project identity and most of all in a geometric descript way compatible for further mould flow melt simulation and analysis.

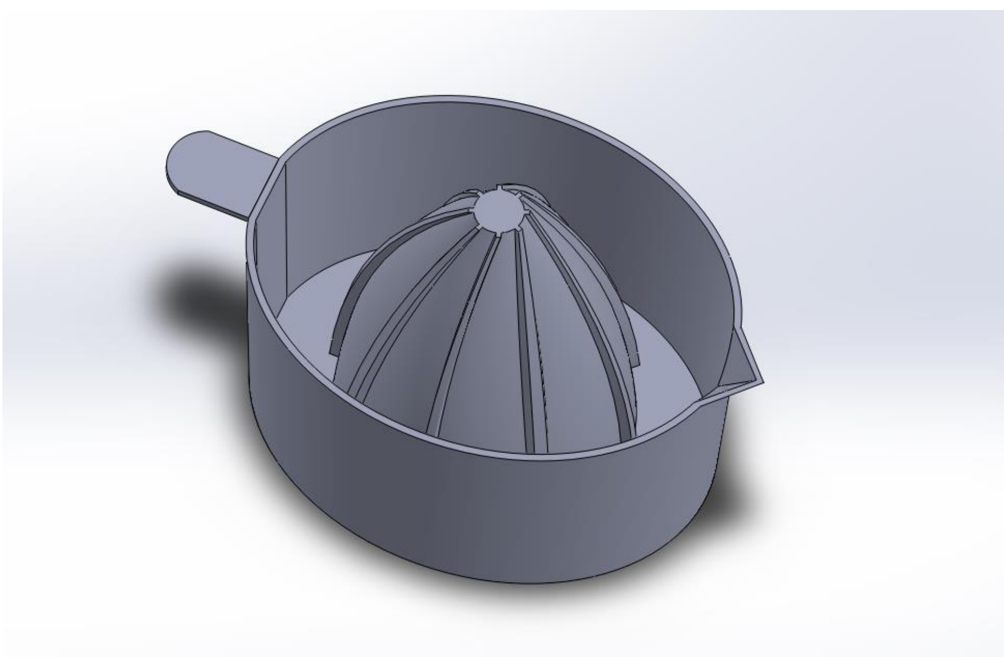


Figure 19: *Isometric view result of the Manual Juice Maker*

Here, it is presented the draft view of the manual juice maker, which is an aggregates of the Dom and the containing jar which has been studied and tested separately, done at one shot for the final Manual Juice Maker part design result with top, front, section and detail views as shown below.

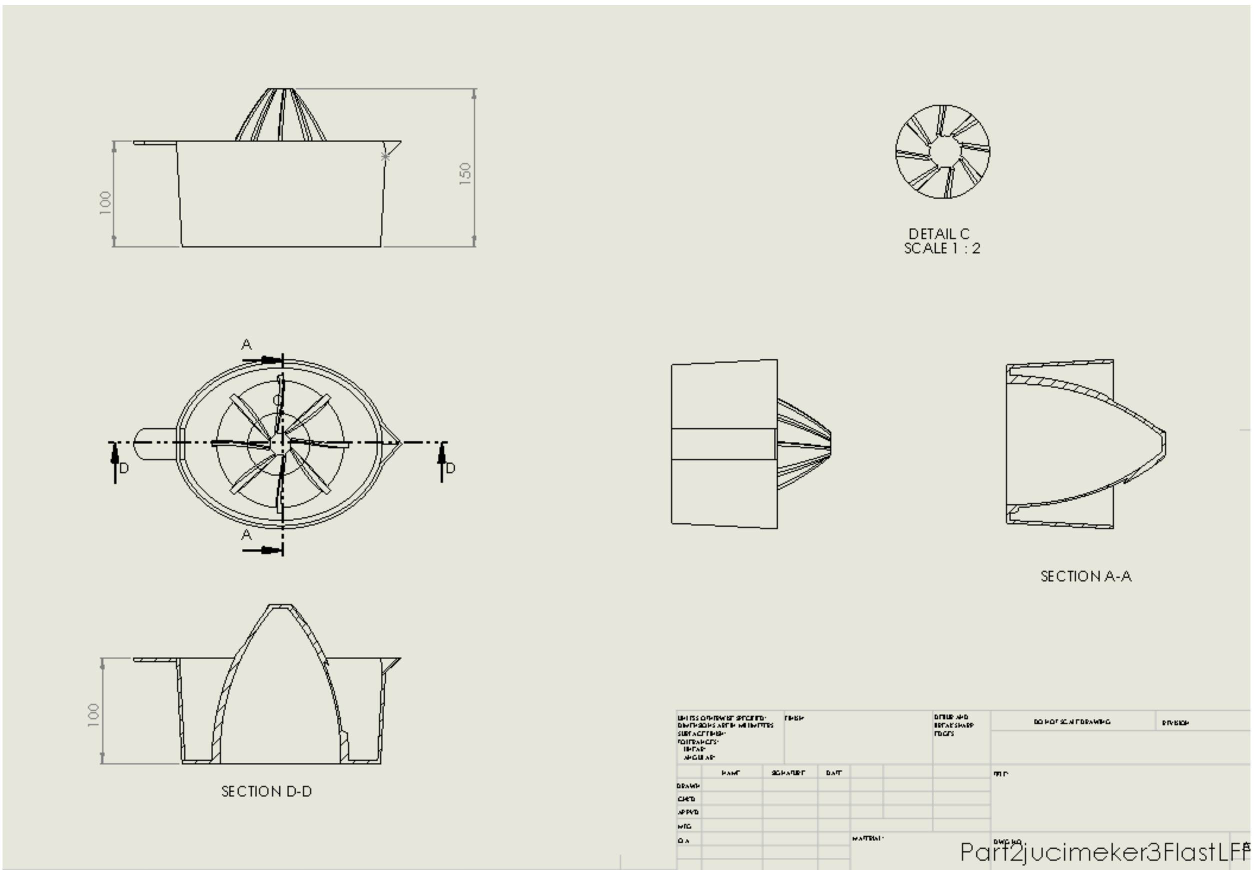


Figure 20: *Draft View Result of the Manual Juice Maker*

3.4 Mould Flow Melt Simulation and Analysis

3.4.1 Autodesk Moldflow Synergy

On this thesis project, the software Autodesk Moldflow Synergy is employed on the second phase of the designing plastic part product simulation and analysis. The software optimize the part design which is the Manual Juice Maker through simulation setups, It analyses the material type of the part design that best suites, which is polypropylene melt for results, defines and interpret optimum injection points for maximum filling of the cavities, optimum clamping force and optimum fill time, based on the operating pressure and clamp force of the setup of the software and assumptions made at initial start, it generates optimum possible results for production of the:

- Average flow velocities
- Temperature
- Clamp force
- Fill time
- Injection pressure

After setting the basic operation parameters of the injection mould machine in the software and choosing the injection point on the part design then after the melt material defined, it results optimum melt simulation and analysis values for the product part design at the final stage after trying several analysis. In this section of the method and result part for the melt flow several trials are carried on using the software Autodesk Moldflow Synergy where the final best result is obtained on the last trial based on customization and close looking of the first few trials.

Here in the following section the basic parameters for the melt flow of the injection moulding machine and characteristics of the part design are first defined and then analysis is carried on in several successive orders until optimum values are obtained, the method procedures and the related results are compiled in a flow pattern and presented as follow.

3.4.2 Melt Flow Parameters of the Manual Juice Maker

There are parameters and some values of the part design need to be discussed and assign a value before running the melt flow simulation and analysis setup. For the sake of professional analysis and under lying theory and basic principle of the injection moulding process, some points are mentioned and covered below as a pre-request setting and parameters before simulating and obtaining a melt flow analysis results.

- ✓ Screw speed of the machine:

  The screw speed of an injection machine ranges on average from 10-130 mm/s and hence for this analysis an injection machine which commonly uses is assumed for the speed of the analysis to be 80 mm/s and also commonly used radius of the screw to be 15 mm.

- ✓ Volume Flow Rate :

  The volume flow rate for the melt analysis is assumed also based on the data of the screw speed and screw diameter values and calculated as:

  $\boldsymbol{Q} = \pi r^2 * V$

  $3.14 * 15^2 * 80$ mm ^{3} /s

  56520 mm ^{3} /s

  <u>*56.52 cm ^{3} /s*</u>

- ✓ Projected Area Of the part design

  The projected area of the part design which is along the clamping unit normal is generated from the software SolidWorks and its value is

  Projected Area, $\mathbf{A_p}$ = *652.6470 cm ^{2}*

  <u>*65264.7 mm ^{2}*</u>

- ✓ Material Type of the Melt:

  As been discussed in the literature part the material type of the plastic part design is polypropylene due to various facts discussed, and hence in the melt flow analysis the simulation is carried on assigning PP material

- ✓ Operating Injection Pressure

The injection Pressure for a moulding machine usually ranges to a maximum of value as 220 MPa and for this melt analysis optimum machine injection pressure value is chosen for applicable range and set as 110 MPa

Operating injection pressure, $\mathbf{P_o}$: 110 MPa

- ✓ Clamp Force of the Machine

Applied clamp force of the machine, $\mathbf{F_c}$: 100 tonne

- ✓ Calculated mould clamp force

The force holding the mould closed during the injection process is the clamp force, and for this analysis its calculated value is found as a multiplication of the projected area and the injection pressure and finally the simulated optimum value is generated by the melt analysis.

Hence, the calculated mould clamp force is

Clamp Force $\mathbf{F}_{cc}$ = A (projected) * P (injection Pressure)
652.6470 cm ^{2} *110 MPa
7.179 m ^{2} MPa
7179 kN
718 tonne

- ✓ Intensification:

Intensification is the rate at which the hydraulic injection moulding pressure is converted in to the injection operating pressure at injection point and for the analysis a common optimum value is chosen from the range.

Intensification: 11

3.4.3 Melt Flow Analysis

As been said, several trials are carried on regarding the melt analysis of the part design where a logically feasible and optimum value is obtained in the final one based on the first few trials.

On all procedures common methods are carried on at the beginning where the basic simulation parameters are defined, the difference in the trials is mainly changing the injection mould machine advanced parameters for optimum results. The general procedures are covered below and then the final optimum procedures methods with the following results are compiled after wards.

3.4.3.1 Meshing and meshing methods

Meshing is the collective name for the geometric subdomains that is a descriptive entities lying under for the software to recognize the geometries and analyse the subsequent fluid melt flows.

In Autodesk Moldflow Synergy there are three types of product design geometric meshing, in this thesis project of the part design, dual domain meshing methods is used as of the importance of less time and rapid melt simulation and analysis results. In general perspective the three Meshing types are:

- ✓ Dual Domain Mesh
- ✓ Mid plane Mesh
- ✓ 3D Technology

Meshing Methods

- Export the file in SolidWorks with IGES format for analysis on Autodesk Moldflow Synergy
- Import the part In Autodesk Moldflow Synergy
- Mesh Now, using dual domain type

✓ After the meshing command is executed and processing finished the following mesh part result of the Manual Juice Maker is obtained for dual domain mesh.

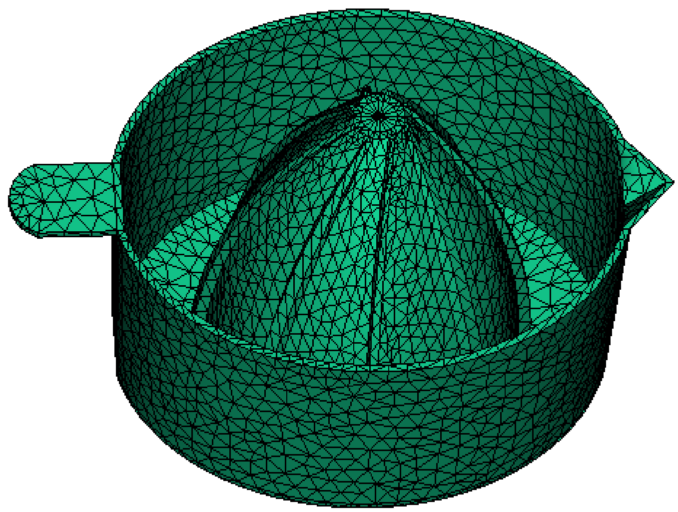


Figure 21: *Meshing result of the Manual Juice Maker*

The implication is that the software disintegrate the part design in to triangular or /and tetrahedral elements and joined by nodes, at the node connection points been a transferring point of load and force for the analysis. From the above mesh analysis it is done perfectly to result the mesh version of the product design and revels that the part design in SolidWorks is completely compatible and matches for melt simulation and analysis. Hence, there are no overlapping nodes so that the simulation carried out and generates the result that would have been not so if the mesh is not proper.

3.4.3.2 Mesh Statistics

Before analysing and running the fill simulation and analysis details, as mentioned above the meshing procedure should be carried on first for the software analysis to recognize the part design for the required analysis. After meshing procedure is done here comes an important point called mesh statistics, this parameters and the statistics data must be cheeked for consistency and proper flow pattern for the analysis.

The mesh statistics has important data values of parameters need to be observed, they are mainly:

- ✓ Entity Counts and
- ✓ Edge Details

Based on the above main statistically obtained mesh data categories the following details need to be cheeked for right values and data also.

Connectivity Region:

This parameter describes the integrity of the part design as a single entities for analysis, indicates that the part design is acquired as a single whole connected body for the analysis.

Edge detail:

This party of the mesh value related with the part design mesh edges and has the following main sub categories of free edge, manifold edges and non-main fold edges.

Free edge value of the mesh statistics cheeks and analyse the connectivity of the part design meshed edges element single entities and all the edges are joined together whereas the manifold edge element is where in the mesh constituent another element of edges are attached.

Table 8: *Mesh statistics table data of the Manual Juice Maker*

Model details :	
Mesh Type	= Dual Domain
Mesh match percentage	= 93.9 %
Reciprocal mesh match percentage	= 94.5 %
Total number of nodes	= 5260
Total number of injection location nodes	= 1
The injection location node labels are:	2670
Total number of elements	= 10520
Number of part elements	= 10520
Number of sprue/runner/gate elements	= 0
Number of channel elements	= 0
Number of connector elements	= 0
Average aspect ratio of triangle elements	= 2.0570
Maximum aspect ratio of triangle elements	= 59.4945
Minimum aspect ratio of triangle elements	= 1.1593
Total volume	= 215.8506 cm^3
Volume filled initially	= 0.0000 cm^3
Volume to be filled	= 215.8506 cm^3
Sprue/runner/gate volume to be filled	= 0.0000 cm^3
Total projected area	= 652.6470 cm^2
Filling phase results summary :	
Maximum injection pressure (at 6.2244 s)	= 20.1124 MPa

As can be shown by the mesh result data, the simulation analysis statistically obtained data is correct with no overlapping edges or nodes and it generated the nodes of the value given for dual domain mesh type, the total number of nodes obtained is 5260, besides the simulation analysis interprets a single entity injection point as desired for the optimal filling at the tip of the Dom. On top of this, the meshing generated the total volume of the product part design of 215.8506 cm ^{3}, in addition to this the part projected area is 652.647 cm ^{2}. To conclude that, having the above found results, the meshing and the statistically obtained data is correct and the design is simulate able to proceed and can be analysed to generate the simulation analysis results perfectly.

3.4.3.3 Fill Analysis

In this thesis study project of the Manual Juice Maker a fill analysis is made for a single cavity mould. Before running the analysis a feasible injection point and parameters is set as described in fill analysis methodology part. Accordingly, the injection point is set at the upper tip of the Dom for efficient, fast filling time and optimal filling.

Hence, as a subject matter of this study, running the fill analysis is important for the designing and manufacturing of a plastic product in a way that it determines the flow characteristics of the plastic melt of the designed product and predicts the proper fill pattern. Most importantly, from this software analysis it makes understandable most important melt parameters such as the melt flows, pressure distribution, mould temperature and optimum clamp force.

3.4.3.4 Mould Flow Analysis Methods

After the general procedures carried on, meaning importing the file in Autodesk Moldflow Synergy and meshing it according to the method described above. Then after the right mesh statics achieved the melt flow procedures proceeds as follow:

- ***Importing*** IGES file in Autodesk Moldflow Synergy
- ***Meshing***
- ***Fill:***

  Material customizes:

  Generic PP, Basell Polytifia PP

- ***Set Injection Point***
- ***Processing Setting:***

  Filling control
  Flow rate: 56520 mm^3/s
  56.52 cm^3/s
  Injection Pressure: 110 MPa

- ***Temperature Setting***

  Mould Temperature, 40 degree Celsius
  Melt Temperature, 240 degree Celsius

- ***Injection Moulding machine Advanced Setting:***

  Injection Moulding Machine:

  Maximum Machine Injection pressure: 220 MPa
  Hydraulic pressure Intensification Rate: 11
  Machine Clamp Force: 100 tonne

- ***Analyse Project***

3.4.3.5 Mould Flow Analysis Results

1. Average Velocity

Average melt velocity is the rate at which the polymer melts flows in the cavity. Hence, from the simulation analysis of the manual juice maker it is resulted as shown below in the figure in the blue region which is an optimum value of about 23.52 cm/s. Moreover though the analysis shows allowable range of maximum average velocity possibility of 94.07 cm/s it is quite enough for the product to be in the range of the blue zone and finally, it is also resulted average melt flow of the melt duration to be 8.035 s to cover all the part.

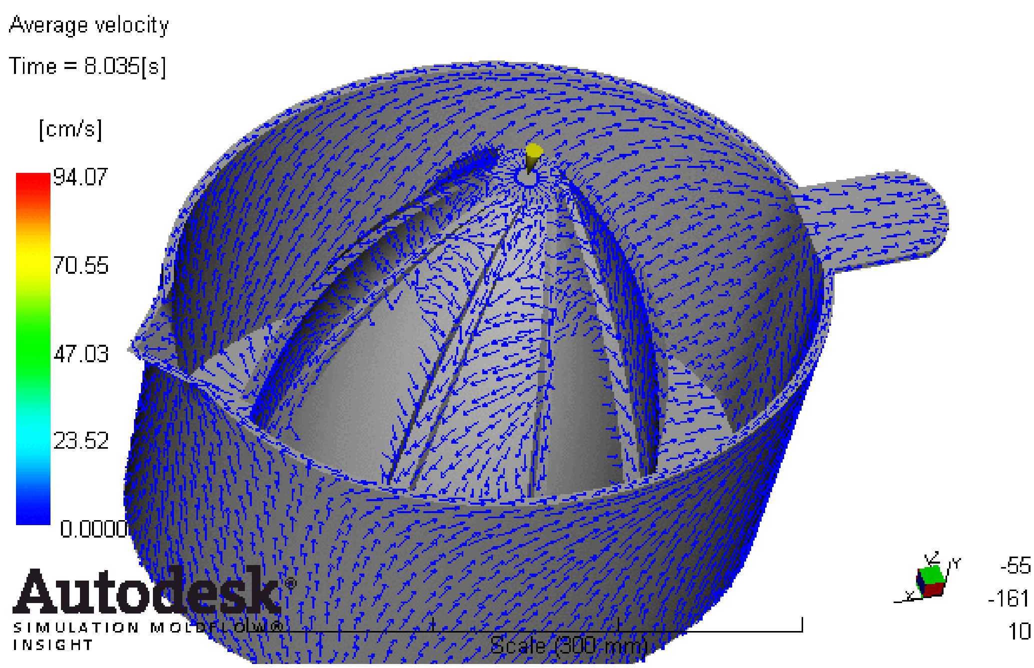


Figure 22: *Average velocity analysis result*

2. Temperature

This part of the analysis result is based on the idea that there is a heat exchange phenomena in the mould between the melt and the walls of the mould, practically once the melt is injected in the cavity. Hence, this heat exchange is shown from the analysis result and optimum feasible temperature exchange rates are achieved.

Regarding the temperature simulation analysis has been stated in the melt analysis methodology setting part of the mould temperature is set to be 40 degree Celsius and the melt temperature is set on 240 degree Celsius. Based on this the simulation analysis resulted as shown in the figure bellow of about a maximum of 240 degree Celsius on the red zone of the melt part on the cavity which then drops after filling with a time interval of 6.889 s to a blue part on the region to a temperature range of 70 degree Celsius where then on wards thermal stabilization achieved and the mould temperature stays in equilibrium.

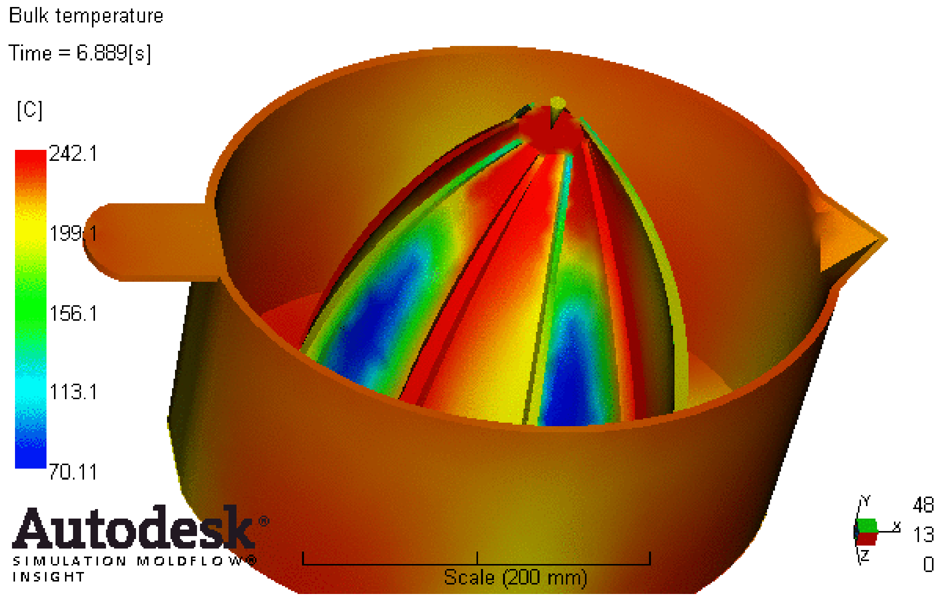


Figure 23: *Temperature analysis result*

3. Clamp Force

From the analysis result a clamp force of 70 tonne is required for the maximum filling of the cavity as an optimum result. On the analysis setting for the machine the clamp force is assigned as 100 tonne after several trials and it is near the simulation analysis results and feasible being optimum for the result. Moreover the analysis tells there need this 70 tonne amount of clamp force for the part product manufacturing which is on average feasible range of IM machine operation practically. Below the plot of the clamp force versus time is shown.

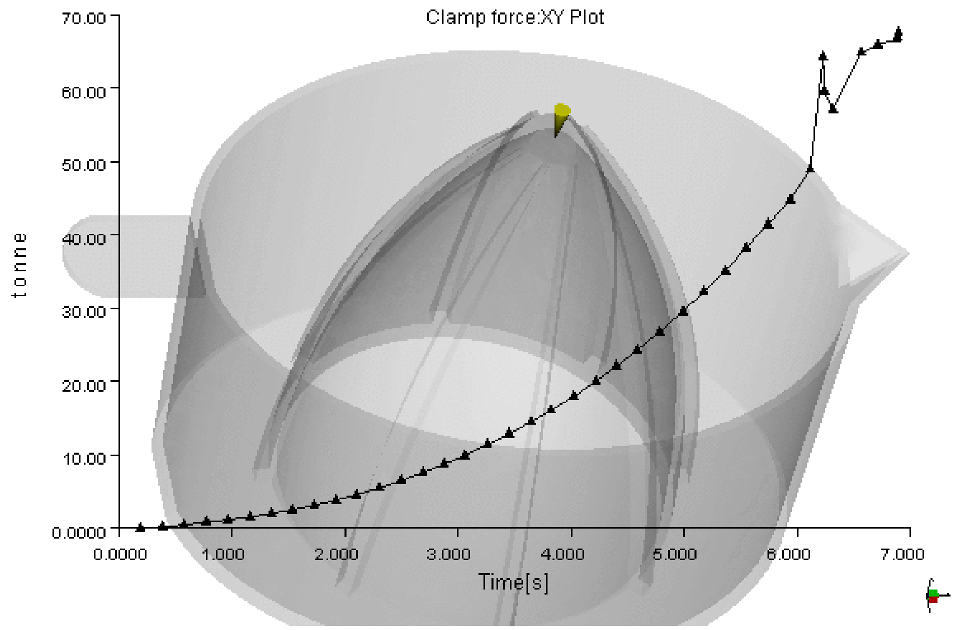


Figure 24: *Result of the clamp force analysis*

4. *Fill Time*

The concept behind this fill time simulation analysis is that, there is a duration that the machine from the nozzle injects the melt and fills the cavity, this is the fill time.

As can be shown from the practical melt analysis simulation that perfectly fits with the theory part stated, filling time starts with the blue region where the optimum injection point set for the maximum filling and propagates to peripherals of the cavity as it passes the blue, green and finally maximum filling time of red region which takes for all full filling of the cavity to be a duration of 6.889 s. The detail plot of the fill pattern with time is shown below.

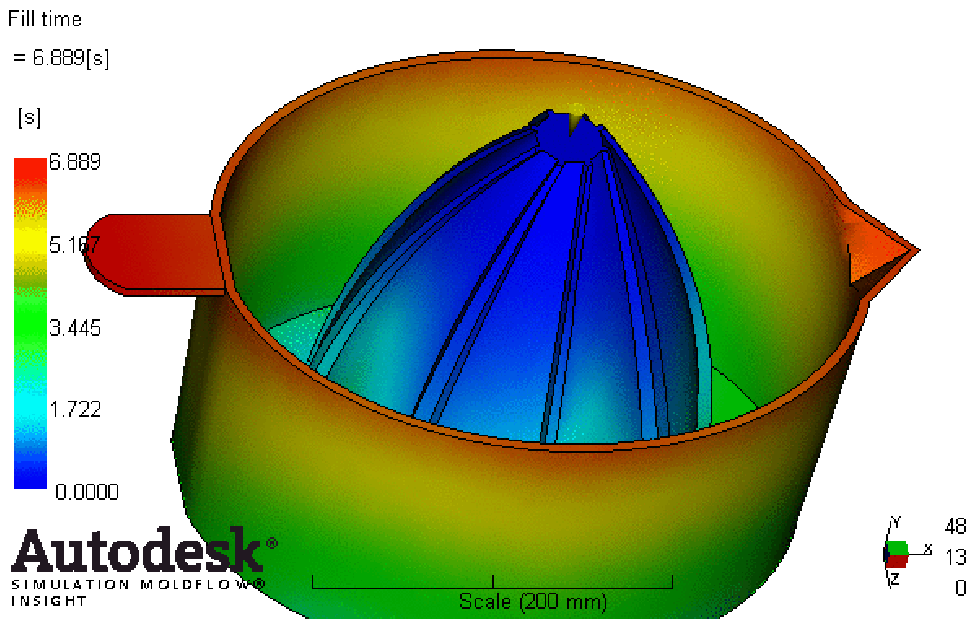


Figure 25: *Fill time analysis result*

5. Fill Pressure

To begin this portion with, at the time of injection the melt injected with high pressure at the point of injection, which is the pressure from the hydraulic machine that is converted in to injection pressure based on a normal intensification value of 11 at a specific injection point shown by arrow in the figure. The analysis is considered with an injection machine of maximum pressure 220 MPa and operating pressure of 20 MPa as a start basic setting of the analysis.

The result obtained from the simulation of the analysis shows that though a maximum pressure in the red zone is allowed as high as 20 MPa the melt can completely fill the cavity with the yellow region of maximum optimum operating pressure of 15 MPa that would be enough for complete filling, and hence this pressure is enough for the operating pressure of the mould. Gradually this injection pressure drops after filling the cavity with this optimum pressure to lower values as shown on the figure bellow passing the green and then blue region as it propagates and drops to the peripheral parts of the cavities.

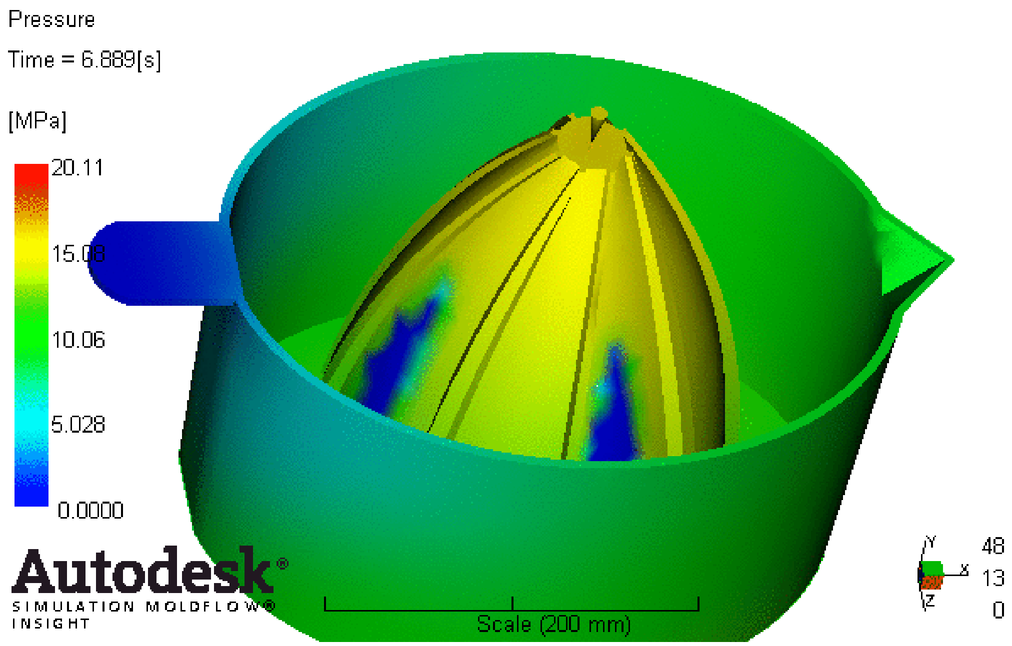


Figure 26: *Pressure analysis result*

On summarizing, comparing with initial different simulation and analysis trials and contrasting the melt simulation and analysis results, the assumed parameter assumptions, based on the standard average range IM screw diameter and feasible sate of volume flow rate, the final optimum simulation results compiled on the table below. As a result of this, at the final simulation and analysis of the manual juice maker a logical, practically applicable and approximated simulation results obtained after trying the simulation analysis several time and obtaining optimum results on the final stage. Hence, operating pressure, clamp force and fill time is on feasible range of practical IM machine. For the record, the calculated clamp force and assumed operating pressures can be further optimized and approximated to assumption of the simulation analysis by reducing the lateral projected area of the part design and/or changing the volume flow rate to increase further and dropping the operating injection pressure of IM machine setting. But what have been found is optimum much enough based on the initial assumptions, background of several analysis trials carried on and at the final stage the result is in a feasible operating range being applicable in reality.

To conclude, these results obtained from the assumptions and parameters and based on previous initial trials, with relating to the final simulation results are optimum and closed so practical and feasible at the final stage. The simulation and analysis was carried on several times before obtaining this much of optimization with approximation to the assumption and the final stage simulation analysis is quite feasible and practical, hence, this obtained values and data can be considered applicable, forecasted, projected and used on a real industrial manufacturing of the Manual Juice Maker in IM machine.

Table 9: *Simulation and Analysis Results*

SIMULATION ANALYSIS	RESULTS
1. Average Velocity	23.52 cm/s
2. Temprature	242 Degree Celsius
3. Clamp Force	70 tonne
4. Fill Time	6.889 Seconds
5. Fill Pressure	10.6 MPa

4. Discussion

On further study, discussion and elaboration of this thesis study, though the part design finally achieved is based on several trials and the underlying research theory and capability of maximum squeezing potential, from the point of view of manufacturability on the ground level, compatibility with melt simulation and analysis and mould production some amendments can be done like reducing the lateral projected surface area that directly influence the clamping force reduction for IM operation. Also this customization of the part design further may result other simulation generated results of the melt flow analysis that can be fit for real world production more though what already achieved is quite feasible and practical, resulting parameters such as reduced operating pressure and clamp force. All this facts and other points results on a possibility of a lower load injection moulding machine applicability for the production of the part design with a reduced clamp force which makes the Manual Juice Maker more realistic for manufacturing in industries on a ground level easily.

Furthermore, the mould aggregate containing the core and the cavity as a main constituent can be projected for further study and designed in a more industrial level with adding different enhanced features. On mentioning the most basic, the venting system can be incorporated along the parting line of the mould which enables for air outlet during mould clamping and ventilation for the part design in the mould avoiding burn marks on the product and enabling transparent and clear surface finishes. In addition to that, most importantly the cooling system, which dramatically increases the regulation of the thermal system in the mould aggregate, can be designed along the lateral surface of the core and the cavity for sophisticated and multi- production moulding with fast cooling time that reduce the cycle time for IM with direct implication of reduce cost and economic manufacturing.

Finally, after the core and the cavities designed and the mould aggregate achieved the next step can be employed for the prototyping simulation using Mastercam. Hence, the core and the cavities can be simulated using this software and the resulting milling analysis can be achieved, at the end the G code can be generated with this software which feeds directly in to the CNC machine for the manufacturing of the core and the cavities practically using the most suitable material steel.

5. Conclusion

To summarize this thesis study, which is a product designs for an injection moulding machine and simulation of a Manual Juice Maker product design, a comprehensive designing study using the software SolidWorks carried on and basic theory of mould designing for IM is covered besides at the beginning basic plastic material analysis was discussed. The Dom, keeping its complexity in mined was tried designing several times until the desired functional part geometry is achieved from the perspective of enhanced maximum squeezing potential and capability with logical understanding of its real life manufacturability, then incorporated with the jar, that is designed along with the Dom in appealing and ease of use to result the Manual Juice Maker product. At this stage the software SolidWorks, keeping its designing capability and functional integrity is studied in detail and applied to achieve the product design needed after several trials.

Along the progress of the study assumptions made and studied regarding the melt simulation and analysis parameters of the manual juice makers as described in detail in the methodology part, values are found for the calculated clamp force and assumed injection pressure and volume flow rate based on the value ranges of an average injection moulding machine. Once the assumptions and parameters of the manual juice maker analysed and found following the simulation analysis is carried on in all aspects of the melt using the software Autodesk Moldflow Synergy, hence the flow of the melt material which is selected to be polypropylene is analysed and accordingly caring out several trials and finally came up with optimum analysis of the melt fill time, clamping force, average velocity and fill pressure results. These calculated values and assumptions are compared and contrasted with the melt flow analysis simulation generated results and the results are discussed, and to sum up, the final melt result is compiled and optimum values are obtained doing the analysis and simulation for polypropylene material with results obtained are feasible injection moulding machine pressure, clamp force and also operating injection pressure. The results finally obtained are a value of 70 tonne clamp force and 20 MPa operating pressure, which are so practically possible and feasible for the manufacturing of the Manual Juice Maker using an average load Injection Moulding machine but cannot be manufactured using Arcadas IM machine since it is over 50 tonne.

The thesis study can further be projected with the designing of the core and the cavity, which is a bit complex and more time demanding due to the geometry of the Dom and its detail contents. The mould assembly can be done using the software SolidWorks part and assembling designing option to result the whole mould aggregate design of the Manual Juice Maker product design. At the final stage the real industrial production of the mould can be done using most suitable material of steel. The core and cavity can be manufactured with mould making milling machining such as CNC after making the prototype simulation with Mastercam which finally makes on the ground level real for the manufacturing of the Manual Juice Maker practically in industries for IM.

References

1) *Toranne and Stefanies Plastics. (2013).Classification of Plastics and History [Online]. Availabel : http://dwb4.unl.edu/Chem/CHEM869E/CHEM869ELinks/qlink.queensu.ca/~6jrt/chem210/Page3.html#Chemical Nature. [Accessed 24November 2013]*

2) *Michigan State University. (2008). Plastics Characteristics. [Online].available: http://www.msue.msu.edu/objects/content_revision/download.cfm/revision_id.493530/workspace_id.-4/01500090.html/ [Accessed 27November 2013]*

3) Rolf Klein. (2011). Material peroperties of plastics (1st ed). [Online]. Available : http://www.wiley-vch.de/books/sample/3527409726_c01.pdf . [Accessed 03Decmber 2013]

4) Enginger. (2014). Plastics Classification. [Online]. Available: http://www.ensinger-online.com/en/materials/basics-of-plastics/plastics-classification/ [Accessed 18December 2013]

5) R J. Crawford, Plastics engineering, 3rd ed . Jordan Hill, Oxford :butter worth heineman,1998

6) PHS, Plastics Historical Society. (2011). What are Plastics. [Online]. Available : http://www.plastiquarian.com/index.php?id=2&pcon. *[Accessed 22 December 2013]*

7) *Ashish Shah. (2014). Selection Criteria for Plastics Materials. [PDF]. Available :* http://www.ideaproductdesign.com/corp/eflyer/dzine/june2009/article.pdf *[Accessed 4January 2014]*

8) *Modern Plastics. (2014).Properties of Plastics. [PDF]. Available : http://www.modernplastics.com/pdf/IAPD_plastics_intro5.pdf. [Accessed 13January 2014]*

9) Axel. (2004).Thermal Peroperties. [Online]. Available : http://www.axelproducts.com/pages/thermal.html *[Accessed 19January 2014]*

10) *Jared Spaniol, Jack Rulander, Mike Leo. (2014). Polypropylene. [PDF]. Available : http://www.personal.psu.edu/users/j/m/jms5157/Polypropylene%20Paper%20Part%20I.pdf. [Accessed 27January 2014]*

11) *The University of York. (2014). The essential Chemical Industry Online. [Online].Available: http://www.essentialchemicalindustry.org/polymers/polypropene.html. [Accessed 27January 2014]*

12) *Iapd, Introduction to Plastics. (2014). Typical Properties of Polypropylene. [PDF]. Available : https://www.iapd.org/bookstore/property_tables/pp.pdf. [Accessed 02Febrauary 2014]*

13) *Azom.com Limited. (2000). Plastic Injection Moulding. [Online]. Available : http://www.azom.com/article.aspx?ArticleID=265#_The_Mould [Accessed 07Febrauary 2014]*

14) *Jim Ammon. (2008).Injection Moulding :Mould Consruction and Part Design. [PDF]. Available :http://idsa.sjsu.edu/Archive%20documents/injection%20molding%20lecture%2009- 04-08.pdf. [Accessed 18Febrauary 2014]*

15) *Herbert, Understanding Injection Mold Design, 1st ed. Cincinnati, USA>Hanser Gardner Publications, 2001*

16) *Menges, Michaeli and Mohren, How to make Injection Molds, 3rd ed. Cincinnati, USA: Hanser Gardner publications,2001*

17) *Global Plastic injection Moulding. (2013). Hot Runner Systems. [Onlin].Available :http://www.global-plastic-injection-molding.com/how-to-pick-a-hot-runner-system.html. [Accessed 24Febrauary 2013]*

18) *Husky Injection Moulding System ltd. (2014).Ultra Hot Runner Nozzles. [Onlin]. Available : http://www.husky.co/newdynamic.aspx?id=297.[Accessed 26Febrauary 2014]*

19) *DSM Engineering Plastics. (2005). Designing Guides: Cold and Hot Runner Systems. [PDF] Available: http://www.dsm.com/en_US/downloads/dep/**design**broch03USweb.**pdf** [Accessed 01March2014]*

20) *Bayer Material Sience LLC. (2000). Part and Mould Design. [PDF]. Available : http://edge.rit.edu/content/P12056/public/Part%20and%20Mold%20Design.pdf [Accessed 28Febrauary 2014]*

21) *DSM Engineering Plastics. (2005). Designing Guides: Engineering Plastics. [PDF] Available: http://www.dsm.com/en_US/downloads/dep/**design**broch05USweb.**pdf** [Accessed 05March 2013]*

22) *Quickparts. (2013). Basics of Injection Moulding Design: The Gate. [Online]. Available:http://www.quickparts.com/learningcenter/basicsofinjectionmoldingdesign.aspx. [Accessed 12March 2014]*

23) *InTech, University of Ulsan. (2012).Design and Simulation :Cooling Channels for Plastic Injection Mould. [Online]. Available : http://www.intechopen.com/download/get/type/pdfs/id/34669 [Accessed 03April 2014]*

24) *EquiStar, Lyondell Company. (2014). Mould Shrinkage. [PDF]. Available : http://www.lyondellbasell.com/techlit/techlit/Tech%20Topics/General/Mold%20Shrinkage.pdf. [Accessed 24March 2014]*

25) Choice Mould Components Inc. (2013). Mould Alignment : Leader Pins and Bushing. [Online]. Available :http://www.choicemold.com/leader-pins-bushings/shoulder-bushings.php [Accessed 06April 2014]

26) Polytechnic Institute of Leiria. (2001).Mould Design, Ejection System : Ejector Pins and Sleeves. [PDF]. Available : http://webhotel2.tut.fi/projects/caeds/tekstit/mould/mould_ejectors.pdf [Accessed 08April 2014]

Appendix:

Plasticos: *To be able to be shaped or moulded by heat.*

Polymer: *Chemical structure that contain long chain carbon molecule.*

Monomers: *constituent molecules consisting of carbons atom that aggregate to form polymer.*

Paraksin: *First developed plyometric material before innovation of plastics.*

Additives: *Chemicals added to plastic to give specific chemical and physical properties.*

Viscoelastic: *Property that material exhibit both viscous and elastic, where viscosity is the diffusion of atoms or molecule inside an amorphous material.*

Elasticity: *change in shape of a material at applied stress and returning back to original shape after the stress removed.*

Young's Module: *Constant value that determines the stiffens of a material.*

Glass Transition Temperature: *A transition temperature above which materials exhibit different physical and chemical properties.*

Elastomers: *Elastically malleable type of plastics that cannot be reshaped after melting but upon removal of load can be restored back to original shape.*

Thermoplastics: *Type of plastics that molecular structure not cross linked and they are thermo formable, gets to a viscous melt up on heating and solidifies back to original shape up on cooling.*

Thermosetting Plastics: *Characterized by molecular structure undergoing a secondary reaction and molecules cross linked, cannot be returning to original shape up on cooling.*

Propene: *The monomer constituting ingredient for the formation of the polymer polypropen.*

Ziegler Natta: *A catalyst used for the formation of polypropen, which is a mixture of aluminium chloride and aluminium alkyl.*

Resin: *A molten plastic material that is plasticized and ready for processing.*

Shot: *A proper dosage of a plastic melt material to fill a cavity at a single injection.*

Barrel: *Horizontally lying a tunnel component of IM machine that contains inside the screw and outer surrounding of heater where the plastic plastsizes and ready for injection.*

Nozzle: *IM Component that connects the barrel to the sprue bushing of the mould forming a seal between the barrel and the mould.*

Mould: *An aggregate of the injection and the ejection sides, that assembles and fit inside the clamping unit of the IM machine which gives upon de moulding the part design needed.*

Cavity: *A concave hollow shape part of the mould that forms the outer shape of the part design.*

Core: *A matching of the cavity that usually have a convex shape.*

Parting Line: *A separation plane that exists between the core and cavity.*

Sprue: *A delivery system that provides passage for the molten plastic from the machine nozzle tip to the cavity part of the mould.*

Runners: *A connection mechanism that connects the sprue to the gates for the flow of the melt plastic material.*

Gate: *A small opening at the ends of the gate where the melt passes and injected to the cavities.*

Venting: *A mechanism of letting out a compressed air during plastic melt filling out of the cavity.*

Cycle Time: *A total duration time for a single production cycle starting from the injection, part solidification and until the end of part ejection.*

Cooling Time: *A duration of time for the part to cool and solidifies in the Mould just before Ejection.*

Shrinkage: *Actual volume change of the melt material in the cavity with respect to the original.*

Alignment: *fitting and guiding of the mould component along with the plates of the IM machine and internal mould component matching fitting system to avoid shifting of core and cavities during injection during the mould opens and closes in the clamping unit.*

Alignment Interlocking: *A leader pin and bushing guiding system that connects the core and the cavities, aligns them with the back plates and slides back and forth for matching the components during opening and closing of the clamping unit.*

Tapper: *A draft angled tilted surface usually for the matching of the core and cavities alignment and used for easy of dismantling the mould.*

Pre Load: *Internal spacing between tapered core and cavity matching before the clamping unit closes.*

Ejector Pin and Sleeve: *A provision of ejection systems that originate from the ejector plates.*

Melt: *A viscous plyometric plastic material that flows and fills the cavity which later solidifies in the mould and dismantled to give the part product.*

SolidWorks: *2D and 3D industrial standard designing software for part design, assembly of product designs and mould aggregates.*

Autodesk Moldflow Insight: *A bundle software package version that includes Autodesk Moldflow Synergy as one software entities for melt simulation and analysis.*

Autodesk Moldflow Synergy : *A graphical user interface form of the software version Moldflow Insight package for the simulation and analysis of polymer melt flow.*