



Internship Report

Master in Product Design Engineering

***Study on water-assisted injection molding
technology***

Name: Viveganandan Ravikumar.

Leiria, June, 2017

This page was intetionally left blank



Internship Report

Master in Product Design Engineering

***Study on water-assisted injection molding
technology***

Name: Viveganandan Ravikumar

Internship developed under the supervision of Doctor Fabio Simões professor at the School of Technology and Management of the Polytechnic Institute of Leiria

Leiria, *June.2017*

This page was intentionally left blank

Acknowledgements

I would like to thank all professors of the mechanical engineering department of the School of Technology and Management and to all teachers from other departments for all the knowledge they have given me throughout my journey in this institution. I must thank my professor Dr. Fabio Simões for giving me an opportunity to gain exposure and knowledge about molds and to complete my report on my internship throughout out my course. I would also like to thank all the people from Moldetipo for helping me and passing over their knowledge to me.

This page was intetionally left blank

Resumo *(in Portuguese)*

Este relatório compreende uma apresentação e análise detalhada do estágio feito na Moldetipo. As áreas que são descritas e discutidas em detalhes incluem moldação por injeção, vários processos utilizados na fabricação de um molde, defeitos num molde e defeitos em peças injetadas. Também analisa a tecnologia de moldação por injeção assistida por água. Este relatório começa por explicar a moldação por injeção e as máquinas de moldação por injeção, seguidas de partes constituintes de um molde e os diferentes processos envolvidos na fabricação de um molde. Este relatório também compreende um estudo de caso de moldação por injeção assistida por água (WAIM) e a formação de vazios nas peças fabricadas por WAIM. Além disso, inclui um estudo de caso sobre diferentes parâmetros que afetam o processo WAIM.

Palavras-chave: Moldação por injeção, peças padrão, defeitos, moldação por injeção assistida por água.

This page was intetionally left blank

Abstract

This report comprises of a detailed presentation and analysis of the internship that is done at Moldetipo. The areas that are described and discussed in detail include, injection molding, various processes used in making a mold, defects in a mold and defects in part . It also analyses the technology of Water Assisted Injection Molding. This report starts by explaining injection molding and injection molding machines, followed by parts of a mold and different process involved in manufacturing a mold. This report also comprises of case study of Water Assisted Injection Molding (WAIM) and the void formation in the parts of WAIM machined. Also, a case study about different parameters that affect the WAIM process.

Keywords: Injection molding, Standard parts, Defects, Water Assisted Injection Molding.

This page was intentionally left blank

List of figures

Figure 1-Injection Molding machine[2].....	3
Figure 2- The Barrel & the zones.[4].....	4
Figure 3- Working cycle of an injection mold.[4]	5
Figure 4- Hydraulic clamp & Toggle clamp.[2]	6
Figure 5-The Deep Rib & Core Cavity approach.[7].....	7
Figure 6-Two-plate mold. (a) Closed & .(b) Opened.[2].....	8
Figure 7-Three-plate mold. (a) Closed & .(b) Opened.[2].....	8
Figure 8-Sleeve Ejection.[8]	9
Figure 9- Push back pin when the mold opens during ejection.[8].....	10
Figure 10- Stripper plate ejection representation.[8]	10
Figure 11- Hot runner for multiple cavity mold[8].....	11
Figure 12- Section view of a sprue[8].....	11
Figure 13- Cooling channel circuit in an example mold.[10]	12
Figure 14- Single cavity mold with feed system[8]	13
Figure 15- Multi-cavity mold with feed system.[8].....	13
Figure 16- An example mold with plates named.[11].....	14
Figure 17- Ejector set (Ejector plates)[11].....	14
Figure 18- Tapered interlocks and Straight side interlocks.[11].....	15
Figure 19- Round end ejector pin.[11].....	15
Figure 20- An example image of a Lifter [11].....	16
Figure 21- Slider in opened position & Slider in closed position.[11]	16
Figure 22- A standard hot runner system.[11]	17
Figure 23- Flat component.[11]	18
Figure 24-Spirial component.[11].....	18
Figure 25- Sink marks indicated in a part.[17]	22
Figure 26- Short shot indicating incomplete component.[17]	23
Figure 27- An image indicating the difference between the required part and the part that has warpage.[18].....	23
Figure 28- Burn mark identified in a part.[17].....	23
Figure 29- a part with flashes in it.[17].....	24
Figure 30- A sprue Gate.[19]	24

Figure 31- A Tab gate.[19]	25
Figure 32- An Edge gate.[19]	25
Figure 33- A Submarine gate.[19]	26
Figure 34- A common 3-Axis CNC machine.[23]	30
Figure 35- A steel bar before being machined as mold plates.[27]	31
Figure 36- Face milling operation.[24]	32
Figure 37- A schematic representation of EDM process .[25]	34
Figure 38- The overall setup of Electro Discharge Machining.[26]	35
Figure 39- A working Wire-cutting EDM machine.[27]	36
Figure 40- The process layout of a wire cutting EDM machine.[28]	36
Figure 41- The Laser Scanning device.[29]	37
Figure 42-Different stages in Gas-assisted injection molding.[31]	40
Figure 43- Effect of cooling in an External GAIM part.[31]	42
Figure 44- The steps for a short-shot WAIM.[32]	43
Figure 45- The steps for a Full-shot WAIM.[32]	44
Figure 46- PvT diagram of a typical semi-crystalline polymer.[33]	46
Figure 47- Void formation with heat flow inside a part.[33]	47
Figure 48- Varying Residual wall thickness in the curved section of a part.[33]	47
Figure 49-Water penetration on mold temperature.[34]	49
Figure 50- Water penetration on Delay time [34]	50
Figure 51-Effect plotted at 40°C.[33]	51
Figure 52- Effect plotted at holding time of 5sec.[33]	52
Figure 53-The Effect at holding time of 5sec.[33]	52
Figure 54- The Effect at holding time of 10sec.[33]	53

List of tables

Table 1- Commonly used material for mold and its applications.[15]	20
Table 2- Chemical composition of P20 steel.[15].....	20
Table 3- P20 Tool Steel Equivalent Steel Grades[15]	20
Table 4- Mold plate numbering and the common materials it is made of.	21
Table 5- Commonly used materials.[16] [17]	22
Table 6- Common Defects in injection molding.....	22
Table 7- The Melt Flow Rate of different plastic resin.[24]	27

This page was intentionally left blank

List of acronyms

CNC – Computer Numerical Control.

MFR – Melt Flow Rate.

GAIM – Gas Assisted Injection Molding

WAIM – Water Assisted Injection Molding.

PL – Parting Line

W – Width of the gate.

L – Length of the gate.

h – Height of the gate.

t – thickness of the gate.

This page was intetionally left blank

Table of Contents

ACKNOWLEDGEMENTS	III
RESUMO (IN PORTUGUESE)	V
ABSTRACT	VII
LIST OF FIGURES	IX
LIST OF TABLES	XI
LIST OF ACRONYMS	XIII
TABLE OF CONTENTS	XV
1. INTRODUCTION	1
2. INJECTION MOLDING	1
2.1. The injection mold.	1
2.2. The injection molding machine.	2
2.3. Design of an injection molding machine.	2
2.4. Injection Units.	5
2.5. Clamping.	6
3. MOLD CONSTRUCTION.	7
3.1. Two-plate mold.	7
3.2. Three-plate mold.	8
3.3. Ejection Techniques.	9
3.4. Feed system.	10
3.5. Cooling system.	11
3.6. Standard parts of a mold.	12
3.6.1. Single cavity mold.	12
3.6.2. Multi cavity mold.	13
3.6.3. Standard mold set with bars and sleeves	13
3.6.4. Ejector pins and special components for ejection.	15
3.6.5. Core moving elements.	16
3.6.6. Runner system components	16
3.6.7. Cascading flow components.	17

3.7.	Material selection of an injection mold.	18
3.8.	Defects in injection mold	22
3.9.	Gates.	24
3.10.	Critical factors for injection molding.	26
4.	ACTIVITIES DEVELOPED	29
4.1	CNC	30
4.2.	Milling	30
4.2.1.	End mill materials.	32
4.2.2.	Observations	33
4.3.	Production/Bancada.	33
4.4.	Erosion	34
4.5.	Injection department.	37
4.6.	Dimensional analysis stage.	37
5.	WATER ASSISTED INJECTION MOLDING (WAIM).	39
5.1.	Gas assisted injection molding.(GAIM).	39
5.2.	Water assisted injection molding.(WAIM).	42
5.3.	Water assisted injection molding at MoldeTipo	45
5.4.	Important parameters for WAIM.	48
6.	CONCLUSION.	55
	REFERENCES	57
	APPENDICES	62
	GLOSSARY	64

1. Introduction

This report comprises the information about the internship in company Moldetipo. Ltd. In this internship the guidance was given by professor Fabio Simões of Instituto Politécnico de Leiria. The internship is as a part of the course Master's in Product Design Engineering in IPL, and it has a duration of about nine months. During this nine months, it was possible to acquire information about injection molding and about getting customers. Along with this, it was possible to learn about a technology called "Water Assisted Injection Molding"(WAIM). So, this report includes a separate section about WAIM, void formation in WAIM and the parameters that affect for the betterment for the process to make a perfect part.

The internship began by the CNC department at the very beginning. Followed by Production, Erosion, Injection and Design. The duration in each department depends upon the time it takes to understand the main theme of the works being done in each department respectively. Despite being in each of the departments, the work was also done in the Dimensional control department at the same time. So, this report consists of a brief information about molds and molding machines at the beginning, after which it has the standard components used in a mold. Explaining all this is extended by my experience and knowledge that was gained during the period of internship. The report concludes with a study of Water Assisted-Injection Molding and a case study information about the parameters that affects the WAIM process.

2. Injection molding

Injection moulding is a technique of producing an object as the name itself infers, the object is produced by injecting its molten raw material. Which is then directed into a cavity, which will be shaped according to the shape of the product. Thereby, the product of the required shape and material is moulded by injection moulding. In Injection molding the polymer is heated to a highly plastic state and forced to flow under high pressure into a mold cavity, where it solidifies. The part is then removed from the cavity. The process produces discrete components. Also, there are multiple cavity molds which is used to produce multiple parts in a single cycle.

The objective here is manufacturing a mold. The cost too matters while designing a mold. It should be worth it and high in quality as most of them are for mass production.

2.1. The injection mold.

If plastic is injected into the mould cavity, this is called plastic injection moulding. These plastics are thermoplastics which, when heated at high temperatures melt and when cooled regain their solid state [1]. An injection mould is comprised of several elements.

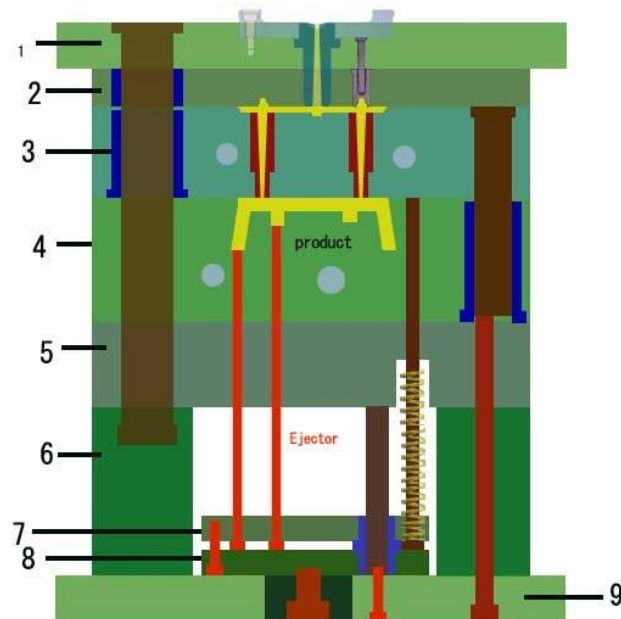


Figure 1- Injection mould elements.[2]

Where,

1. Top Clamping Plate
2. Runner plate
3. Fixed Cavity plate
4. Movable Core plate
5. Ejector Pin support plate
6. Spacer Block
7. Ejector Retainer plate
8. Ejector plate
9. Bottom Clamping Plate

Injection moulding is used to make many products such as wire spools, packaging, bottle caps, automotive parts and components, .etc.

2.2. The injection molding machine.

The injection machine is a machine that plasticizes the molding material inside the heating cylinder and injects this into the mold tool to create the molded product by solidifying inside it.

The injection machine is constructed of a mold clamping device that opens and closes the mold tool, and device that plasticizes and injects the molding material. There are several types in the injection machines, and the difference is made by how these two devices are arranged.

(1) **Horizontal injection machine** : Both mold clamping device and injection device compounded horizontally.

(2) **Vertical injection machine** : Both mold clamping device and injection device compounded vertically

2.3. Design of an injection molding machine.

The Injection machine will be of different types, upon which the type that we want depends upon the volume, clamping pressure, nozzle structure and injection mechanism.

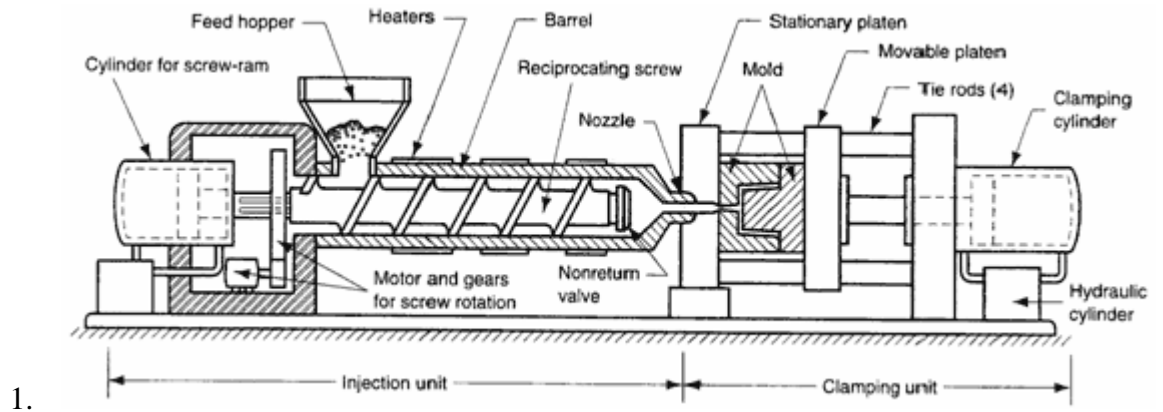


Figure 1-Injection Molding machine[2]

The machine has two units:

- **The clamping unit &**
- **The injection unit.**

The clamping unit has fewer parts than the injection unit. It comprises of a movable platen, a stationary platen, clamps to hold cavity and core with their respective platens, the hydraulic cylinder and the tie rods.[3]

The tie rods allow the movement of the platens, which holds the cavity and core. The hydraulic cylinder exerts required pressure on the platens and thereby the mold is closed while injection takes place.

The injection unit has got more parts than that of the clamping unit. It is the place where the plastic melting and pressurisation happens with the help of the hopper, barrel and the nozzle.

The hopper:

This is shaped like a cone & it is the place where the plastic pellets are poured after they are dried out of moisture completely. This directs the plastic into the barrel for the melting process.

The barrel

The barrel of an injection molding machine is where the plastic pellets are melted and sent to the mold, It also has the reciprocating plasticizing screw. The screw exerts pressure by crushing the pellets and the surrounding electric heater heats, this thereby melts the plastic into melt.

The reciprocating screw:

The reciprocating compresses and melts the plastic thereby conveying it to the mold. It has three zones.

- *the feeding zone*
- *the compressing zone*
- *the metering zone*

The plastic enters the feeding zone from the hopper, it then flows into the barrel which has screw. The screw has a diameter at the beginning and it varies throughout the barrel and the inner wall of the barrel remains in the same diameter. So, when the pellets move forward they are pressurised and also it flows against the inner wall of the barrel which produces a lot of heat due to the viscosity (shear heat) and this eventually melts the plastic pellets. The heaters outside help to maintain the melt in the molten state. A molding machine can have many heater bands or zones with different temperature settings depending on the requirement.

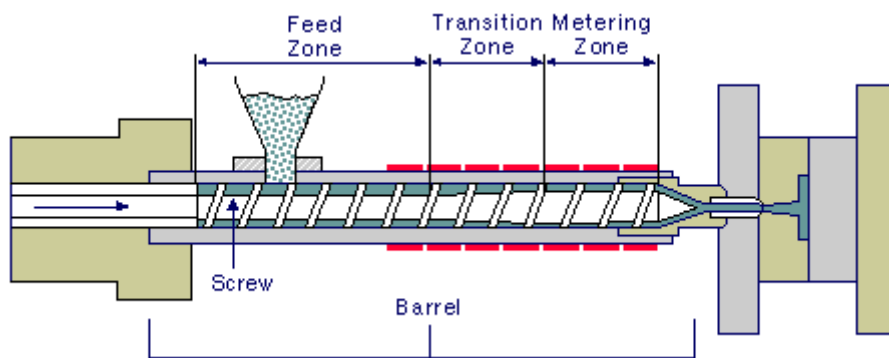


Figure 2- The Barrel & the zones.[4]

The working cycle proceeds in this way,

1. the mold is opened & the plastic that is required to be shaped into the required product is melted and kept in the nozzle.
2. Then, the Mold is closed and clamped.
3. Now, a shot of melt (molten plastic), which has been heated to the right temperature, is injected under high pressure into the mold cavity. The plastic then fills the cavity of the mold and begins to solidify as it gets cooled when it comes in contact with the cold surface of the mold.

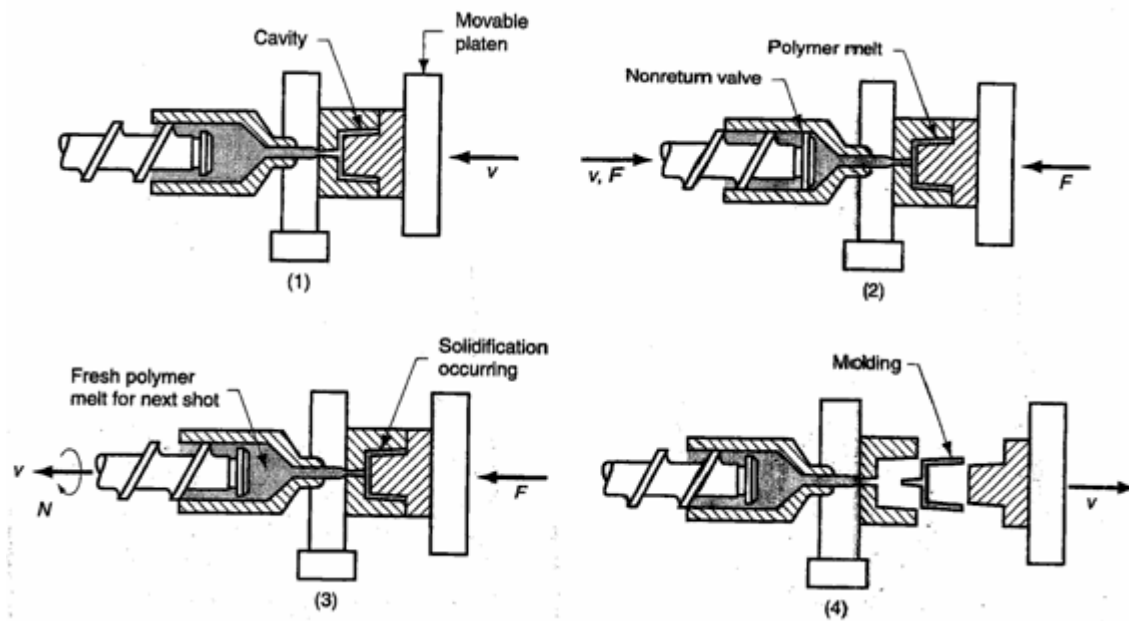


Figure 3- Working cycle of an injection mold.[4]

The cavity is filled in two stages.

At the first stage, the cavity of the mold is filled with high pressure and temperature. At the second stage, a little amount of plastic will be injected in order to make sure that the plastic completely touches the walls of the cavity which might be contracted due to the solidification that happens while cooling.

The screw is rotated and retracted with the non-return valve open to permit fresh polymer to flow into the forward portion of the barrel. Meanwhile, the polymer in the mold will be completely solidified.

At last, the mold will be opened and the part will be ejected with the ejecting pins.

2.4. Injection Units.

There are two types of injection units which are commonly used today: *Reciprocating Screw Type & Screw-pre-plasticizer Type*. [5]

Reciprocating screw type:

This type is the most common type that we use these days. It melts the plastic with the help of a reciprocating screw. The melting and the injection takes place in the same barrel in this type.

Screw-pre-plasticizer type:

In this type, the plastic is kept in a separate container and the injection takes place in another barrel for injecting a polymer. Because of which, this type is also called as two-stage machine.

2.5. Clamping.

There are basically three types of clamp designs, 'Hydraulic clamps', 'Toggle clamps' and 'Hydro-mechanical clamps'.

The type of clamp used in the machine decides the pressure that the machine can produce. Hydraulic clamps are the ones that are used in high tonnage machines. Its range varies from 1300 to 8900 kN..[5]

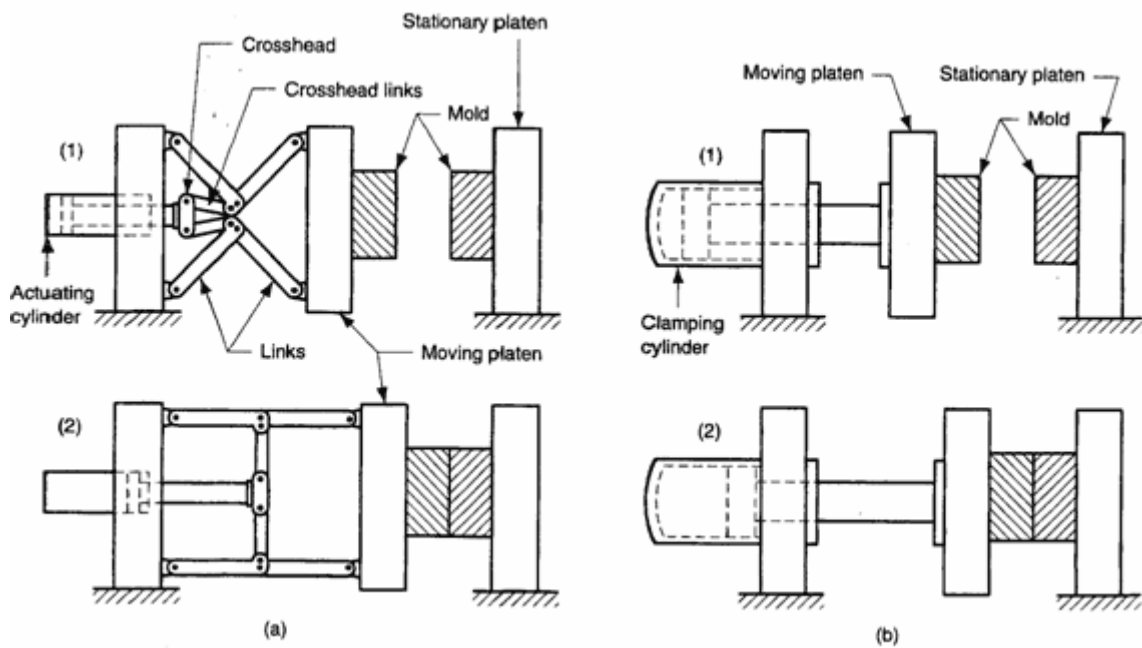


Figure 4- Hydraulic clamp & Toggle clamp.[2]

Also, there are 'hydromechanical clamps', they are used at places where the requirement is more than 8900 kN.

Toggle clamps works with cross-links connected to the piston. This is used to reduce the speed of closing and opening the mold. The presence of crosshead makes controlling the speed easy.

3. Mold Construction.

The mold has two major parts, the cavity and the core. The design of this cavity and core is purely based on the design of the part that we want.[6]

The designing is done based on two approaches.

- **The deep rib approach &**
- **The core-cavity approach.**

The second approach is the best approach and it is cheaper when compared to the former approach

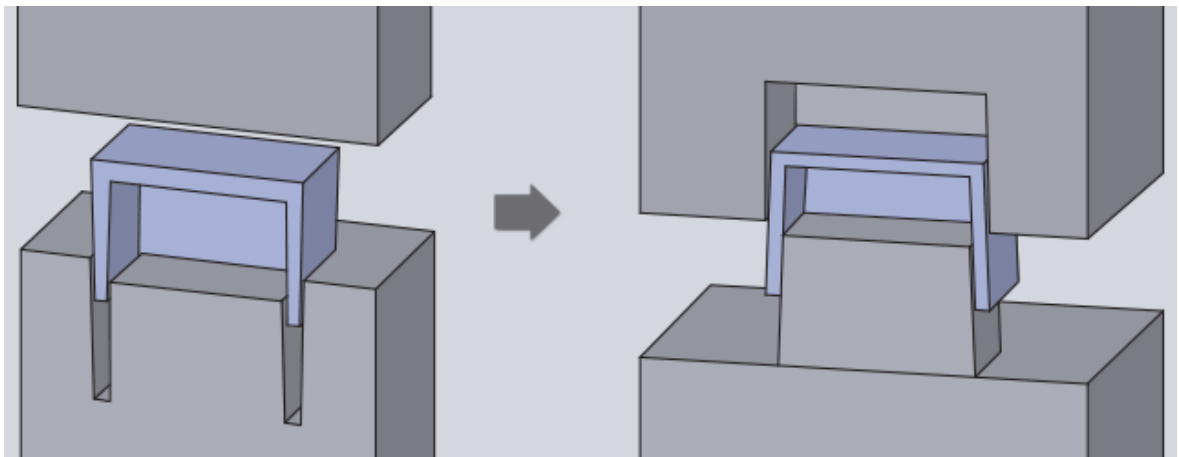


Figure 5-The Deep Rib & Core Cavity approach.[7]

Deep Rib Approach:

Box designed with walls as ribs. Higher cost to machine and polish cavity.

Core Cavity Approach:

Box designed through core-cavity method. could be more effective in many ways than the deep rib approach. The machining is easy. The construction is cost effective and the part removal is also easier than the other.

The molds are basically classified into two types:

- **Two-Plate mold.**
- **Three-Plate mold.**

3.1. Two-plate mold.

This type of mold has got two steel plates at the cavity part of the mold.

In a two plate mold, the plastic is directly injected into the cavity through a sprue. So, when the part is removed, it comes with the sprue attached with it, which should be removed later.

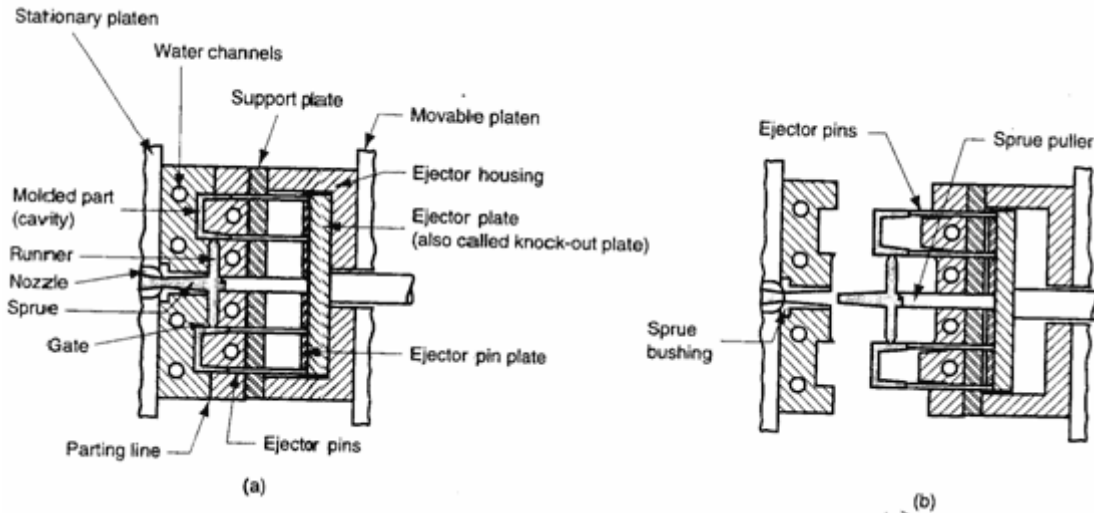


Figure 6-Two-plate mold. (a) Closed & .(b) Opened.[2]

3.2. Three-plate mold.

Here, in this type of mold, the plastic is injected through a runner system and through a gate, which makes it easy for the gate to be removed from the part and give a finished part at the earliest.

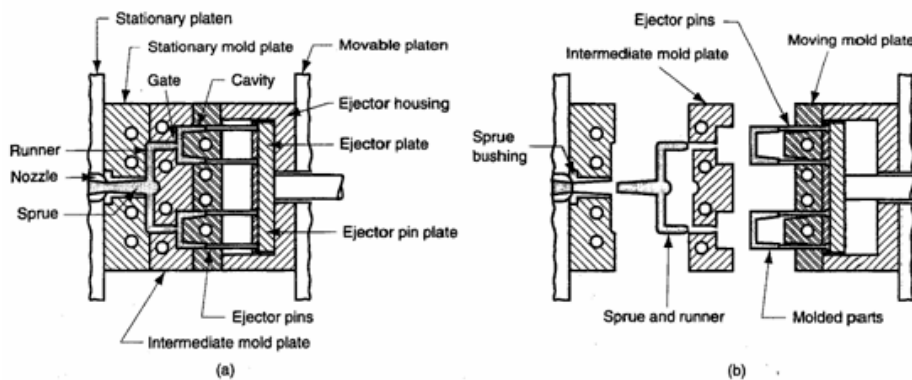


Figure 7-Three-plate mold. (a) Closed & .(b) Opened.[2]

The part removal is done by a separate plate. This plate will have the sprue and the runner which will come separately during removal. The part will be stuck in the moving

half of the mold due to the shrinkage that occurs while cooling and therefore, the part will be held there. On the other side, the separate plate removes the sprue and runner.

3.3. Ejection Techniques.

After the part is manufactured, it should be removed safely without causing any warpage or scratches to the part. For which, ejector pins are used, they are placed in core and in some cases cavity as well in such a way, that it makes the part removal easy and safe.

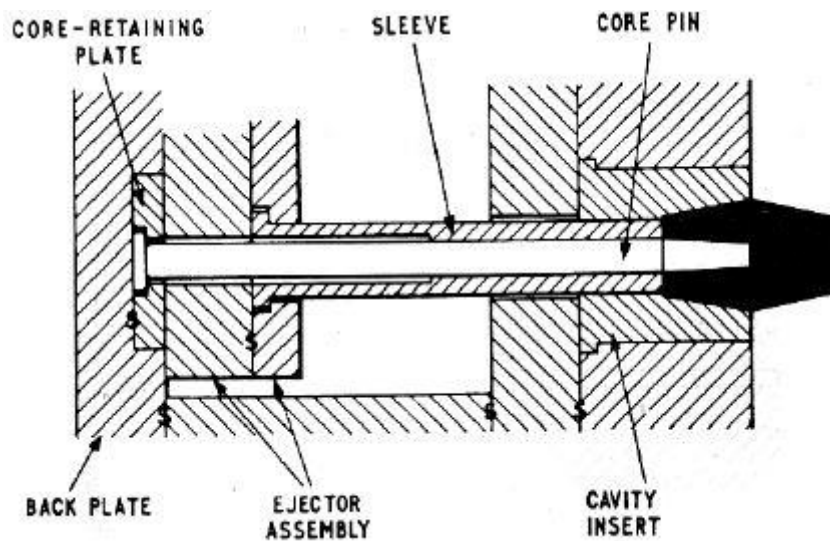


Figure 8-Sleeve Ejection.[8]

Push back pins

These pins are used mainly to make sure that the ejector pins are returned to its initial position when the mold is closed. It is placed at the bottom plate where the ejector pins are fixed. It is like a safety feature for ejector pins and gives assurity.

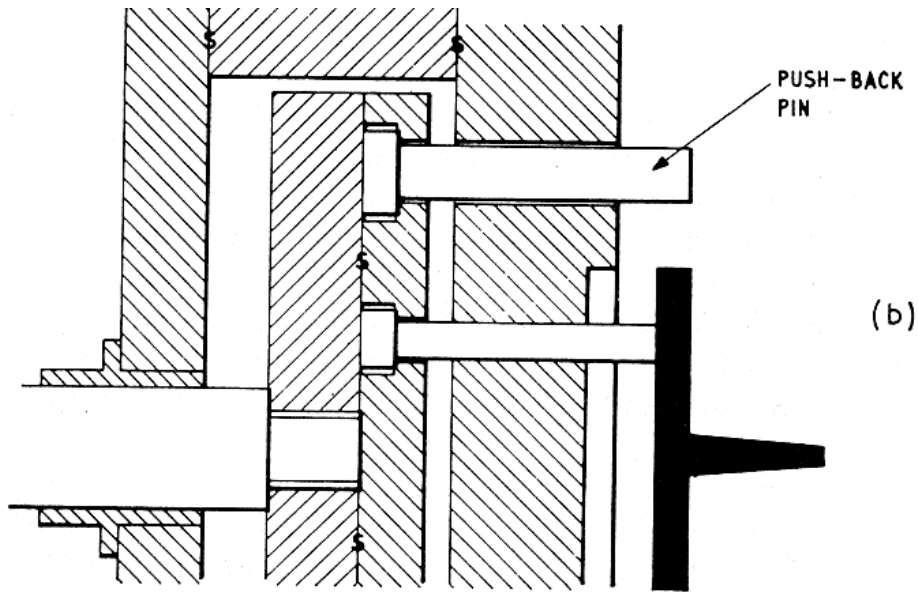


Figure 9- Push back pin when the mold opens during ejection.[8]

Stripper Plate Ejection

This type of ejection is used for simple parts, especially with the ones that have lower part thickness. The ejection is done entirely with the help of a plate which is used instead of ejection pins, as the part thickness is low and needs to be removed safely.

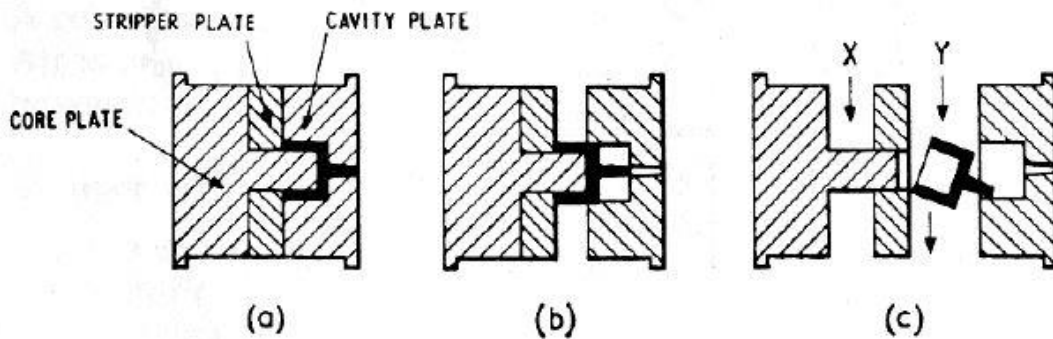


Figure 10- Stripper plate ejection representation.[8]

3.4. Feed system.

The feed system consists of the runner system, sprue and gate. The plastic when injected through the nozzle, enters the runner system at first and this helps to maintain the temperature of melt, which then directs it into the sprue followed by the gate to the part cavity.

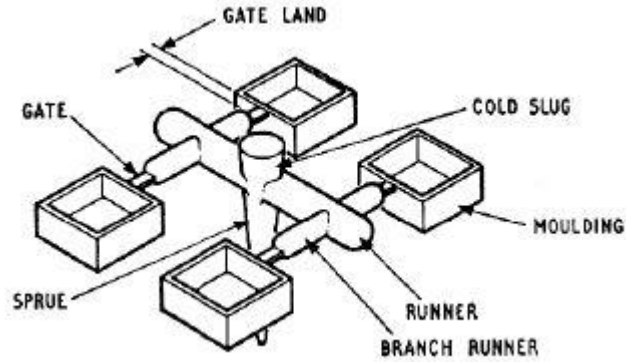


Figure 11- Hot runner for multiple cavity mold[8]

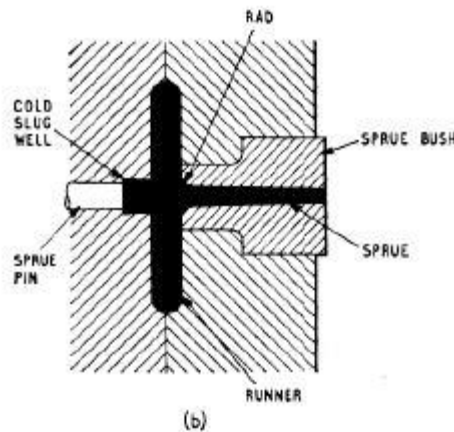


Figure 12- Section view of a sprue[8].

3.5. Cooling system.

A cooling system is a must for injection molds. It consists of an external pump connected to passageways in the mold, through which water is circulated inside the mold to remove heat from the hot plastic while molding happens. Air must be evacuated from the mold cavity as the plastic rushes inside. Most of the air passes through the small ejector pin clearances in the mold.[9]

In addition to these, air vents are also provided, which are engraved into the parting surface; their dimensions are about 0.03 mm (0.001 in.) deep and 12 to 25 mm (0.5 to 1.0 in.) wide depending on the requirement, these channels permit only air to escape outside but they are too small for the viscous polymer melt to flow through.

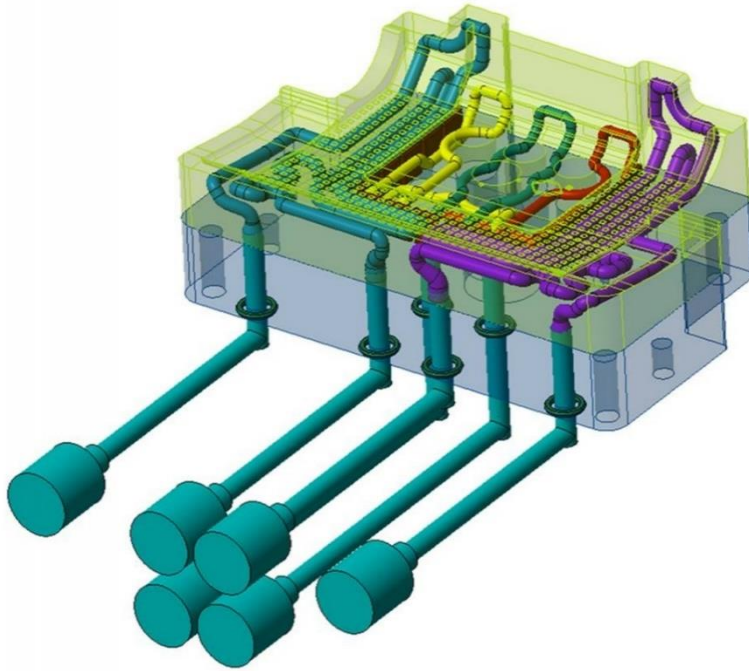


Figure 13- Cooling channel circuit in an example mold.[10]

Usually, the air vents are made in the area close to the molding area. Or, sometimes near the adjustment zone. which makes it easy for the air to get out. If the air venting is not done completely, the part will have burn marks in it due to the accumulation of air at ascertain point, the pressure and the heat at that point gets increased and thereby, it burns the part.

3.6. Standard parts of a mold.

The standard parts of mold includes,[11]

- Guide bars, guide sleeves and other guiding elements.
- Ejector pins and special components for ejection
- Core pins
- Core moving elements
- Runner systems

3.6.1. Single cavity mold.

Single cavity molds are used where the component required is of very unique geometry. In single cavity molds, the mold will be just a little bigger than the size of the component.

So, before designing the mold for single cavity molds, we have to design the size of the mold.

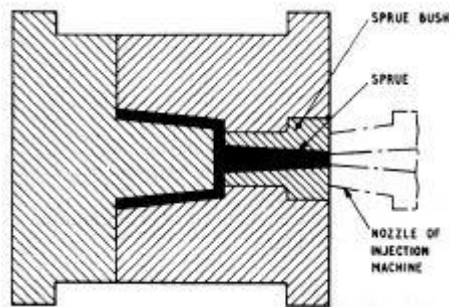


Figure 14- Single cavity mold with feed system[8]

3.6.2. Multi cavity mold.

In Multi cavity molds we have multiple cavities. Basically, this type of mold is created if the component required is of pair and mirrored components.

This mold is economical as we can produce more than one part at the same time.

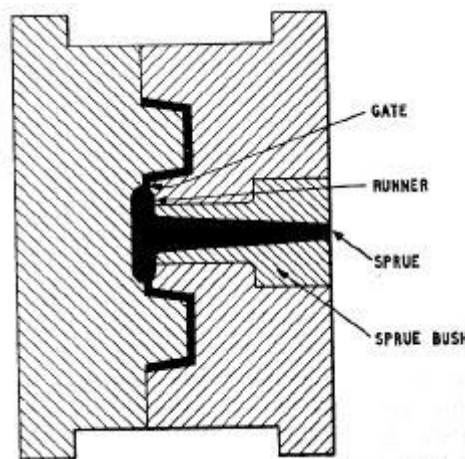


Figure 15- Multi-cavity mold with feed system.[8]

3.6.3. Standard mold set with bars and sleeves

The standard mold set consists of top and the bottom plates, the core and the cavity and the ejection plates.

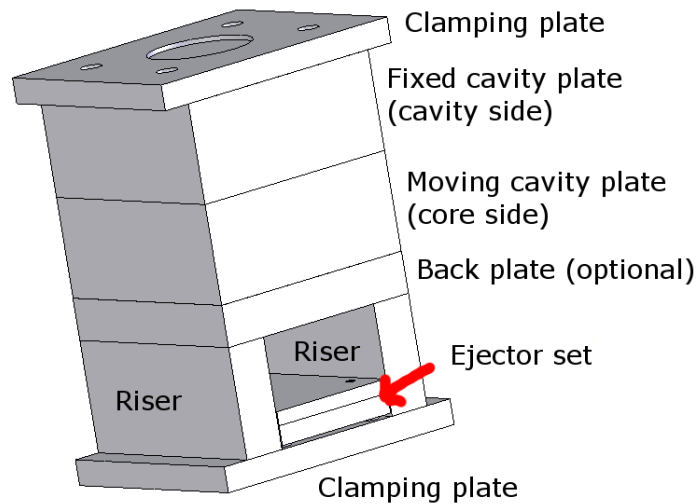


Figure 16- An example mold with plates named.[11]

Ejector set:

There are two plates in the ejector set, one plate is to provide a common base and limit to the pin which is the retaining plate and the other is the ejector plate, that helps to move the pins. There are risers, which restricts the ejector travelling distance. The buffer plates are placed to avoid vacuum creation at the bottom ,making it easier for the ejector plates to move.

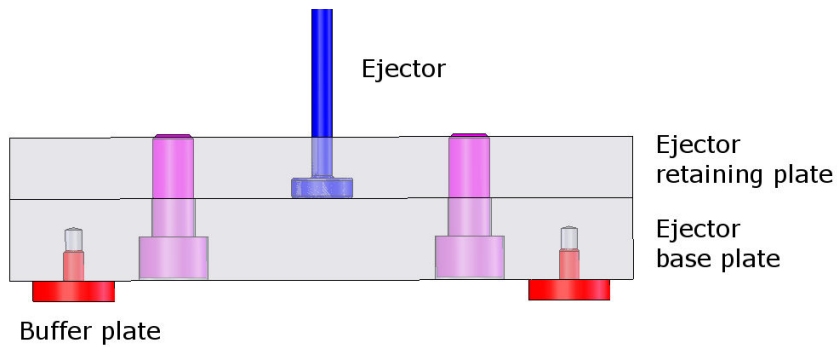


Figure 17- Ejector set (Ejector plates)[11]

Basic guiding elements:

There are some guiding elements which guides the core and cavity to close perfectly during the process of molding. These are usually like supports to the core and cavity. This includes guide pillars, guide sleeve and cavity interlocks..[5]

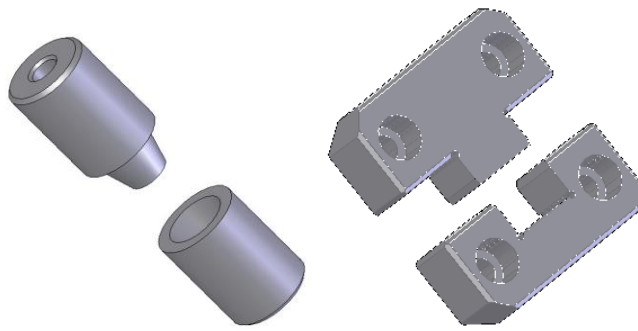


Figure 18- Tapered interlocks and Straight side interlocks.[11]

3.6.4. Ejector pins and special components for ejection.

The basic components of ejection includes :

- *Round ejection pin*
- *Flat end ejection pin and*
- *Lifters.*

There is another thing called Sprue pullers which are used to pull the sprue attached to the part.

The lifters are something that is used to eject the part and also assist to make undercuts easily. They are usually placed as a part of core to make undercuts and also acts as a ejection pin to eject the part after being molded.

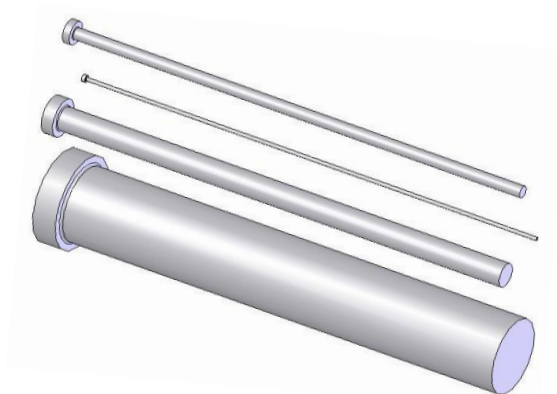


Figure 19- Round end ejector pin.[11]

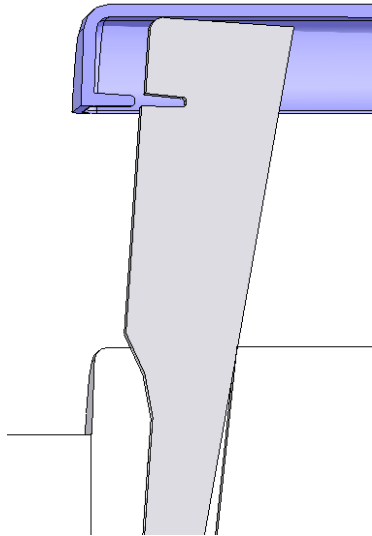


Figure 20- An example image of a Lifter [11]

3.6.5. Core moving elements.

Slide mechanism and standard components:

There are moving elements in core as well. The most common one includes slider. It is used in places where there are negative shapes and undercuts to the part required. This is basically operated through hydraulic cylinders.

It works like, when the cavity with the guide bars moves downwards. The slider or other component that is on the path of guide pin, moves with it. Thereby, when the guide pin goes downwards or upwards, the slider closes and opens respectively.

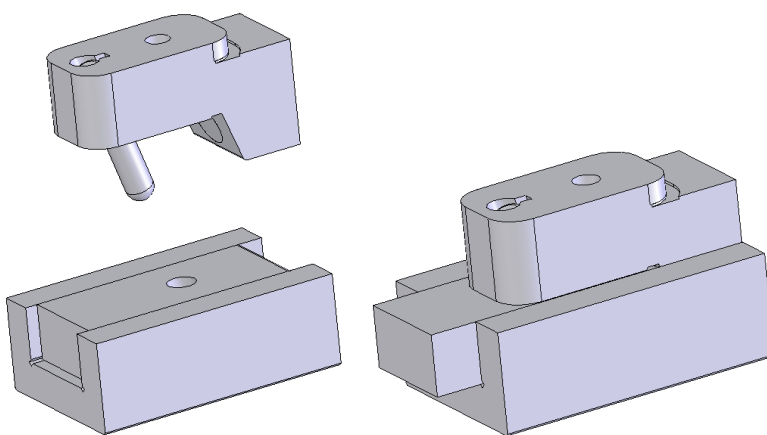


Figure 21- Slider in opened position & Slider in closed position.[11]

3.6.6. Runner system components

Cold runner:

The need of a runner system is to create the melted plastic a specified path from the nozzle to the mould cavity. In cold runner system, the plastic solidifies inside it and therefore, it is removed along with the part and has to be detached separately..

Hot runner:

The sprue and runner in the common two-plate or three-plate mold are mostly a waste material. In many cases they can be grounded and reused; however, in some instances the product must be made of “virgin” plastic (that which has not been previously molded). In such case, The hot-runner mold eliminates the solidification of the sprue and runner by locating heaters around the corresponding runner channels. While the plastic in the mold cavity solidifies, the material in the sprue and runner channels remains molten, ready to be injected into the cavity in the next cycle.

The hot runner system has their heating components inside in most of the cases. The heat is maintained in such a temperature without affecting the melt quality. The position of heating channels describe the hot runner type that is used.

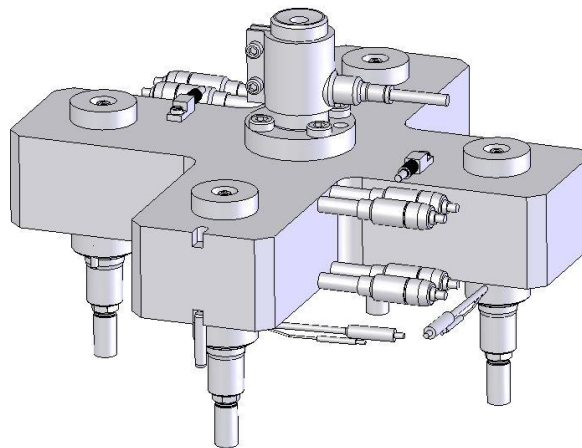


Figure 22- A standard hot runner system.[11]

The runner system here, is completely controlled with a heating system in the hot runner. There are heating channels provided surrounding the sprues of the hot runner. They are controlled according to the needs of the process.

3.6.7. Cascading flow components.

There are parts to direct the liquid with an aim to make the part cooling effectively. These parts include cascading liquid junctions, cooling baffles and cooling cores. These

elements are placed inside the mold. These channels are drilled according to their requirements using deep drilling machines.



Figure 23- Flat component.[11]



Figure 24- Spiral component.[11]

3.7. Material selection of an injection mold.

There is a wide range of materials selected for making of an injection mould . The material is chosen based on these parameters,[12]

- **Part design.**
- **Tool design.**
- **Number of Parts.**
- **Expected Lifetime.**

The materials used are of many types and the common ones include Pre-Hardened steel and Hardened steel.

Hardening is a process of making steel harder than before and improving its mechanical properties.

There are several other processes involved in making the steel better in its properties that before. It depends on the requirement.

They are :

1. **Annealing**

- 2. **Normalizing**
- 3. **Hardening &**
- 4. **Tempering.**

1. Annealing:

Heating the steel to a defined temperature and cooling it in conditioned atmosphere like in furnace. properties like ductility, softness, toughness, fine grain size and machinability. This reduces the hardness.

2. Normalizing:

Heating the steel to above austenitic range or critical temperature and holding it for a particular period of time to let the transformation occur and after air cooled. It makes uniform carbide size, which enables heat treatment process later to produce more uniform final product.

3. Hardening:

After heating the metal, the material is cooled in a medium, i.e having drastic change in temperature. Here during this process, we are not allowing the grains to grow after nucleation resulting in very fine grains. This fast cooling is called quenching.

Quenching can be done with different medium, such as water quench, brine solution quench, oil quench. Quench severity changes with change in medium.

4. Tempering:

Tempering is a process of heat treating, which is used to increase the toughness of iron-based alloys. Tempering is usually performed after hardening, to reduce some of the excess hardness, and is done by heating the metal to some temperature below the critical point for a certain period of time, then allowing it to cool in still air. increase ductility and softness, cutting properties and magnetic properties.

Commonly used material includes H-13 ,P-20 ,S50C ,718 steel. [13]

Material	Applications
H-13 steel	Post hardened steel for high volume molds.
Stainless Steel	Post hardened stainless steel for high volume molds

P-20 Steel	Pre hardened tool steel for middle volume molds.
Nak 80 Steel	Pre hardened tool steel for middle volume molds
S50c Steel	Tool steel for low volume molds
718 Steel	Tool steel for low volume molds.

Table 1- Commonly used material for mold and its applications.[14]

Here, the most commonly seen material is P20. Chromium and nickel are the main alloying elements found in group P steels. P20 tool steels are nitrided or carburized. These steels are capable of being machined into complex and large dies and molds. P20 steels are mostly used in the carburized condition. The presence of chromium and nickel enhances the toughness and hardness of P20 steels.

Substance	Content
C	0.28-0.40
Mg	0.60-1.00
Si	0.20-0.80
Cr	1.40-2.00
Mo	0.30-0.55
Cu	0.25
P	0.03
S	0.03

Table 2- Chemical composition of P20 steel.[14]

P20 Tool Steel Equivalent Steel Grades:

Country	Standard	Grades
USA	ASTM A681	P 20
Germany (Most of Europe)	DIN EN ISO 4957	1.2311
China	GB/T 1299	3 Cr Mo

Table 3- P20 Tool Steel Equivalent Steel Grades[14]

Here, in Portugal we use `DIN EN ISO 4957` standard.

Commonly used steel :

The company usually buy steel plates from ``HASCO``.

The HASCO has a Catalogue in three categories. `K` , `P` and `Z`

Where **K** – Large Steel plates like Top plate, Bottom plate and Intermediate Plates.

P – This consists of the steel inserts , Round parts, Machined plates.

Z – This set consists of Guide Pins, Ejection Pins , Measurement systems
Temperature system , ..etc

PLATE or PART NUMBER.	MATERIALS USED.
1 and 9	1.1730 , 1.2083 ,1.2085 ,1.2311 ,1.2312
2 ,3 and 4	1.1730 ,1.203 ,1.2085 ,1.2162 ,1.2311 ,1.2312 ,1.2767
5	1.1730 ,1.203 ,1.2085 ,1.2311 ,1.2312 ,1.2343 ,1.2436
6	1.1730 ,1.2085 ,1.2162 ,1.2311 ,1.2343 ,1.2436
7 and 8	1.1730 ,1.2085
Ejector Pin (Round)	1.2516
Ejector Pin (Flat)	1.2343
Ejector Sleeve	1.2343

Table 4- Mold plate numbering and the common materials it is made of.

These are the materials that is used for commonly for the manufacturing of an injection mould.

Material	Heat treatment	Properties
1.1730	<u>Soft annealing:</u>	<ul style="list-style-type: none"> • <i>Temperature 680 -.710°C</i> • <i>Furnace cooled</i> • <i>Maximum Hardness is207 HB</i>
	<u>Stress relief annealing:</u>	<ul style="list-style-type: none"> • <i>Temperature 600 - 650°C</i> • <i>Furnace cooled</i>
	<u>Hardening</u>	<ul style="list-style-type: none"> • <i>Temperature 800 -850°C</i> • <i>Water cooled</i>
1.2311	<u>Soft annealing:</u>	<ul style="list-style-type: none"> • <i>Temperature – 710-740°C</i> • <i>Furnace cooled</i> • <i>Maximum Hardness is235 HB</i>

	<u>Stress relief annealing :</u>	<ul style="list-style-type: none"> •Temperature – 500-550°C •Furnace cooled
	<u>Hardening :</u>	<ul style="list-style-type: none"> •Temperature – 830 -870°C •Oil or Hot bath cooling – 180-220°C
1.2312	<u>Soft annealing :</u>	<ul style="list-style-type: none"> •Temperature – 710-740°C •Furnace cooled •Maximum Hardness is 235 HB
	<u>Stress relief annealing :</u>	<ul style="list-style-type: none"> •Temperature – 500-550°C •Furnace cooling
	<u>Hardening:</u>	<ul style="list-style-type: none"> •Temperature – 830 -870°C •Oil or Hot bath cooling –180 -220°C

Table 5-Commonly used materials.[15] [16]

3.8. Defects in injection mold

This section comprises of an important aspect that makes the injection molded part, more better and reliable. As the part comes out from the machine at the beginning, there will be definitely some defects, where some are ignorable and some should be sorted out depending upon the customer requirement. The major defects that could be seen and their possible remedies is prescribed below.

Table 6- Common Defects in injection molding.


DEFECTS	BRIEF.
SINK MARKS: 	<p>The cooling of an injected part is very important during the process. These sink marks occurs due to uneven cooling of the part, which happens often at thicker areas of the part. The inner area gets cooled first thereby starts shrinking, which appears to be a inner bulge.</p> <p>Uniform cooling and increasing holding time can solve this problem.</p>
SHORT SHOT:	<p>The melt volume to be injected per shot should be slightly more than that of the volume of cavity so that the part would</p>

Figure 25- Sink marks indicated in a part.[17]

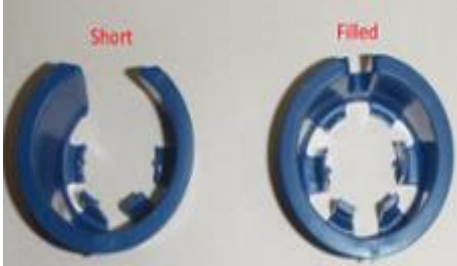
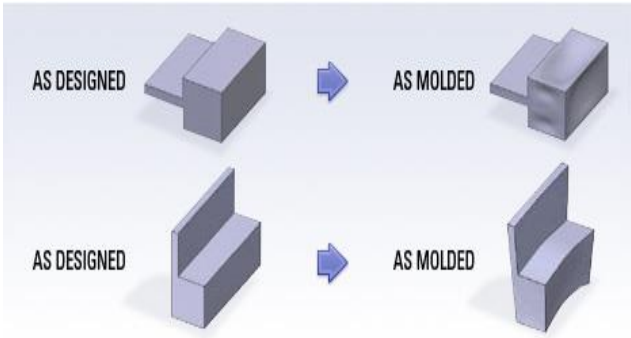
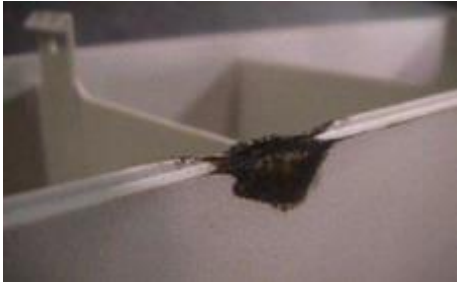
 <p>Figure 26- Short shot indicating incomplete component.[17]</p>	<p>be complete. It also occurs, if the melt viscosity is more and not enough heating is provided.</p> <p>Increasing mold & melt temperature and melt pressure increasing during injection will reduce this issue.</p>
<p>WARPAGE:</p>  <p>Figure 27- An image indicating the difference between the required part and the part that has warpage.[18]</p>	<p>Lengthy parts would often experience warpage at the ends. Parts that are not long will also have warpage in it but it is solvable.</p> <p>Providing supports in weaker part sections in the form of bosses and gussets tend to decrease warpage to a greater extent.</p>
<p>BURN MARKS:</p>  <p>Figure 28- Burn mark identified in a part.[17]</p>	<p>When the mold doesn't have enough venting for the air to escape during the process of injection, the air gets accumulated at a certain point. Which in turn increases pressure and heat and as a result, it burns the part at that specific area.</p> <p>Providing sufficient air vents to easy venting would decrease this problem.</p>
<p>FLASH:</p>	<p>When the plastic flows out of the cavity that is intended for the creation of part, there are extra unwanted attachments to the part which is called flash. This usually occurs if there is no complete closing of the core and cavity or any</p>



Figure 29- a part with flashes in it.[17]

damage in the adjustment zone of the mold.

By reducing the injection pressure and melt temperature, flashes can be avoided.

3.9. Gates.

A Gate in injection molding is an opening through which the molten plastic can enter into the cavity and form the part. The size of the gate , location and type of the gate have many effects on the quality of the part that is manufactured.

COMMON TYPES OF GATES:

DIRECT or SPRUE GATE:

As the name says, this type of gate is one common type that is used most commonly in many molds where gate marks in part is acceptable. This gate is directly provided into the cavity at the thicker section of the part

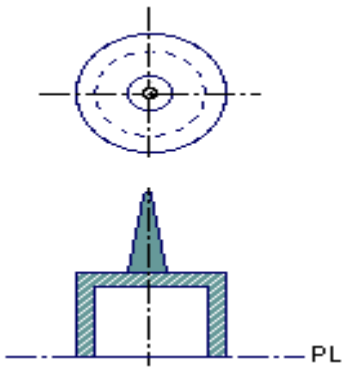


Figure 30- A sprue Gate.[19]

The advantage here is that, the injection pressure is less and the feeding time as well. whereas, the disadvantage is the gate mark, which will be left in the part when we cut the gate off from the part.

TAB GATE:

This type of gate has been used parts with large area and less thickness as these type of parts requires uniform flow of melt and that too without jetting. Tab gate is made out of the requirement, it is sometimes spread over the whole shorter side of the part ensuring uniform flow of melt.

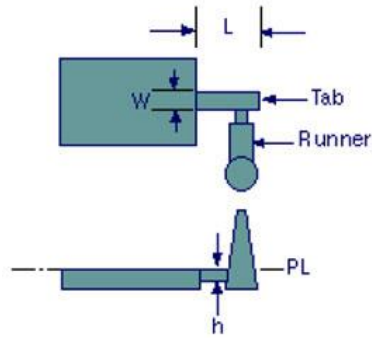


Figure 31- A Tab gate.[19]

EDGE or STANDARD GATE:

This is so similar to that of the previous type of gate, the only difference here is that , the gate will be provided at an edge of the part. It might be from the sides or perpendicular to the part edge as well.

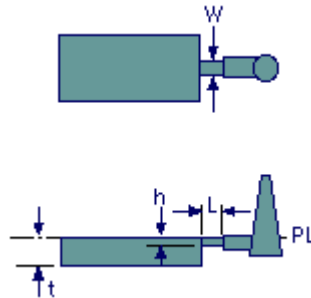


Figure 32- An Edge gate.[19]

These gates are also used for parts with larger area and thin wall

SUBMARINE GATE:

This is one of the mostly used gates in two plate molds. This gate will be machined in the mold in such a way that it will be tapered into the cavity below the parting line.

The runner will be connected to the cavity in an inclined angle inside the metal part before joining the cavity. So, as the part is ejected the gate will automatically be detached and it stays in there.

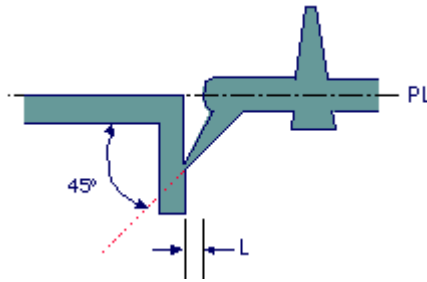


Figure 33- A Submarine gate.[19]

3.10. Critical factors for injection molding.

This chapter discusses about the key parameters that takes part in the process of creating a perfect molded part in injection molding. It is always safe to ensure that these factors are set optimum ensuring a good quality final product. The factors include:[20][21]

- 1) Barrel Temperatures
- 2) Plastic Flow Rates
- 3) Plastic Pressure or "Screw Back" Pressure
- 4) Nozzle Temperatures
- 5) Plastic Cooling Rates and Times
- 6) Plastic Melt Temperatures

1)Barrel and Nozzle Temperatures:

The main sources of heat that is found in injection molding includes the barrel, nozzle and the mold. Each of these has its part in deciding the flowability of plastic inside the mold in the process of achieving a perfect. The nozzle & barrel has got a major role in heating the melt.

This is the reason why temperatures are set between melting points and thermo-decomposition temperatures.

If this temperature is so high than required. Then, it results in over melting of plastics resulting in flash due to very low viscosity.

If the temperature is so less. Then, it results in lines of plastic flown over the part and improper filling like short shot.

2) Plastic Flow Rates

The flow rates for plastic injection represents the ease of plastic melt to flow inside the mold to fill it.

The flow rate determines the viscosity of the plastic by the time it flows inside the mold cavity. In such a case, if the holding pressures is too high, it creates flash and overflow of melt and if the pressure is too low, the melt flow doesn't exert pressure to vent air outside the cavity resulting in void formation.

The plastic flow is good if the melt flow rate (MFR) is high and is bad if the MFR is low.

Name of the Plastic resin	Melt Flow Rate
Polypropylene	5 to 60
Polystyrene	5 to 25
AS resin	10 to 40
ABS resin	5 to 60
Acrylic resin	1 to 35
Polyamide 6	10 to 80
Polyamide 66	10 to 80
Polyacetal	3 to 70
Polycarbonate	2 to 30
PBT	20 to 100
PPO	7 to 30
PSU	7 to 18
PES	17 to 30
Cyclo olefin polymer	7 to 60
TPX	21 to 80
PEEK	1 to 20

Table 7- The Melt Flow Rate of different plastic resin.[22]

3) "Screw Back" or Plastic Pressure

Screw pressure is nothing but the pressure created due to the feeding screw. It is created between the screw and the plastic when the cavity is filled and the pressure is exerted back onto the screw by the plastic. It is normally controlled by valve.

If the injection pressure is high, it creates problems like overflowing and flash and when the pressure is too low, voids might be formed.

4) Nozzle Temperatures:

The temperature of the nozzle should be lower than that of the temperature of the barrel. By this, the plastic inside the nozzle remains in a good quality. If not, the plastic gets decomposed.

Too high temperatures, makes some plastic to flow outside even if the nozzle is closed and if there is not enough heat, melt solidifies and blocks the nozzle and resulting in bad quality parts produced.

5) Melt and Mold cooling:

The range of cooling usually depends on the temperature of melt and the mold. Temperature of the mold depends on the temperature that the client recommended and the temperature that has to be used for the material used in the mold.

The dependency of cooling and timing is that, it depends largely on the client's expectation. Also , on the time required to produce perfect shaped part and perfect finish

6) Melt Temperatures:

To attain the perfect melt temperature is one important factor to be considered. Because, the melt to be injected inside the cavity should be at the perfect temperature to make it completely flow inside the mold and make the perfect part. It could be achieved by adjusting the temperatures of nozzle, barrel and the mold.

4. Activities developed

I did my internship at Moldetipo, a leading molding company in Portugal. They manufacture excellent molds and provide it to their customers.

Here in this section, the process cycle of an injection mold which is studied practically during the period of internship is about to be discussed in detail

Each section will be elaborated in a sequence how an injection mold is made. The operations that took place at each section with the observations are described

4.1. CNC

At first when I started my internship, I was in the CNC section in the beginning. The CNC section is where, the mold starts getting manufactured. The major carving of the mold cavity and the core is completely done in the CNC.

Let's first talk about the basics of what a CNC machine is and the commonly seen types.

CNC (Computer Numerical Control) is a machining is a type of machine that is most commonly used in molding industry to machine molds. The CNC machines are of many types and depending upon our requirement, the type is chosen.

The most common type of CNC machine is 3-axis machine. Which has movement or translation in X,Y and Z directions.



Figure 34- A common 3-Axis CNC machine.[23]

The CNC machines of 5-axis and 6-axis have a lot more applications than 3-axis machines. But, at the same time, the cost is very much higher when compared to other less axis machines.

In molding industry, the common operations done in a CNC machine is Milling, Drilling and Threading.

Now, let's look at the look at the different operations and tools used to machine a mold.

4.2. Milling

The first and the foremost process commonly done in a steel plate when it arrives is the milling. This is done based on the design in the design department.

Milling is a process by which a tool contacts a workpiece and removes material from the workpiece. The tool moves in almost every direction. So, when the path is specified to the machine, the tool travels in the specified path and removes material in the direction. The removal of material is achieved by the tool which rotates at a certain RPM. The RPM depends upon the finishing of the part.

The **drilling** is done in the direction of the axis of the spindle. Whereas, the milling is done in the other direction or along the surface of the part where the material is to be removed.

Milling produces 3-dimensional shapes with a rotating multi-edge cutting tool. The cutting tool can be programmed to move against a fixed workpiece in almost any direction. Milling tools remove material by their movement in the machine and from their shape.

Face milling is the first operation ,that is done to the steel bar. This is the operation that removes the surface of the mold according to design of the mold.

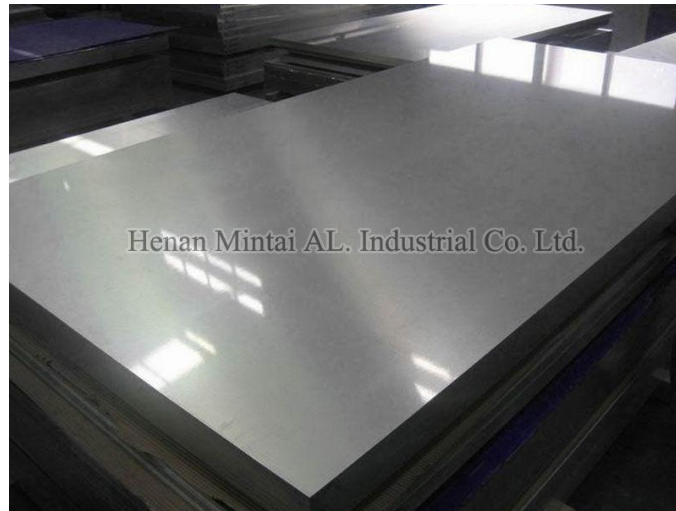


Figure 35- A steel bar before being machined as mold plates.[27]

The steel plate arrives like this, on which the machining is done at first to remove the unnecessary.

Face milling:

Face milling is a common type of milling operation used often to cut shallow in flat surfaces. It is usually used in large surfaces.

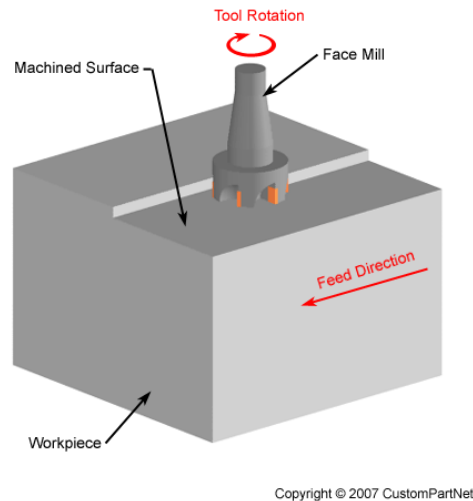


Figure 36- Face milling operation.[24]

A face mill is a tool that holds multiple carbide inserts in its cutting edge, which cuts the part. The more the inserts are, the faster the metal removal rate is. Cutters which are inclined 45° towards the part, or having that lead angle are used most frequently.

4.2.1. End mill materials.

End mills are made out of either cobalt steel alloys (known as high speed steel, or HSS), or from tungsten carbide in a cobalt lattice (shortened to "carbide").

High Speed Steel (HSS): Provides good wear resistance and costs less than cobalt or carbide end mills. HSS is used for general purpose milling of both ferrous and non-ferrous materials. While usually inexpensive, HSS does not offer the tool life or speed advantages of cobalt and carbide end mills.

Cobalt: Cobalt is also used as an end milling tool in the form of steel with an 8% cobalt content. Cobalt is one of the naturally occurring magnetic metals, because of which it is more expensive but provides better wear resistance and toughness than HSS. Because the tool can run 10% faster than HSS, metal removal rates and finish are better than HSS.

Solid Carbide: Carbide is considerably harder, more rigid, and more wear resistant than HSS. However, carbide is brittle and tends to chip instead of wear. Carbide is used primarily in finishing applications. Carbide tools are best suited for shops operating newer milling machines or machines with minimal spindle wear. Rigidity is critical when using carbide tools. Carbide end mills may require a premium price over the cobalt end mills, but they can also be run at speeds $2 \frac{1}{2}$ times faster than HSS end mills.

The choice of tool material depends on the material to be cut as well as on the maximum spindle speed of the machine. Smaller milling machines may not be capable of reaching the spindle speeds recommended for carbide end mills.

4.2.2. Observations

The major operations and processes of a CNC machine has been understood at the period of being there in the CNC department. The major issues and their remedies which contributes mostly to the betterment of the mold is analysed to an extent in the internship.

CNC Program

The CNC programs were basically controlled by a numerical controller. The company where the internship is done uses numerical controllers provided by FANUC for its CNC machines.

At first, the program is constructed in a simulation software to check the result of it. and if everything seems okay, then it is fed into the CNC machine. After which follows the program and makes the machining.

The main considerations that needs to be addressed are, machining a perfect 90° surface at the bottom of two planes intersecting each other is a task to be accomplished. To machine a perfect 90° surface in CNC is really a difficult task and also, if we have to machine a part like that, then it's not going to allow the other part to go inside and come out easily. There will be complete friction between the two surfaces and thereby creates wear and material removal.

The adjustment zone and molding zone surfaces are of different finish from the remaining area's finish. The reason being, it is the place where the plastic is molded. Since, the plastic should spread around the complete cavity without any lag. It is so important to machine the molding zone to machine differently in a finely polished surface with friction for the part to stick.

Drilling perfect water circuit for cooling is another important aspect to be considered. Which is done by deep drilling machines. These machines have lengthy drill pits, with which they drill the holes for water circuits.

4.3. Production/Bancada.

The mold after getting machined in CNC then goes to the production bench. Where it gets adjusted and then all the inserts that need to be put in there are inserted.

Operations performed at this stage:

- The water circuit machined is checked with water pump. And if there is any leakage are merge between two different lines .
- The lines are connected with its respected connectors.
- Pressure plug is used to block the line at a certain point.
- Baffles are used to divert the coolant in a specific direction to prolong the period of coolant to flow in the molding area cooling the plastic.
- The adjustments at the molding zone are done. These includes, making sharp edges blunt and inserting inserts.

4.4. Erosion

The mold after finishing its initial role in production, it is then sent to erosion. This is a department where, the mold will be machined in such a way that, the metal from the mold is met with high voltage electric sparks from a machine through an electrode, which then removes the material though erosion.

The machine used for is called **EDM** (Electric Discharge Machining).

Electric Discharge Machining:

The working of the machine begins when the electrode comes closer to the workpiece, an electric spark is produced which melts the material. This spark is created by high voltage and it is of very high temperature, about 8000 to 12500 degree Celsius.

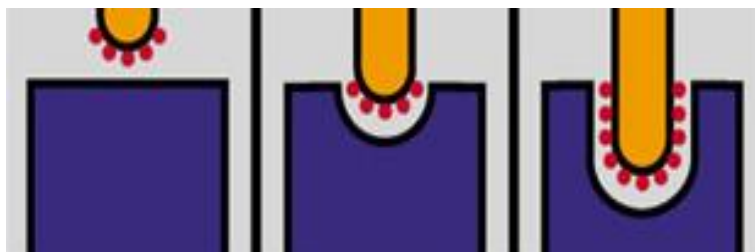


Figure 37- A schematic representation of EDM process .[25]

So, when this spark melts the metal and erodes it, this gets settled down at the bottom of the machining space. The workpiece is immersed in a dielectric fluid and so, the entire process happens immersed in the dielectric fluid

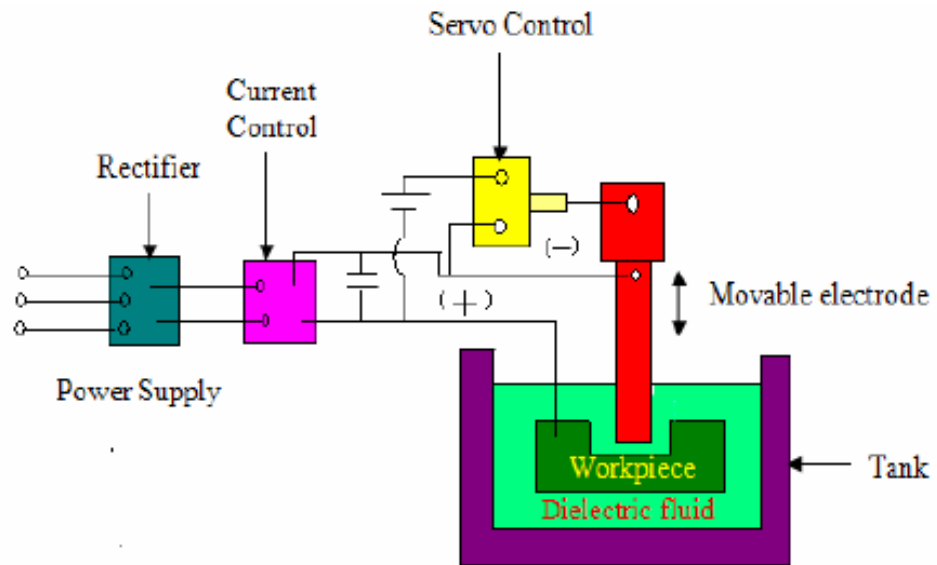


Figure 38- The overall setup of Electro Discharge Machining.[26]

The dielectric also acts as a coolant and erodes the metal away as well. The spark gap, which is also called the “discharge gap” is the distance between the electrode and the workpiece. The discharge gap varies from 0.005mm to 1.0mm. The more the gap, the bigger the spark is and vice versa.

The sparks are produced about 1000 - 100,000 times per second between the electrode and the metal by which, it carves a shape of the electrode to the metal. The machining takes place till the required shape and finishing is achieved.

The electrode is commonly made of graphite or copper, depending upon the workpiece and the operation to be done.

If the electrode accidentally touches the workpiece, the spark stops and therefore, there won't be any erosion. There is a discharge servo, which makes the movement of the electrode up and down. So, it takes the electrode up when this kind of things happen.

Wire Cutting EDM.

This is the same mechanism which uses a wire to cut the material. The wire EDM is used in places where contour or tapered cuts are needed. Such cuts are not easy or in some cases not possible to cut in CNC. So, wire EDM is used to cut such difficult shapes.

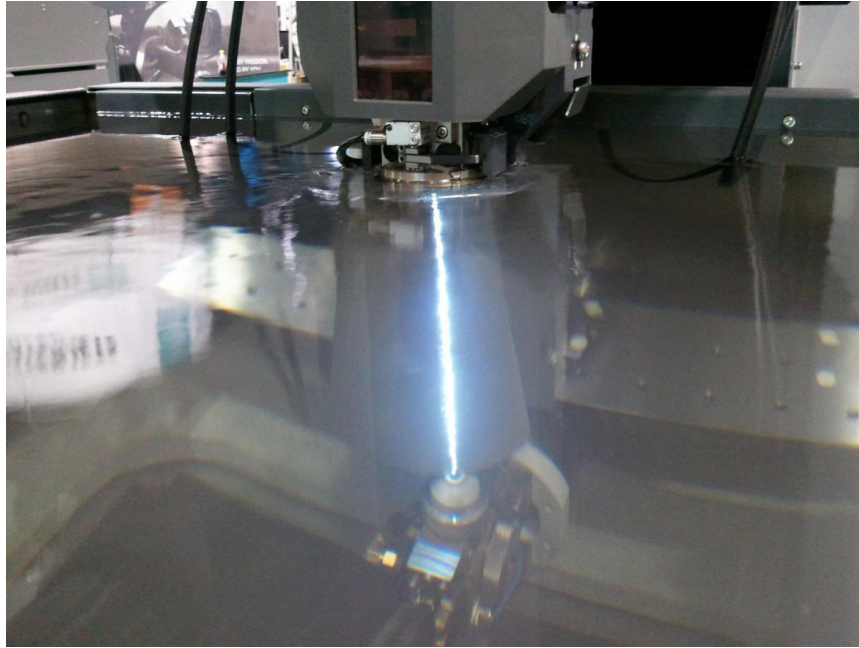


Figure 39- A working Wire-cutting EDM machine.[27]

Cutting is made on the workpiece where ,the start of machining is by first drilling a hole in the workpiece or can start from the edge of the workpiece. In the machining zone, each discharge creates a crater in the workpiece and thus have impact on the tool. The wire could be inclined, thus making it possible to make parts with taper or with different profiles at the top and bottom depending upon the design. There is never any mechanical contact between the electrode and workpiece. The wire is usually made of brass or stratified copper, and is between 0.1 and 0.3 mm diameter.

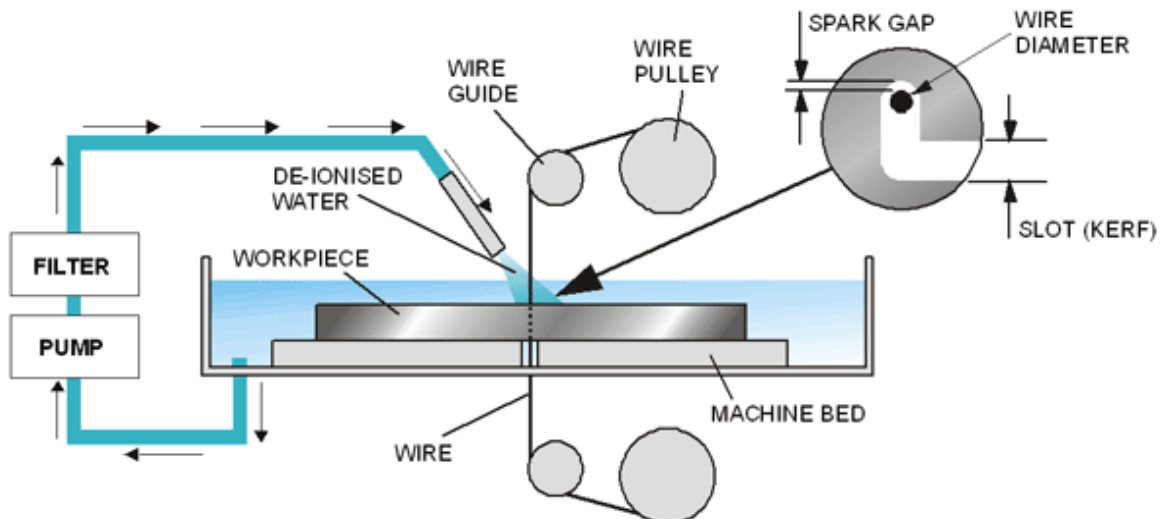


Figure 40- The process layout of a wire cutting EDM machine.[28]

The part after being manufactured in Erosion department, is sent back to the production where, they inspect the parts to be added and add them to the mold. Also, the polishing of the molding area of the mold is done if needed.

4.5. Injection department

The defects are the primary thing that needs to be inspected thoroughly and then the machine parameters are changed accordingly. The major defects that could affect the parts are discussed before.

Also, the defects are rectified by adjusting different parameters of the machine , which is also discussed before. The main thing that needs to be taken care are the mold temperature and pressure. The holding time also affects the part in the defect of warpage sometimes.

4.6. Dimensional analysis

As it is a necessary thing to check the dimensions of the part after it is rectified with its defects. The dimensions are basically done in accordance with the tolerance values given by the client., i.e., the values that the client provided for a specific area in the part should come within the specified tolerance.



Figure 41- The Laser Scanning device.[29]

This is the laser scanner from “Steinbichler”. This is the equipment that is used in the dimensional analysis. This has a scanner and a scanning probe.

Equipment setup:

- The scanner and the probe are connected to a common monitoring device.
- That device is connected to a specific laptop provided by steinbichler.
- The controller is provided by a power supply.

This is how the complete equipment is setup.

Working:

The part that needs to be scanned is placed in front of the scanner within its scanning limit.

1) The device is powered on and the scanning software in the laptop is opened. This acts as an interface between the equipment and the laptop to provide us the visual information about the part.

2) After opening the software, the file is given a name and the scanning parameters such as, range, intensity, details and other things are predefined.

3) Then with the scanning probe, the part is scanned. The scanning is done when two separate light sources merge and form the detail of the surface that it is scanning.

4) In some cases, the part needs to be scanned many a times if it is beyond the measuring range. Like big parts or the rear surface of the part needs to be scanned as well. Situations like this is dealt by taking a reference on the previously scanned surface and merging the both.

5) Thus, after scanning the required part in such a way, the part is optimized by deleting unwanted surfaces and disturbances scanned during scanning.

6) Then, the part is meshed to make it easy for the dimension measuring software to accept it and provide accurate results.

7) The software called “**Inspect plus**” is used to measure the part which was scanned in the software called “**Colin 3d**” using the scanner.

After measuring the dimensions and other things. It is required to make a report having all the details that are measured.

5. Water Assisted Injection Molding (WAIM).

ASSISTED INJECTION MOLDING.

The injection molding normally involves manufacturing of the product requested by the customer. In which, there are many challenges while manufacturing a part. To, overcome such circumstances, we adopt assisted injection molding.

It involves the usage of air or water to achieve the requirement for the requested part. Especially at the places where there is a huge change in thickness of the part and to reduce the weight of the part.

Hollowness in a part is created by this method. It either uses air or water and in some cases both.

5.1. Gas assisted injection molding.(GAIM).

Gas Assist Injection Molding is one of the assisted molding technique, which uses low-pressure nitrogen gas or other inert gases for applying even pressure for uniformity throughout the molded part.

The gas is let go through the arears where the thickness is more and also to the areas like edges.[30]

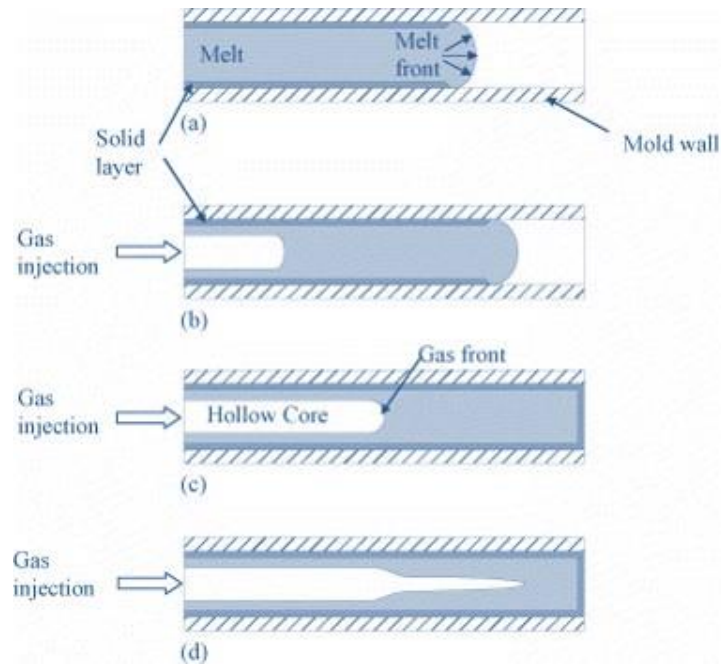


Figure 42-Different stages in Gas-assisted injection molding.[31]

How it works ?

The working is basically, when the plastic is injected inside the mold

- The plastic which is at the walls, contacting the walls will be cooled quickly than the one that is present after that.
- This cooling is done by the cooling channels in the mold.
- So, when we inject air inside, it automatically takes away the core plastic and not the plastic at the walls.
- The air that is injected also acts as a cooling agent and cools the plastic inside quickly.

At first, the mold should be closed and after that, the plastic will be injected inside the cavity. After the plastic is being injected. The air from a nozzle will be injected into the area of the plastic part where the plastic is to be removed. So, as the air is injected inside . it travels inside carrying away the plastic that comes with its force. There will be a reservoir at the end where the removal of plastic should be finished. So, the plastic that was carried away by the air is solidified there. Thereby, the plastic is either reused or thrown away depending upon the quality of the plastic and the requirement of the part. So, this is how a gas assist molded part is made.

In this method, there are two different cases:[21]

- The mold is filled short shot at first and then the gas is injected. So, as the gas goes inside the pressure of the gas pushes the plastic towards the wall and creates the hollowness. Mostly, in this case there won't be a reservoir at the end to collect the plastic as we inject less plastic at the very beginning itself.
- The other case is the one with the reservoir at the end to collect the plastic at the end. Here, the plastic is filled completely in the mold.

Applications:

- It cools the part from the inside as well. Which eliminates the possibility of warpage, sink marks and internal stress.
- This helps reductions in clamp tonnage, cycle time, and part weight and rigidity of the part is increased.
- Parts with a lot of ribs, bosses and gussets are made perfectly flat without sink marks and evenness.
- A lengthy part shape could be made less in drops. Also, eliminates weld lines & tooling costs.
- In many gas assisted molds, the clamp tonnage and other parameters will be reduced mostly by 50% or more.

External Gas Assist Injection Molding:

This is a bit different from internal gas assisted injection molding. Here, the gas will be injected from the core part of the mold. The gas will be injected as a thin layer in-between the core and back of the part.[3]

By this method, we will get a perfect and smooth surface at the front of the part, as it normally cools the part surface evenly. We will not get sink marks in this method.

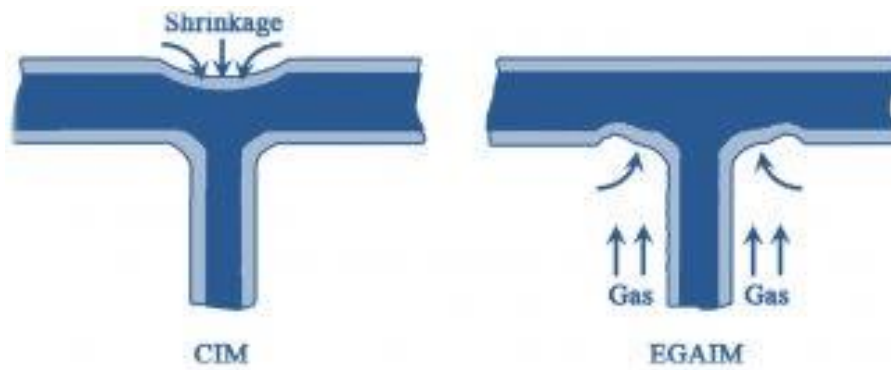


Figure 43- Effect of cooling in an External GAIM part.[31]

In figure 43, the left one is normal conventional molding where the cooling is done at the bottom side, i.e. at the ribs side.

So what happens here is that, the thickness of plastic at the junction of rib has more plastic and makes it difficult to be cooled at the same time when the other areas having even thickness get cooled.

So, the plastic at the rib gets cooled at first and it starts to shrink, which is propagated towards the root where the junction lies. This is the reason why we get shrinkage.

At the right image, we have shrinkage at the sides of the rib since we have the gas layer at the top of the part. It creates even cooling at the surface thereby, the shrinkage occurs evenly at the top area leaving the shrinkage at the other sides.

5.2. Water assisted injection molding (WAIM)

This is the next type of molding found after gas assisted injection molding. This has a lot of advantages over gas assisted molds.

As the name indicates, the main difference of this from gas is that it uses water to take the plastic instead of gas. It has received a lot of appreciation in the recent years because of its light weight products. This helps a lot to reduce the weight of the molded part when compared to the former.

Ahmed Zia & Amir Hossein Behravesh [32] mentioned that, "In 1998, the first report was made on WAIM at the "Institut für Kunststoffverarbeitung" (IKV), a plastics processing development centre in Germany".[33]

Water-assisted injection-molding technology has received extensive attention in recent years, for its light weight products, relatively lower resin cost per part, faster cycle

time, and its flexibility in the design and manufacture of plastic parts. WAIM is especially well suited for a number of hollow-parts applications, such as automotive fluid handling tubes for oils and coolants, automotive door handles, oven and refrigerator handles, chain saw handles, office furniture chair arms and so on.

WAIM has been under development for more than ten years and is already successfully commercialized in Europe.

WAIM is the newest way to mold hollow or partly hollow parts. It's basically similar to GAIM as a means to core out thick sections. Generally, according to the volume of injected melt, WAIM can be divided into short-shot process and full-shot process.

The main difference between these two is the way of filling the melt inside the cavity. If the filling is done partially, it is short shot and if the filling is done completely, it is full shot molding.

Short shot :

At first, the mold will be filled with the polymer partially and then, the water will be injected inside. Which, when injected exerts pressure on the inner walls of the plastic and forces the plastic to get completely in contact with the walls of the mold and it retains for a while until the plastic melt solidifies as the required part. The cooling of the melt is carried out by the same water that is injected at the beginning of the process. The water is let to stay inside for a specific duration ensuring a perfect part.

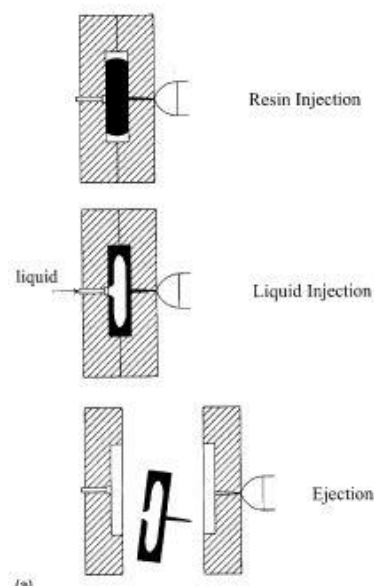


Figure 44- The steps for a short-shot WAIM.[32]

After that, the water will be released or sucked outside the mold.

Full shot:

Here, the plastic will be filled inside the mold almost completely and only a very small area will be left where, the plastic needs to be removed. Then, the water will be injected with a high pressure inside the mold. The injection of water will take place shortly after the plastic is filled, as the plastic touching the should be solidified.

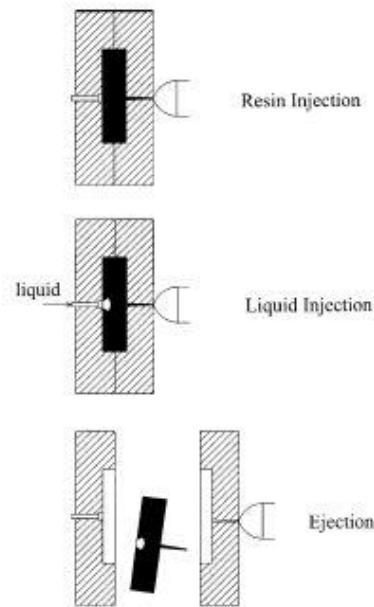


Figure 45- The steps for a Full-shot WAIM.[32]

Then, the plastic will take away the plastic in its path and travel towards the place where it was directed. As the water flows inside the mold, it also cools the plastic from inside.

It then pushes the plastic outside the part and to the reservoir made in the mold adjacent to the part.

The extra attachment (removed plastic) will be cut from the part later. One of the disadvantages in short-shot water-assisted injection-molded parts is the switchover mark on the surface of molded products. Full-shot water-assisted injection-molding has the advantage of eliminating the sink marks on the surface of molded parts. In a full-shot molding, the mold cavity is first completely filled with the polymer melt followed by the high pressure water into the core of the polymer melt in order to avoid shrinkage of the polymer part.

5.3. Water assisted injection molding at MoldeTipo

There are many water injection molds that are being manufactured in company. The parts that we manufactured had many defects and they were rectified as well. From which I have learned some information about water injection molds and the process.

The reason why WAIM is chosen over GAIM are,

- Water assisted molds cools the part very rapidly. Thereby reduces the cycle time by 30% over gas assisted.
- This rapid cooling is due to the specific heat of water, which is more than 4 times that of the gas. Also, the thermal conductivity is about 40 times higher.
- The plastic removed is much more compared to gas.
- The surface finish of the inner wall is smooth and the material removal rate is even.

Although, WAIM has many benefits over GAIM. The main problem here is the voids. The formation voids in the residual walls of the part is the main problem in WAIM parts.

Why voids are formed ?

- The fluid injected inside the plastic part to make it hollow includes Water and silicone.
- The first reason for the formation of void is that the flow of the fluid in forming the hollow section. If the fluid flow is unstable inside the mold during the melt removal, it create voids.
- The secondary one is due to the varying residual wall thickness, that paves way to shrinkage and eventually forms void.
- This varying cross section usually is formed in a curved area of the part. This is due to the difference in the melt solidifying time at the outer and inner walls of the part.

The void occurrence at the residual wall of the hollow section is highly influenced by semi-crystalline polymer, as it has large volumetric shrinkage. Which in turn, creates solidification unevenly and forms void.

Furthermore, if glass fibre is added to the polymer to increase its reinforcement of strength, it makes the polymer even more prone to voids.

In general, approximately 20% of the specific volume changes during the process due to change in temperature and pressure of the polymer. This change in specific volume creates shrinkage in the part. The volumetric shrinkage mostly occurs during the cooling process which affects the dimensional accuracy and quality of the final product.

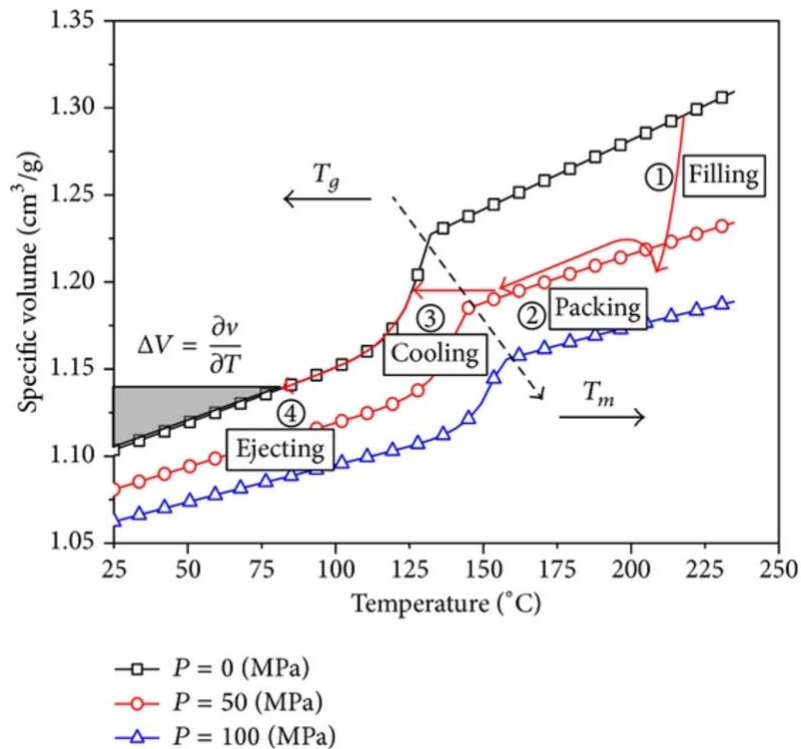


Figure 46- PvT diagram of a typical semi-crystalline polymer.[34]

The above image shows the PvT diagram of a typical semi-crystalline polymer in injection molding. When the polymer filling takes place, the shift is to position 1. This is due to the change in specific volume which is caused by increase in injection pressure. After which, @ 2, the packing pressure is in action so, there is a decrease in temperature and pressure as the part begins to solidify and the specific volume changes as well. @3, the cooling occurs which reduces the pressure of the molded part to atmospheric pressure. And @4, the change in specific volume is huge as the temperature zones passes through the melting and glass transition temperature.

The void formation is clearly interpreted in the below image,

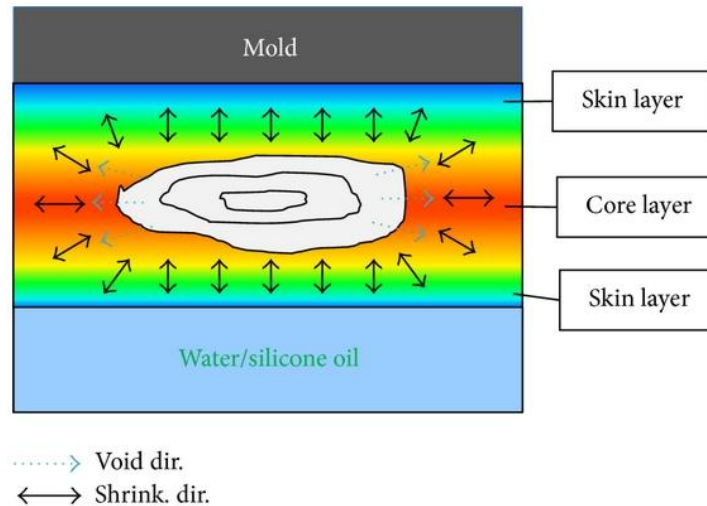


Figure 47- Void formation with heat flow inside a part.[34]

Here, as it is clear that, the void is formed inside the wall of the part as the area, is yet to be cooled or solidified. This is the main reason why internal voids are formed more often.

The dotted arrow indicates the void propagation direction and the double-sided arrow indicates the shrinkage direction. The different colour indicates the heat intensity. The red indicates the highest, while the blue indicates the lowest. i.e., the red is unsolidified melt and the blue is the melt solidified. So, the shrinkage occurs when the solidifying area pulls its neighbouring surface and thereby forms voids in the unsolidified or area that needs to be made solid.

The void due to shrinkage, first grows rapidly due to internal shrinkage and it stops growing after a certain period of cooling as the part will be solidified already.

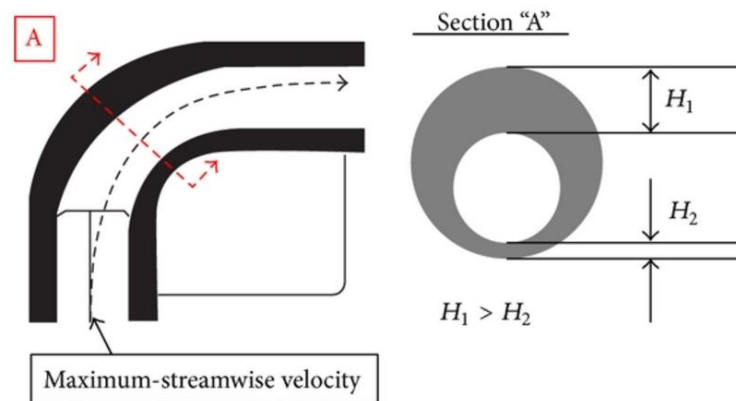


Figure 48- Varying Residual wall thickness in the curved section of a part.[34]

At the time of fluid injection to form hollow section, the removal of plastic and the residual wall thickness is even in straight areas, whereas in curved areas, the thickness of the residual wall is uneven and not uniform.

The outer thickness of the wall will be more than that of the inner wall thickness. The reason for this is, mainly the centrifugal force generated by the fluid flowing in the curved area.

There are other parameters that contribute for the development of WAIM. The water penetration during the creation of hollow section is very important. There are factors that affect the penetration of water inside the part during the process of injection.

5.4. Important parameters for WAIM.

To understand the WAIM process even more deeply, a study of different parameters that effects the process of WAIM is done. From which the important aspects that needs to be considered are studied precisely.

Ahmed zia & Amir Hossein behravesh published an article titled “ An Experimental investigation on water penetration in the process of water assisted injection molding of polypropylene”. Where they have taken some important parameters that effects the WAIM process and experimented them to identify the ideal requirements to achieve maximum penetration for a best product .[35]

Another study conducted by the same duo, **Ahmed zia & Amir Hossein behravesh**[36] on **Effects of process parameters in water assisted injection molding of ABS**, have made an experimental analysis of the parameters delay time, holding time and mold temperature. The results gave an idea of how effective these parameters are when manufacturing a water assisted injection molding part.

A series of experiments has been conducted to understand the process intensity at the change of each parameter.

ANALYSIS ON POLYPROPYLENE:

The part is shaped like a branched tree with two heads, one is straight and the other is bent to an angle from the straight to know the effect clearly.

The effect of delay time on water penetration at various mold temperatures and holding time is depicted in figure 49.

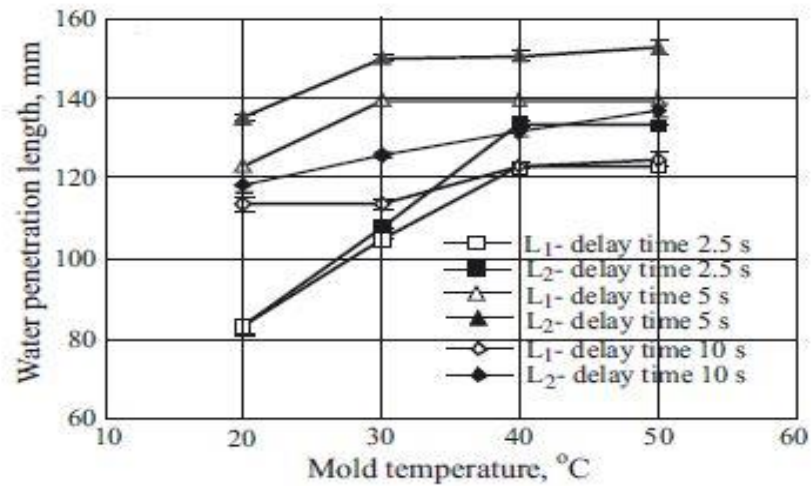


Figure 49-Water penetration on mold temperature.[35]

This is the graph plotted with the holding time of 5sec, it is evident from it, that the penetration increases with increase in mold temperature at different mold temperatures.

The reason being, the warm temperature. As the temperature is warmer inside the mold, it slows down plastic becoming solid and also, the temperature assists penetration by decreasing the plastic resistance.

The result was almost the same for the remaining, where the maximum penetration occurred at 40°C also the mold temperature is more responsive at lower delay time(2.5s). This is because the plastic just starts to plasticize giving more penetration length for water.

Then in figure 50, the effect of delay time on water penetration is plotted ,

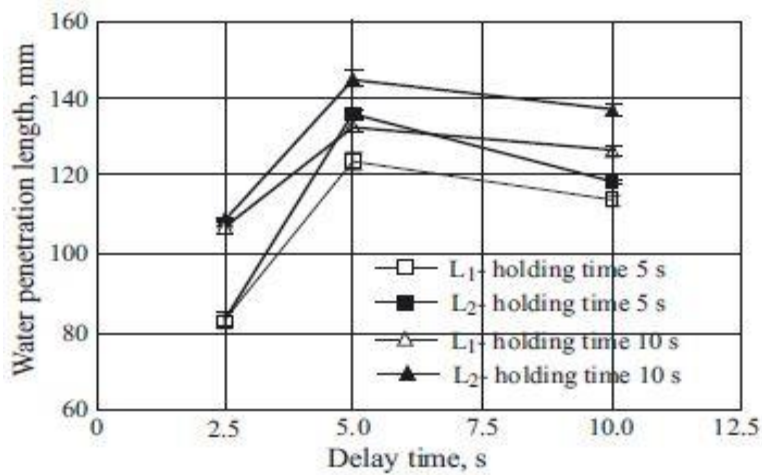


Figure 50- Water penetration on Delay time [35]

This gives us the penetration length, which is highest at a delay time of 5 sec, falls shortly after preceding delay times. This is due to the increasing cooling time of the plastic which creates more resistance on the surface of the melt, thereby obstructing the flow of water.

Also at times below 5sec time, lower penetration length is observed, which is because, the plastic will just start to solidify leaving a large amount of melt melted and so, the water injected won't have a specific path to flow and starts to flow in many directions and thereby reducing the penetration.

ANALYSIS ON ABS:

The same part, that is shaped like a branched tree with two heads, one is straight and the other is bent to an angle from the straight is taken here for the ABS as well.

The effect of delay time, holding time and mold temperature on water penetration and hollow core diameter are studied. The delay times of 0, 2.5, 5 and 10 sec are chosen. Holding times of 5 sec and 10sec are chosen and mold temperatures of 30°, 40° and 50°C are chosen.

The measurement starts at 50mm from the gate and ends for the straight leg at 140mm and for the curved leg at 153mm. In figure 51, The effect of delay time and holding time on water penetration length at the straight leg(L1) and curved leg(L2) of the part at different mold temperatures are measured and plotted

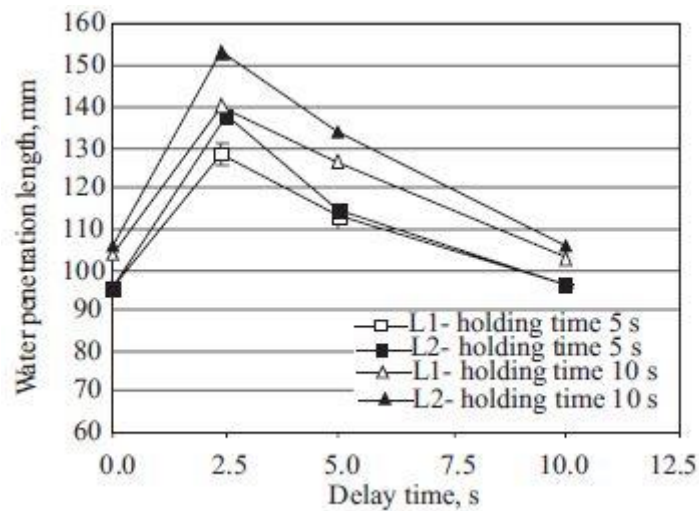


Figure 51-Effect plotted at 40°C.[34]

From the graph, it is clear that the water penetration increases with increase in delay time up to a certain point and then decreases after that. The maximum water penetration occurred at a delay time of 2.5sec at all the mold temperatures

It behaved with a slight difference when the mold temperature is 50°C, where it lasted till 5 sec of delay time.

At the higher delay times the penetration decreased because the solidification of the melt inside the mold takes place and thereby, the melt that should be removed by the water injected is solidified at the inner wall of the part and due to which the water penetration reduces as delay time increases.

Even at zero delay time, There is penetration. But it is very less as compared to more delay time. In this case, we will have sink marks in the part surface as there would be a huge thickness difference in the residual wall and it will also have voids.

The effect of mold temperature on water penetration length at various delay times are justified in figure 52.

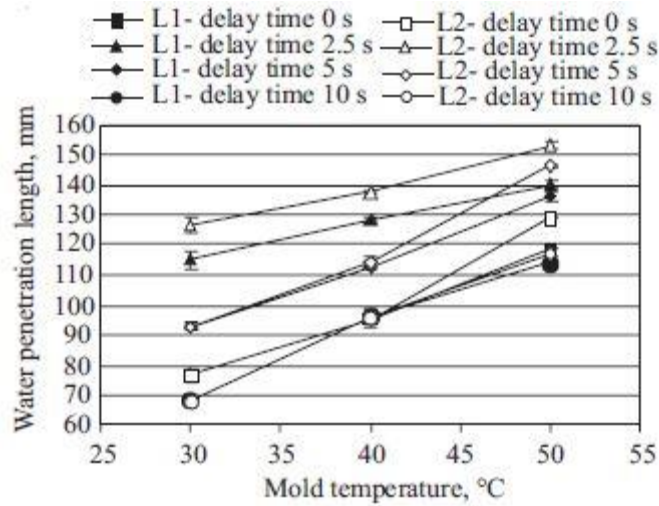


Figure 52- Effect plotted at holding time of 5sec.[34]

Here, from the graph it is clear that the penetration length increases with increase in mold temperature. And as usual, the perfect delay time which gives the penetration maximum is 2.5sec.

The maximum penetration occurs when the temperature is 50°C. The length decreases when the delay time increases, which clarifies that the melt gets solidified and thereby providing less penetration, it tends to sharply increase when the temperature changes from 40°C to 50°C.

In figure 53, The effect of mold temperature and delay time on hollowed core maximum and minimum diameter difference at different holding times are depicted.

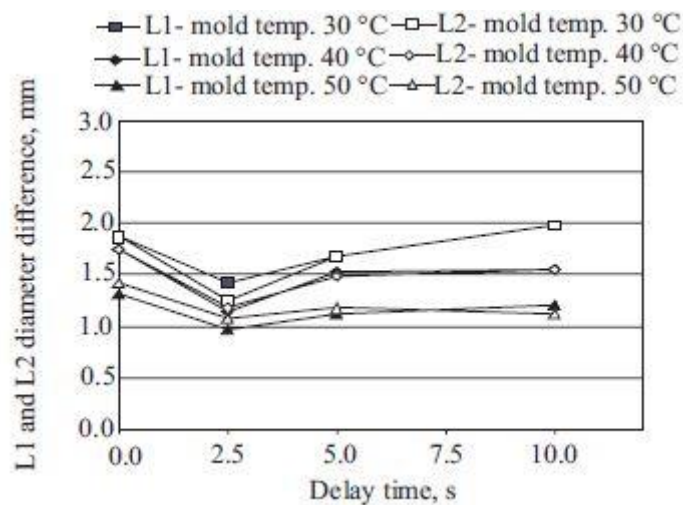


Figure 53-The Effect at holding time of 5sec.[34]

Figure 53 represents the experiment done at a holding time of 5 sec and figure 54 indicates the holding time of 10sec

From the figures, it could be seen that the thickness difference tends to be more in the beginning at zero delay time and also at the end delay time of 10sec. whereas the difference is lowest at the usual delay time of 2.5sec.

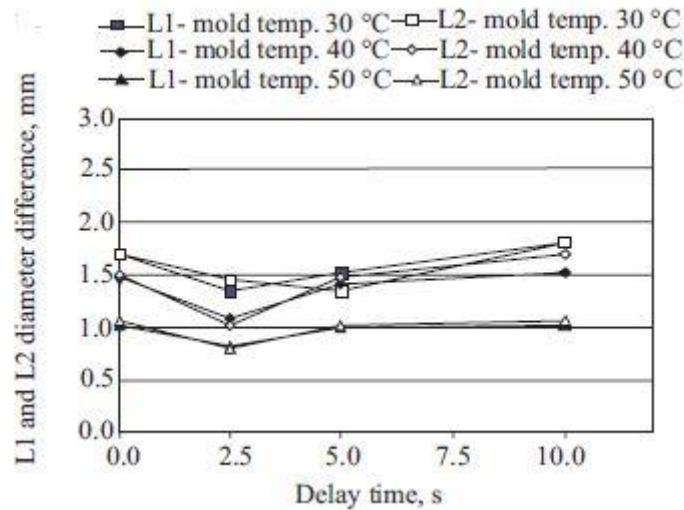


Figure 54- The Effect at holding time of 10sec.[34]

The first graph represents the uniformity of the hole diameter and the second one represents the eccentricity of the hole. As the pipe is a branched pipe, the hole at both the ends of the heads seems to be eccentric in the opposite direction.

Thus, with these results from the literature review done. The behaviour of water when it is injected over a mold filled with melt. The process assessment gave an understanding of how important is the weight to strength ratio for a part manufactured through injection molding. And the importance of uniform wall thickness for a part made by WAIM. Also, the material selection for a part that needs to be made through WAIM is the most prominent step. Overall, an effective way of manufacturing a part through Water Assisted injection Molding(WAIM) is understood.

A part is approved good, when it makes sure that it won't fail its designated use and it could be made by understanding the parameters that improves, effects and helps to make them perfect.

These details, that are prescribed above clearly shows that, a WAIM part could be improved by adjusting delay time, holding time and mold temperature. Hence, the requirements to achieve a perfect part is studied clearly with this study.

6. Conclusion.

Thus, from my internship, I have acquired fundamental knowledge about injection molding and the perks in the industry. This industry is one such industry, where the dimensional accuracy is one of the most important part in making any product. The tolerance of the part that the client has requested should be achieved perfectly from the mold. So, we obviously have to manufacture the mold in such a way that it should be able to produce the part in best values of quality and dimensions. My internship has given me a lot of information and experience about almost the whole process of making a part out of mold. I have also been into the commercial department of my company, so I know how my colleagues invite clients to have their project done in my company.

The main thing that I understood from the production in my company is the problem solving skill. The way they solve the problem was a lesson for me. I saw them making multiple solutions for a problem, and these solution comes by detecting the possibilities for the problem. Some problems were so difficult, that they could not solve it even after attempting certain solutions and they were solved with a rational way of approaching the problems, which in turn have given solutions. I got to know how, precise cuts and designs that are not possible in conventional machining are machined perfectly in erosion department. Then at the injection department where the part is being made gave a lot of answers to me. Which is understood by inspecting the part that is made and comparing it with the design given by the client. By identifying the flaws in the part we can find multiple reasons for it, which would be rectified in the production / Bancada. So, thereby I got an idea how a flaw is corrected. Also, it is in the injection department, that I gained and understood how a water assisted injection molding part is made, what are all the parameters that should be given priority while manufacturing a WAIM part, advantages, drawbacks and other information. So, from my entire internship, I learned a lot of lessons and skills not just on the molding also on other maturity to tackle with problems and patience.

References

- [1] G. Potsch and W. Michaeli, *Injection Molding An Introduction*. 2008.
- [2] “Machine parts,” *sinotech*, 2017. [Online]. Available: <https://www.sinotech.com/resources/tutorials/injection-molded-parts/>. [Accessed: 02-Mar-2017].
- [3] F. Johannaber, *Injection molding machines : a user’s guide*. Munich, Germany; Cincinnati, Ohio: Carl Hanser Publishers ; Hanser Gardner Publications, 2008.
- [4] “Machine components,” *Engineering education*, 2011. [Online]. Available: http://www.dc.engr.scu.edu/cmdoc/dg_doc/develop/process/control/b1000002.htm. [Accessed: 25-May-2017].
- [5] M. L. Berins and S. of the P. Industry., “SPI plastics engineering handbook of the Society of the Plastics Industry, Inc.” Kluwer Academic Publishers, Boston; New York, 2001.
- [6] R. Dangel and R. Dangel, “2 – Mould Types,” in *Injection Moulds for Beginners*, 2016, pp. 3–42.
- [7] “Design Approaches,” *Proto labs*, 2017. [Online]. Available: <https://www.protolabs.com/resources/design-tips/improving-part-moldability-with-draft/>. [Accessed: 15-Mar-2017].
- [8] S. Dalton, “Mould Design,” 2011. [Online]. Available: <http://staffweb.itsligo.ie/staff/sdalton/ptooling/Mould-Design-slides.pdf>. [Accessed: 20-Mar-2017].
- [9] W. Michaeli, H. Greif, G. Kretschmar, and F. Ehrig, “Training in Injection Molding,” in *Training in Injection Molding*, Carl Hanser Verlag GmbH & Co. KG, 2001, pp. i–4.
- [10] “Cooling channel.” [Online]. Available: <http://www.lbc-engineering.de/en/conformal-cooling.php>. [Accessed: 23-Mar-2017].
- [11] Tuula, “Mould standard parts.” 2007. [Online]. Available: http://webhotel2.tut.fi/projects/caeds/tekstit/mould/mould_standard_parts.pdf. [Accessed: 25-Mar-2017].

- [12] “Steels used in Injection Mold making,” 2015. [Online]. Available: <http://info.crescentind.com/blog/bid/66085/Understand-Common-Steels-Used-in-Injection-Mold-Making>. [Accessed: 27-Mar-2017].
- [13] “stratasysdirect,” 2017. [Online]. Available: <https://www.stratasysdirect.com/materials/tooling-molding/>. [Accessed: 29-Mar-2017].
- [14] A. Materials, “Data sheet of P20 steel composition.,” 2017. [Online]. Available: <http://www.azom.com/article.aspx?ArticleID=6239>. [Accessed: 20-Sep-2017].
- [15] Doerrenberg, “Data sheet of 1.2312 40CrMnNiMo steel.,” 2005. [Online]. Available: http://metallurgicaveneta.it/pdf_eng/acciai-da-utensili-per-lavorazioni-a-caldo/1.2312.pdf. [Accessed: 20-Sep-2017].
- [16] Doerrenberg, “Data sheet of 1.1730 steel.,” 2005. [Online]. Available: www.doerrenberg.de/uploads/tx_c1x1downloads/1.1730_en.pdf. [Accessed: 20-Sep-2017].
- [17] “Major defects in plastic Injection molding.,” 2016. [Online]. Available: <https://www.creativemechanisms.com/blog/what-cause-injection-molding-defects-and-how-to-fix-them>. [Accessed: 28-Apr-2017].
- [18] “Plastic warpage,” 2009. [Online]. Available: <http://exploringcad.blogspot.pt/2009/03/plastic-componets-design-considerations.html>. [Accessed: 30-Apr-2017].
- [19] “Types of Injecrion molding gates.,” *Engineering education*, 2011. [Online]. Available: http://www.dc.engr.scu.edu/cmdoc/dg_doc/develop/design/gate/32000002.htm. [Accessed: 01-May-2017].
- [20] “Important factors to injection molding,” 2015. [Online]. Available: <http://info.crescentind.com/blog/bid/59146/5-Factors-Important-to-Injection-Molding>. [Accessed: 02-May-2017].
- [21] R. J. Crawford, *Plastics engineering*. Butterworth-Heinemann, 1998.
- [22] “Melt Flow Rate,” 2017. [Online]. Available: <http://www.misumi-techcentral.com/tt/en/mold/2012/01/107.html>. [Accessed: 14-May-2017].

- [23] D. Industry, “3-Axis CNC,” 2017. [Online]. Available: <http://www.directindustry.com/prod/diversified-machine-systems/product-101789-1360335.html>. [Accessed: 21-May-2017].
- [24] “A brief about the process of milling,” *Custompartnet*, 2017. [Online]. Available: <http://www.custompartnet.com/wu/milling>. [Accessed: 20-Sep-2017].
- [25] “How edm works.” [Online]. Available: <http://www.xactedm.com/edm-capabilities/how-edm-works/>. [Accessed: 01-Jun-2017].
- [26] “EDM Principles,” 2017. [Online]. Available: <http://mechanicalinventions.blogspot.pt/2016/01/electrical-discharge-machining-edm.html>. [Accessed: 07-Jun-2017].
- [27] “An Intro to Electrical Discharge Machining,” *KEEN home*, 2017. [Online]. Available: <https://shop.keenhome.io/blogs/community/an-intro-to-electrical-discharge-machining>. [Accessed: 07-Jun-2017].
- [28] “Precision Cutting,” *ORIGIN Engineering*, 2016. [Online]. Available: http://www.originengineering.com.au/precision_cutting.html. [Accessed: 07-Jun-2017].
- [29] “Steinbichler – 3D-Scanner aus Deutschland,” 2017. [Online]. Available: <https://3druck.com/scanner/steinbichler-3d-scanner-aus-deutschland-2724830/>.
- [30] H. Belofsky, *Plastics: Product design and process engineering*. Hanser Gardner Pubns (April 1995), 1995.
- [31] Sajarplastics, “What is gas-assist-injection-molding,” 2017. [Online]. Available: <http://sajarplastics.com/what-is-gas-assist-injection-molding/>. [Accessed: 20-Sep-2017].
- [32] S.-J. Liu and Y.-S. Chen, “The manufacturing of thermoplastic composite parts by water-assisted injection-molding technology,” *Compos. Part A Appl. Sci. Manuf.*, vol. 35, pp. 171–180, Feb. 2004.
- [33] T. Jüntgen and W. Michaeli, “THE WATER INJECTION TECHNIQUE (WIT) AS AN ATTRACTIVE ALTERNATIVE AND SUPPLEMENT TO GAS-ASSISTED INJECTION MOLDING (GAIM) Plant and injector technology Comparison of GAIM and WIT,” *Proc. ANTEC 2002*, 2002.

- [34] K.-H. L. Hyung-Pil Park, Baeg-Soon Cha, Soo-Bin Park, Jae-Hyuk Choi, Dong-Han Kim, Byung-Ohk Rhee, "A study on the Void Formation in Residual Wall Thickness of Fluid-Assisted Injection Molding Parts," *Adv. Mater. Sci. Eng.*, vol. 2014, p. 06, 2014.
- [35] A. Z. Ahmadzai and A. H. Behraves, "An experimental investigation on water penetration in the process of water assisted injection molding of polypropylene," *Polimery/Polymers*, vol. 54, no. 7–8, 2009.
- [36] A. Z. Ahmadzai and A. H. Behraves, "Effect of processing parameters on water penetration in water assisted injection molding of ABS," *Polimery/Polymers*, vol. 56, no. 3, 2011.

This page was intentionally left blank

Appendices

This page was intentionally left blank

Glossary
