Internship Report

Masters in Product Design Engineering

Internship Report and Study concerning the Threats and Opportunities posed by Additive Manufacturing to Moulding Industry

Name: Anurag Ayyagari Venkat

Leiria, *December*,2015

Esta página foi intencionalmente deixada em branco

Internship Report

Masters in Product Design Engineering

Internship Report and Study concerning the Threats and Opportunities posed by Additive Manufacturing to Moulding Industry

Name: Anurag Ayyagari Venkat

Master's internship carried out under the supervision of Doctor Carlos Fernando Couceiro de Sousa Neves, Professor of the School of Technology and Management of the Polytechnic Institute of Leiria.

Leiria, *December*,2015

Esta página foi intencionalmente deixada em branco

Aknowledgement

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 MOLDEGAMA for giving me an opportunity to gain information and knowledge that was required for carrying out my course. First I would like to thank Engineer Gonçalo Cordeiro for willing to clarify any questions or solve any problem that arose in support of this report. I would also like to thank all the people from MOLDEGAMA for helping me and passing over their knowledge to me.

Esta página foi intencionalmente deixada em branco

Resumo

Neste relatório é feita apresentação de várias das peças e sistemas utilizados em moldes para a injeção de plásticos. Inicia-se esta revisão com uma explicação sobre os moldes de injeção e sobre o próprio processo da injeção de plásticos, seguindo-se-lhe a descrição de peças normalizadas e o mapeamento dos principais passos do processo de fabrico dos moldes. Este relatório comporta ainda uma revisão de casos de estudo acerca de métodos de fabricação híbrida, aditiva e subtrativa. Finalmente, é também feito um estudo sobre as oportunidades e as ameaças postas pela fabricação aditiva à indústria de moldes, com o objetivo de compreender as tendências do futuro da produção de peças plásticas.

Esta página foi intencionalmente deixada em branco

Abstract

This report will make a presentation and analysis of various parts and systems used in injection moulds for plastics. This review starts by explaining the injection moulding process of injection, followed by a description of mould standard parts and a process map of mould making. The report also comprises case studies regarding hybrid manufacturing methods of subtractive and additive manufacturing processes. A study regarding opportunities and threats posed by additive manufacturing to the moulding industry is also made in order to understand the future trends of plastic part production.

Keywords: Mould Design, Injection Moulding, Additive Manufacturing

Esta página foi intencionalmente deixada em branco

List of Figures

Esta página foi intencionalmente deixada em branco

List of Table

Esta página foi intencionalmente deixada em branco

List of Symbols

PVT	Pressure Volume Temperature
P/L	Parting Line
AM	Additive Manufacturing
SLA	Stereolithography
FDM	Fused Deposition Modelling
3DP	3D printing
LOM	Laminated Object Manufacturing
SLS	Selective Laser Sintering
SLM	Selective Laser Melting
LMD	Laser Metal Deposition
mm	Millimetre
STL	Standard Tessellation Language
UV	Ultraviolet
ABS	Acrylonitrile butadiene styrene
CNC	Computer Numerical Control

Esta página foi intencionalmente deixada em branco

Table of Contents

Part II - Study concerning the Threats and Opportunities posed by Additive Manufacturing to Moulding Industry

Esta página foi intencionalmente deixada em branco

PART - I

INTERNSHIP REPORT

1. Introduction

This document is a study about the process of mould making and plastic injection. I would also like to make a brief presentation of the company Moldegama: Moldes Técnicos SA, where I performed an internship as a Product Design Engineer for a duration of nine months. After this period in the moulding industry, I have got an insight into the practical aspect of the working process. Therefore, along with Engineer Gonçalo Cordeiro (supervisor in company) and Professor Carlos Neves (academic supervisor), it was decided to make a report on the internship relating to the mould making industry. Along with the internship report, a study concerning the opportunities and threats that recent developments in additive manufacturing pose to the moulding industry was also developed.

In the internship program, I had to follow the production of a mould from start to finish because it was important to have an understanding about what moulds are and how they are made. During the process, I gained knowledge about the different components of a mould, the machining processes, the adjustment of moving components, the parameters influencing plastic injection moulding and also about problem solving methods. I was also provided training in ZW3D CAD software for mould designing during this time. After the mould trials, I started to learn the designing of electrodes for Electro Discharge Machining. It was interesting to understand how the design has to comply with the machining parameters of CNC. During this time, I was provided another training in the CIMATRON software for modelling and mould designing. As soon as the training ended, I was sent to the designing area of Moldegama where I had to observe how the moulds are designed and understand the designing of various components of a mould. Soon, I started making some 2D drawings of parts and modifications in the moulds. Later, during the last few weeks of internship, I came back to the production area in order to understand and reason each and every step of the mould making process. This helped me to relate the process of designing and that of mould production in a better way. This report has a brief description of what I learned during the course of the internship.

The document consists of two parts. The first part is the presentation, description and analysis of moulds, moulding processes and plastic injection in moulds (Figure 1) and the second part of the report is a study on developments and future prospects of additive manufacturing and how additive manufacturing is having an impact on the moulding industries.

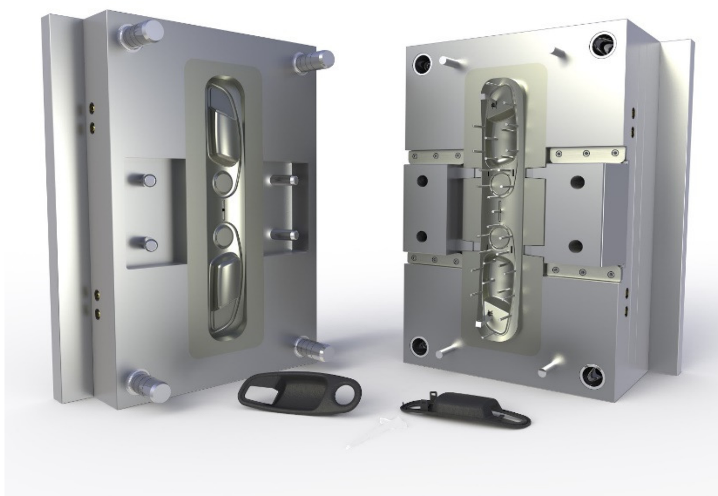


Figure 1 - Plastic Injection Moulds (XCENTRIC)

2. About the Company

Moldegama is a 28 year old experienced mould making company. It was founded in 1987, as a result of the will and entrepreneurship of 11 technicians of four different leading companies inside the industry, who decided to gather their knowledge and experience. At the beginning, Moldegama decided to dedicate its work to the manufacturing of tools for house appliances and houseware. However, after decades of change and development, Moldegama became one of the leader companies in the mould industry not only nationally but also abroad, specifically focusing on to the automotive industry.

Following the growth of projects, in 2009, the HMM Group (Figure 2) was formed as the parent Moldegama under which the company was divided into four different sectors, namely – Moldegama Innovation, Moldegama Moulds, Moldegama Plastics and Moldegama III Services.

Figure 2 - HMM Group (S.A., 2015)

Moldegama Innovation dedicates itself entirely to research & development, searching for solutions in what the industrial and manufacturing products in the field of automation, aeronautics, pharmaceutical, agricultural and telecommunications are concerned.

Moldegama Moulds is a part of HMM Group which is a mould making industry where the designing, production and modifications in moulds are carried out.

Moldegama Plastics is another venture of HMM Group where the mould trials and part production in done from the moulds.

Moldegama Services III is situated in Valencia, Spain. It also dedicates itself to modifications and trials of moulds.

Name of the company	MOLDEGAMA, S.A.
Address	Rua da Forcada, n° 6 – Ferraria 2445-712 Pataias, Portugal
Telephone number	+351 244 587 080
Fax number	+351 244 587 086
Email address	moldegama@moldegmama.com
GPS coordinates	8° 58' 24.64" W 39° 41' 01.82" N
Website	www.moldegama.com

3. Injection Moulding

Injection Moulding is a manufacturing process in which the parts are produced by injecting molten material in the mould. A wide range of materials can be used to make parts through injection moulding like metals, glass, elastomers, confections, thermoplastics and thermosetting polymers. Figure 3 shows the general layout of an injection machine.

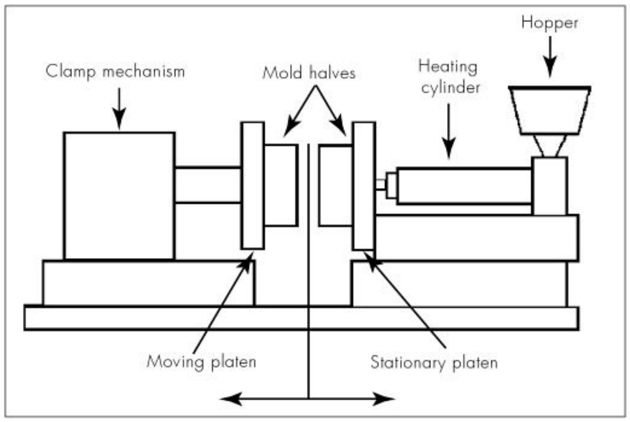


Figure 3 - Moulding Machine Layout (Bryce, 1996)

The material to make the part is fed to the machine through a heated barrel which is then mixed and forced into the mould cavity through a nozzle. In the cavity, the material cools and hardens to form the shape of the cavity of the mould. Normally, the movable part has an extractor system to ease the part removal process from the mould. The moulds are usually made up of metal, either steel or aluminium. Injection moulding is widely used when a large quantity of parts have to be manufactured, part size ranging from any of the smallest component to a complete automobile body panel.

Injection Moulding with PVT Diagram: Injection moulding is a cyclic process consisting of four phases (Wang, 2012): filling, compressing (packing), holding and cooling, as shown by the typical PVT diagram in Figure 4(a), cavity pressure profile in Figure 4(b), and cavity temperature profile in Figure 4(c). In the explanation below, it is assumed that the material to be injected is plastic.

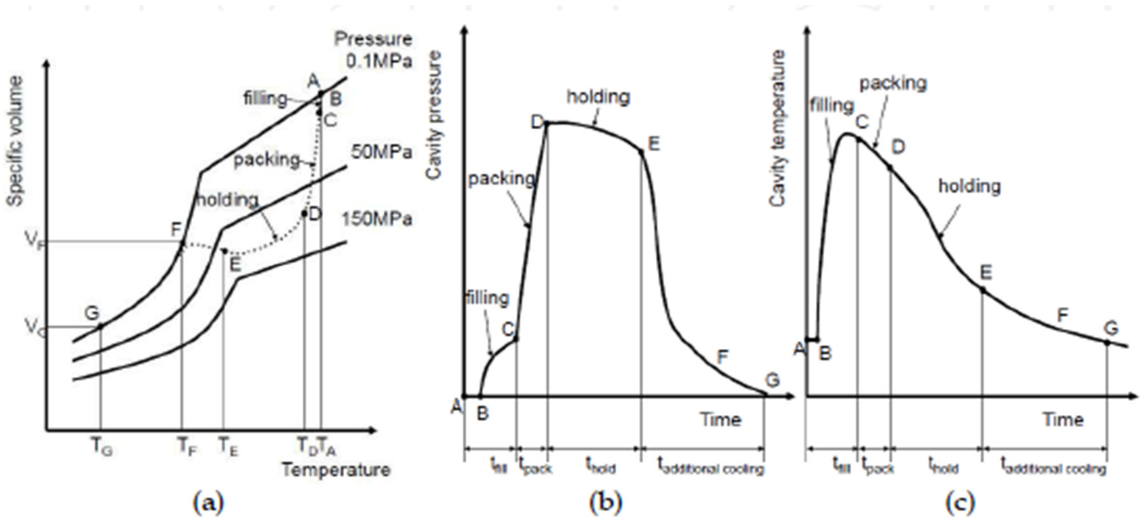


Figure 4 - (a) PVT diagram, (b) cavity pressure profile and (c) cavity temperature profile (Wang, 2012)

The filling process in an injection mould begins at point A. Cavity pressure comes into play at point B where the pressure increases steadily as the filling proceeds. In the filing phase, the cavity is filled volumetrically by the melt plastic without any compression. At point C, the filling phase completes. Next, the packing process takes place and the pressure rises rapidly to the peak value at the point D. At this point, the injection pressure switches over to the holding pressure. Thereafter the melt in the cavity is maintained at an assigned pressure during the holding phase so that the mould gets completely filled. This process continues until the gate is frozen. Point E is the end of the holding phase. At the beginning of point E, a phase of constant volume is maintained. This phase decides the dimensional accuracy of the moulding part. Reaching point F in a uniform way is important for the constant weight and dimensions of the moulding. After point F, the moulding cannot be influenced anymore. The plastic part shrinks depending on the ambient temperature.

4. Process Map of Mould Making

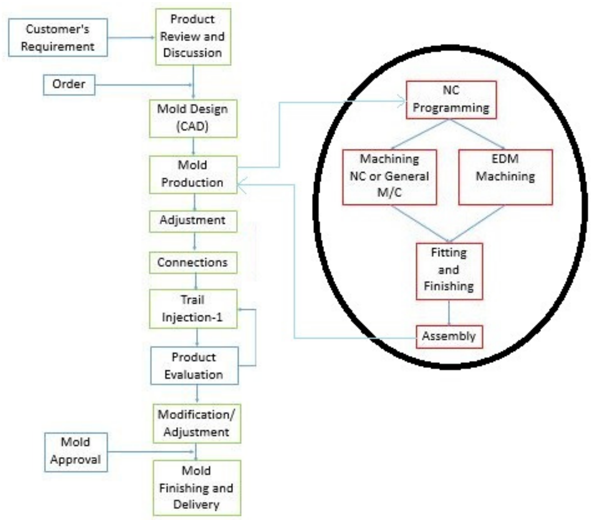


Figure 5 - Process map of mould making

A short history (Herbert, Understanding Product Design for Injection Molding) of injection moulding will help us to understand the needs and requirements of a well designed and manufactured mould in a better way. After the Second World War the plastic technology began to grow but there were no "mould designers". When a mould was needed, it was produced by artisans in tool and die making shops. These artisans were skilled in building accurate steel tools and dies, and the owner of such shops would work closely with the moulder. The moulder sketched, often roughly, how a mould should look, and the owner would supervise the machinists as they built the mould components. Then by assembling and testing the mould himself, he would decide if the mould is a well-functioning mould or not. This approach of making moulds was suitable at that time as there were fewer plastic materials available for injection moulding and the expected standard of moulds was low. But over the years, many new and better plastics have been developed and used in a wider range of products, often requiring different moulding parameters. At the same time, the demand for increase in productivity has become a high priority. Figure 5 shows the process map of mould manufacturing. The sequence of processes is only for explanation, and maybe altered depending on the mould.

4.1 Customer Requirement

This sows the seed of a mould. The customer (client) sends the design of the part and specifies the requirements of the mould. The customer always sends a file containing the information about the type of plastic to be injected and the injection machine in which the mould would be used. This is called the 'technical file' of the mould. Sometimes the designer has to modify the part so as to make a mould work better. The modifications are usually done because of inappropriate draft angles and to make the moulds more reliable. A few suggestions are also made by the company to the client regarding the type of injection system to be employed in the mould, use of other injection machine if available for better mould performance, etc. These modifications and suggestions are then sent to the client for approval.

4.2 Product Review and Discussion

The part sent by the client is revised and the probable areas of problems are discussed in order to solve them in the initial stages of the project. This is the stage where the technical and the commercial proposals are made. The cost of the mould and delivery time is decided and then the quotation is sent to the client.

4.3 Order

If the client approves the quotation, the order is placed. The part is then sent to the designer who starts designing the mould.

4.4 Mould Design (CAD)

There are many rules for designing moulds. Each company has its own set of regulations for mould design, for example, colour of the parts in design, nomenclature of components, etc. The design of the mould depends on the number of cavities, cavity shape, shrinkage of the part, type of runner and gates, injection pressure, characteristics of injection machine in which the mould would be placed, etc.

Number of Cavities: A lot of moulds, particularly moulds for large products, are built for only one cavity space. But if the products are small in width as compared to length, sometimes the moulds are built with 2 or more cavities (Figure 6). The reason is purely economical as it takes less time to produce more parts.

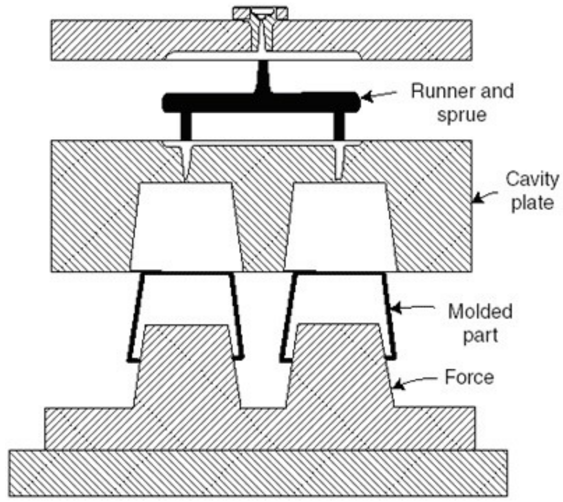


Figure 6 - Two cavity mould (Injection Mold Tech Basics, 2006)

Most of the multicavity moulds are built with either 2, 4, 6, 8, 12, 16, 24 number of cavities. These numbers are selected because the cavities can be easily arranged in a rectangular pattern. A small number of cavities can also be placed in a circular pattern with odd number of cavities, such as 3, 5, 7, 9.

Cavity Shape and Shrinkage: The shape of the cavity is essentially the 'negative' of the shape of the desired product. The dimensional allowances are added to accommodate the shrinkage of the plastic. When the plastic is injected in the mould, the plastic is in molten state, hence hot. The plastic tends to cool inside the mould to become a solid part. Every material expands as its temperature increases and returns to its original volume if cooled to its original temperature. All the plastics have the property of compressibility. When pressure is applied to the plastics, they compress significantly in proportion to the amount of pressure applied. This may be as high as 2% of the original volume. This number is the shrinkage of a material.

Parting Line: To be able to produce a mould, we must have at least two separate mould halves, one is the core and another the cavity. The separation between these plates is called the parting line (P/L). Actually it's a parting plane or area, but by convention, it is referred to as line. In Figure 7, we can see that line on a mould.

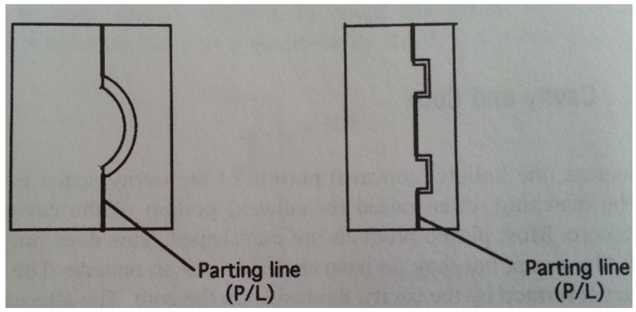


Figure 7 - Parting Line of Mould (Herbert, Understanding Product Design for Injection Molding)

The parting line can have any shape, but for ease of manufacturing, it is preferred to be a single plane. If the parting line is poorly finished, the plastic will escape from the moulding zone and will shows up on the product as **flash** (explained in Chapter 7).

Runners and Gates: Figure 8 shows a mould with sprue, runner and gate. A mould needs to have a provision for the plastic to enter the cavity at a pressure which would be enough to fill it completely before the plastic freezes. The passage from which the machine nozzle flows the melt plastic in the mould is the **sprue**, then the **runners** distribute the plastic to individual cavities and the **gates** lead the plastic from the runner into the cavity space.

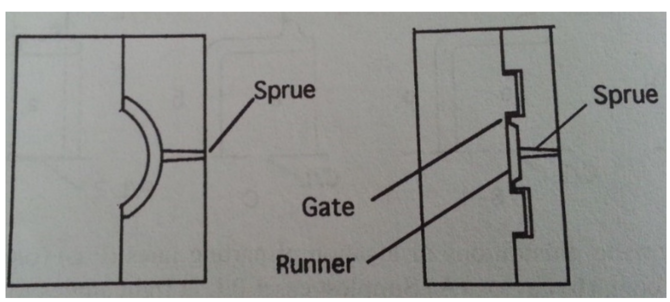


Figure 8 - Sprue, Runner and Gates of mould (Herbert, Understanding Product Design for Injection Molding)

Venting: As the plastic flows from the gate into the cavity space, the air trapped in the cavity must be allowed to escape. Typically, the trapped air is pushed towards the farthest points away from the gate by the rapidly advancing plastic. If the air does not exit from the mould, the temperature attained by the air during compression is high enough to burn the plastic. So the moulds are equipped with vents on the parting line to help the air to exit.

Cooling: After injection, the plastic has to be cooled so that the moulded piece becomes rigid enough for ejection. For a mould it is very important to remove heat efficiently. There are several rules based on which the cooling system is designed;

Rule 1: Only the moving coolant is effective in removing heat.

Rule 2: The mould must be cooled with the same coolant flow (quantity of coolant per unit of time) at a temperature that is different from that of the cavity and the core.

Rule 3: The amount of heat removed depends upon the quantity (volume) of coolant flowing through the channels. The faster the coolant flows, the better it is.

Rule 4: The coolant should flow in a turbulent flow pattern instead of a laminar flow. The turbulence causes the coolant to swirl in the channel bringing fresh and cool liquid in contact with the hot metal walls of the cooling channel.

Rule 5: The channel size (diameter and length) must be calculated carefully so that there is always more flow capacity in the preceding sections to feed all the channels in the following sections equally.

Rule 6: The difficult to cool areas should be considered first.

Rule 7: The study of the product to locate heavy sections of plastic is important. The presence of heavy sections towards the end of the plastic flow is bad because there is less pressure to endure filling. The part of the client has to be modified towards the ends to save cooling time and plastic.

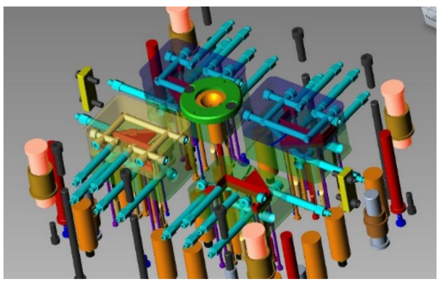


Figure 9 - Cooling circuit as designed in CAD (Hill)

Ejection: After the plastic in the cavity spaces has cooled sufficiently and is rigid enough for removal, the mould halves move apart, allowing sufficient space between them for the removal of the product. If the part is simple, then there is no need to have any provisions for the extraction of the part. A jet of air may be directed along the parting line to lift off the part from the core. But this is not possible for most of the moulds. So a system of extractors is set up.

Usually, the products are ejected by one of the following methods: pin (and sleeve) (Figure 10) and stripper plate or stripper ring (Figure 11).

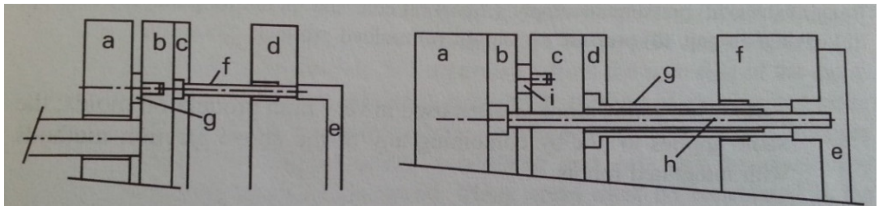


Figure 10 - (left) Section through ejector pin mould: (a) backing plate; (b) ejector plate; (c) ejector retainer pin; (d) core plate; (e) moulded part; (f) ejector pin; (g) stop pin.. (Right*) Section through sleeve ejector mould: (a) backing plate; (b) core pin retainer plate; (c) ejector plate; (d) sleeve retainer plate; (e) moulded product; (f) core plate; (g) sleeve ejector, (h) core pin; (i) stop pin (Herbert, Understanding Product Design for Injection Molding)*

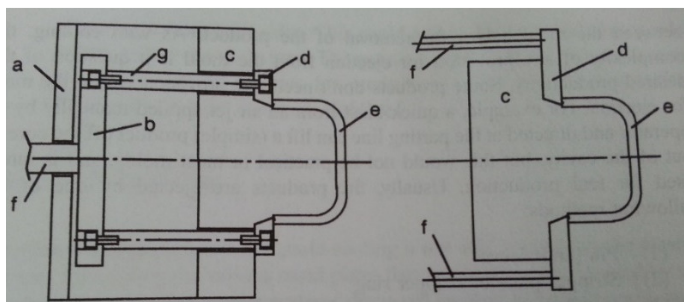


Figure 11 - (Left) Section through stripper ring mould: (a) mounting plate, (b) ejector plate, (c) core plate, (d) stripper plate, (e) moulded product, (f) machine ejector, (g) connecting sleeve. (Right*) Section through stripper plate mould: (a) mounting plate, (b) ejector plate, (c) core and mounting plate, (d) stripper plate, (e) moulded product, (f) machine ejector. (Herbert, Understanding Product Design for Injection Molding)*

Alignment: There are various methods to align the core and the cavity plates. The method of alignment depends on the shape of the product, the accuracy of the product and even on the expected mould life.

- **No Provision for Alignment**

  In case of a flat product, with core in one half and the cavity entirely in the other half, we do not need provisions for alignment. For example, in a mould of floor mat, there is no need for alignment as most of the dimensions can vary is by the amount of play between the machine tie bars and tie bar bushing.

- **Leader Pins and Bushings**

In cup shaped products with heavy walls, there is no need of alignment within the mould as the clearance between tie bars and their bushings is much less than the tolerance of product wall thickness. The main reason to have leader pins is to protect the projecting cores from any physical damage when handling the mould. The leader pins have to be always provided on the core (Figure 12).

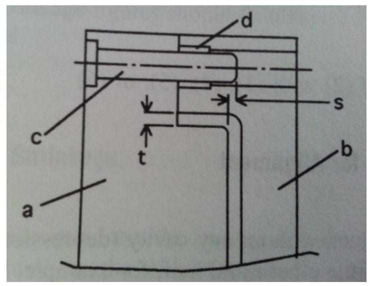


Figure 12 - Typical mould with leader pin and bushing alignment: (a) core plate; (b) cavity plate; (c) leader pin, (d) leader pin bushing, (s) safety distance of pin above core, (t) wall thickness of the plastic product at parting line. (Herbert, Understanding Product Design for Injection Molding)

- **Taper Lock**

Figure 13 shows three possible configurations of taper or wedge locks. On the left, the tapers in male and female member perfectly match. But because of manufacturing tolerances, it is impossible to achieve such precision and accuracy. To solve this problem, the male member is dimensioned a little larger than the female member and the male member is adjusted to align with the female member. The amount that the pieces stay apart before final clamping (d) is called preload in Figure 13 (centre). On the right, the female member is larger than the male member. This taper lock is useless because the tapers don’t touch each other.

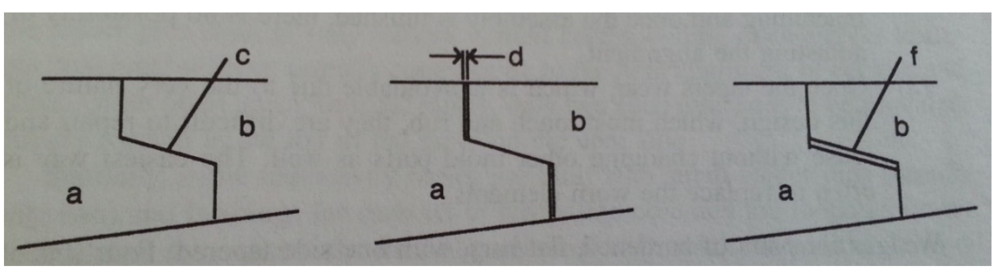


*Figure 13 - Taper lock: (a) male member, (b) female member, (c) taper. (*Left*) Ideal condition. (*Center*) Correct Application. d is the preload. (*Right*) Useless taper (Herbert, Understanding Product Design for Injection Molding)*

4.5 Mould Production

After the mould is designed, the design is sent for the programming of machining patterns. First of all, the mould design is sliced into layers, e.g.: 10 layers, and each layer has a specified path of motion for the tool. The tool for machining is selected depending on the height of the material to be removed and the radius of the fillet along the adjacent edges.

Primarily, the roughing of the steel/aluminium slab is done in 3-axis or 4-axis machines and subsequent finishing stages are carried out usually in either 5-axis or 6-axis machines. The machining of mould parts is done in CNC machines (Figure 14*(left)*), conventional lathes or mills, hobbing machines and in EDM machines (Figure 14(*right)*).

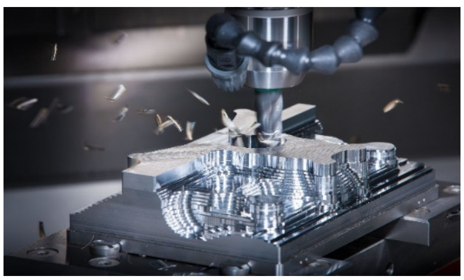


Figure 14 - (left) CNC machine (Aleck, 2015), (right) EDM machine (Gestion De Compras)

EDM machining is a complex process. Initially, the graphite tool for machining is designed which is the exact negative of the shape to be machined. The tool is then machined in a CNC machine. This tool is then inspected for dimensional accuracy as the process of EDM is very expensive as well as precise. The tool is then fixed in the EDM machine. The mould is lowered in a dielectric fluid and the graphite tool is lowered to the location of machining. Enough gap is maintained to develop an electric arc which is responsible for the removal of material from the mould.

Hole Making and Drilling: Hole making and drilling are typical machine operation which are done in lathes, drilling machines and machining centre. Hole making can be classified as:-

- Drilling
- Trepanning

- Core drilling
- Reaming
- Boring
- Deep hole drilling
- Step and face drilling

The drills are usually made with provisions for breaking and conveying of chips. The drilling operation requires a cutting fluid as coolant. Cutting fluid makes the chip formation easier, decreases friction and helps in removing the chips. Cutting fluid is provided by the CNC machine at the point of machining, but nowadays drills are being made with the provision of flowing cutting fluid from inside of the drills. The common drills are:-

- HSS – high speed steel drills, diameter 0.15 – 100mm, with an angle of 120 degree
- Carbide brazed drills, diameter 9.5 – 30.4mm, with an angle of 140 degree
- Solid carbide drills, diameter 0.3 – 20mm
- Indexable insert drills (U-drills), diameter 12 - 110mm

Drilling is started on a surface which is perpendicular to the axis of drilling, otherwise the drill may float. To assure accurate positioning, a pilot hole is done by a rigid short hole drill. The pilot hole makes it easier and safer to continue with long drills.

Deep Hole Drilling is done where the hole depth is 3 to 5 times larger than its diameter (D). The length can be even 150 times the diameter. Deep hole gun drilling produces a better surface quality and surface roughness (DS, Deep Hole Drilling, 2008). The following factors affect the machining:-

- Diameter and tolerance of the hole;
- The length and the shape of bottom of hole;
- Surface roughness;
- The workpiece material to be drilled;
- Cutting conditions;
- Disturbance;
- Productivity;

Deep hole drilling can be done in three different ways:-

Use of Traditional Drills (Figure 15)**:** This type of drilling is done in cycles. The drill is pulled out at intervals so that the chips from the hole flush out each time the drill is retracted. This cycle is continued, for example, to a depth of 2.5 x D, 6 x D and 10 x D where D is the diameter of the hole. Long carbide drills are more efficient than gun drills.

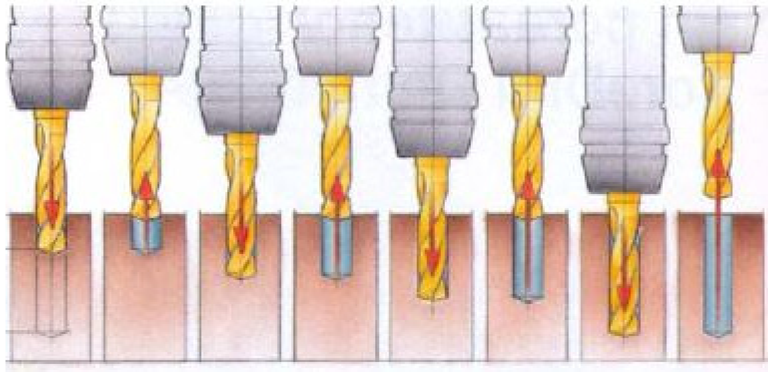


Figure 15 - Deep hole drilling cycle by traditional drilling tools. (DS, Deep Hole Drilling, 2008)

Use of Tube Type Tools (Figure 16)**:** This type of machining is done when the work piece is in a horizontal position. Special machine tools are needed where both the workpiece and tool rotates. Cutting fluid is injected through the tool shank in this machining.

Figure 16 - A special machine tool with oil injection for horizontal deep hole drilling by ejector system. (DS, Deep Hole Drilling, 2008)

Holes of large diameter are produced by using trepanning or solid drilling. Indexable inserts and brazed drilling heads are used for this kind of machining. This type of drilling is a continuous process.

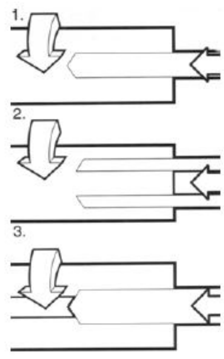


Figure 17 - Three different kinds of deep hole drilling methods – 1. Solid drilling, 2. Trepanning, 3. Counterboring a pre drilled hole. (DS, Deep Hole Drilling, 2008)

Use of Gun Drills in a Horizontal Position: The small diameter cooling and ejection system in the moulds are made by this process. A special geometry at the gun drill tip as well as the cylindrical carbide tip of the shank stabilize the cutting process and the guiding tool. The cutting oil is pumped to the tool tip with sufficient pressure and amount through a groove in the shank. The tool shank is smaller at the tip. A long V-shape grooving outside the tool shank is used for chip convey (Figure 18).

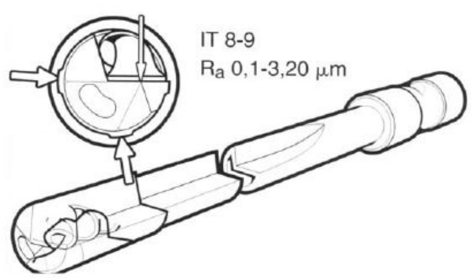


Figure 18 - A gun drill for a small diameter deep hole drilling. The chips are transferred outside the tool in a V-shape groove with the cutting oil. (DS, Deep Hole Drilling, 2008)

<u>Polishing:</u> Finishing of the moulding zone of cavity is usually not done by EDM or CNC machining as the cavity has the visible part of the product. It should have superfine surface finishing. This is dominated by manual methods. The superfine polishing methods are – lapping, lapping and polishing, and finish polishing.

Figure 19 - Polishing (Hotech Polerteknik)

Initially, the surface is prepared for polishing by roughing the surface with an abrasive disk. The abrasive disk removes the surface irregularities and gives a base for polishing. The surface is then polished with either sand paper or abrasive sticks with diamond paste till the surface is smooth and there are no lines on it. A well-polished cavity helps in enhancing the cosmetic value of the part and also helps in part release.

4.6 Assembly, Adjustment and Connections

Once all the sliders, lifters, extractors and mould core are machined, the fitting and fixing of the parts is done. The parts are fixed with the help of screws and nuts. After the mould is assembled, it is sent for the adjustment of parts.

The sliders, lifters and extractors are adjusted to obtain free movement in and out of the mould. In this process, it is also important to make sure that there are no gaps between the mould core and these parts as the plastic will flow in cracks and crevices and form flash. After the core is adjusted and reassembled, the cavity mould is adjusted.

The cavity adjustment is done to ensure the complete closing of the mould. This is done with the use steel strips of sizes as thin as 1/8th of an inch. The moulding zone of the core and cavity is never adjusted as it can affect the final part dimensions.

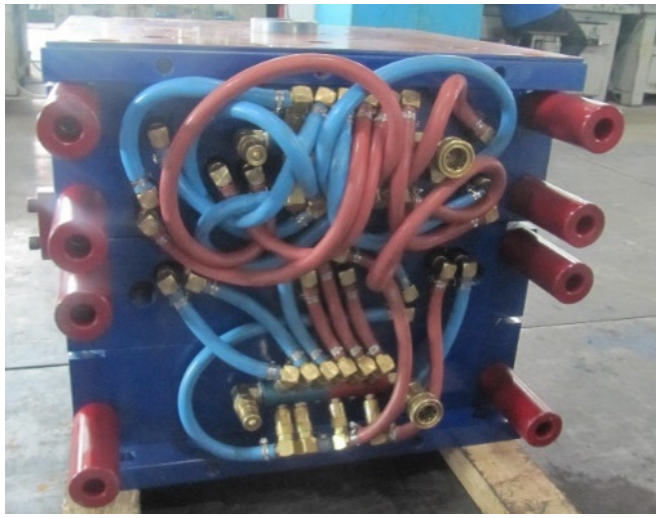


Figure 20 - Cooling system connections (CN Moulding)

Once the adjustments are finished, the cooling system (Figure 20) and oil channels to trigger hydraulics/pneumatics are assembled on the mould. These connections are then checked for functioning and leakage.

4.7 Trail Injection-1

The mould is clamped to the injection machine and the first tryout of the mould is carried on. In this trial, the functioning of the mould, part release, optimum parameters of injection and cosmetic value of the part are observed. Usually the parameter selection is based on the experience of the machine operator or the official who is supervising the injection process.

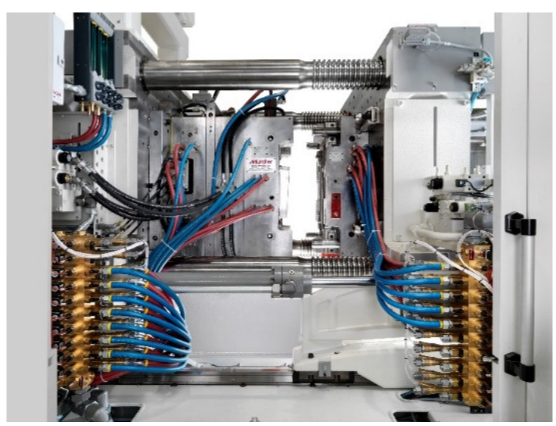


Figure 21 - Mould trail in injection machine (Maffei, 2013)

4.8 Product Evaluation and Modification of the Mould

The parts/products of trials are evaluated for weight, dimensions and quality. Based on the faults in the product, the mould may be modified and adjusted and again sent for trials. This process is repeated until the right product specification are obtained.

4.9 Mould Approval and Delivery

When the mould is operating properly and the parts produced are fulfilling the customer requirement, the mould is delivered to the client. The mould is painted for protection.

5. Standard Parts of a Mould

At the most basic level, the mould consists of two main parts: the cavity and the core. The core forms the main internal surfaces of the part. The cavity forms the external surfaces. The standard parts of a mould can be grouped under the following categories:-

- Standard mould set;
- Ejector pins and components for ejection;
- Ejector set guiding elements;
- Core pins;
- Core moving elements;
- Cold runner system components;
- Hot runner system components;
- Tempering device fixtures, insulating elements;
- Fastening and closing elements, elements for lifting the mould;
- Feeding system of the mould.

5.1 Standard Mould Set

A standard mould set consists of two clamping plates, a cavity plate, a core plate, an optional back plate, risers and an ejector set (Figure 22). The cavity and the core are made on terms of 3D-CAD design. Most of the cavities are usually placed on the fixed side and the core on the moving side of the mould. This assures that the part is on the ejection side when the mould opens.

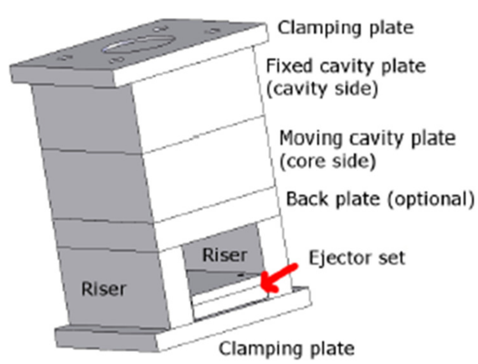


Figure 22 - Standard Mould Schematic (DS, Standard Components, 2008)

The ejector set consists of two plates: an ejector retaining plate and an ejector plate. These are fastened together with four bolts and the foot of the ejectors is placed between them. Usually these act as buffer plates between the ejector base plate and the clamping plate (Figure 23).The buffer plate maintain a gap between ejector base plate and clamping plate in order to avoid crashing.

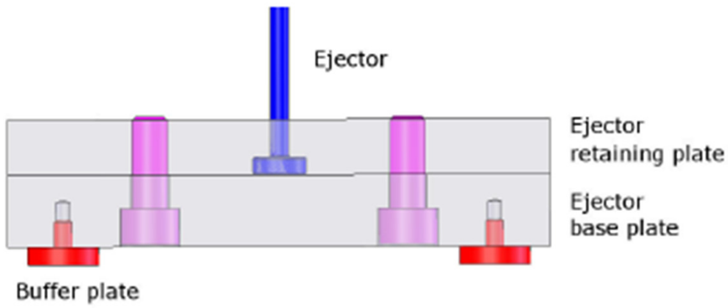


Figure 23 - Ejector Set Components (DS, Standard Components, 2008)

Moving and fixed mould halves are guided towards each other with the help of guiding elements. Basic guiding elements are guiding pillars, guiding sleeves and centring sleeves (Figure 24).

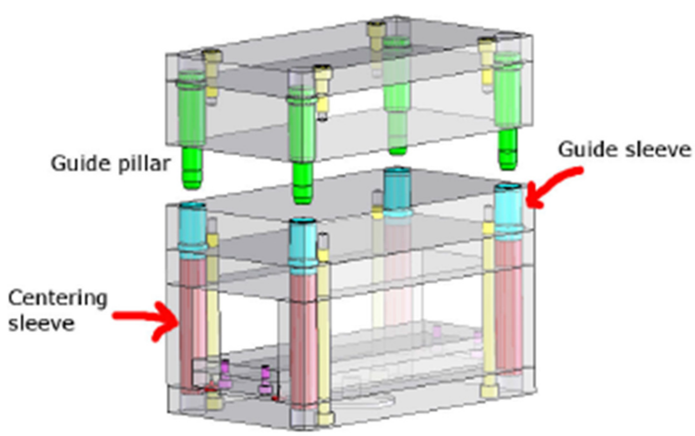


Figure 24 - Basic Guiding Elements (DS, Standard Components, 2008)

5.2 Ejector Pins and Components for Ejection

The basic parts in the mould ejection system are ejectors and spruce pullers. There are different elements in an ejector system - round ejector pins or flat ejector pins, ejector sleeves, etc. Sprue pullers are specially shaped or specially placed ejectors, which stick to the sprue and pull it out from the sprue bushing (Figure 25).

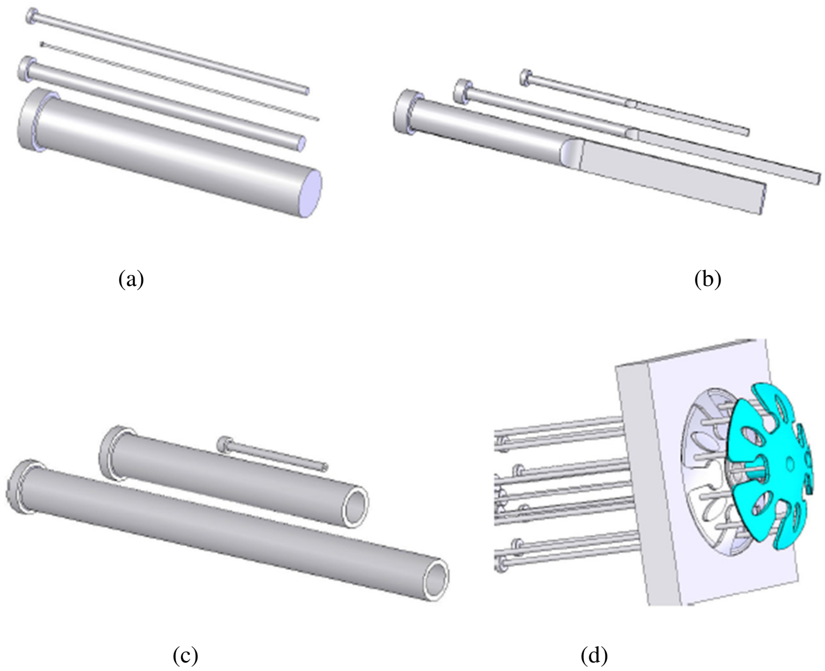


Figure 25 - (a) Round ejector pins; (b) Flat ejector pins; (c) Ejector sleeves and (d) Ejected Part (DS, Standard Components, 2008)

Fixed Position Ejectors: These ejectors are attached under the ejector set plate with a collar at the end of the ejector. The collar of such ejectors are not round, but are machined to be semi-circular. This flat surface of the collar puts the ejectors to a fixed position (Figure 26).

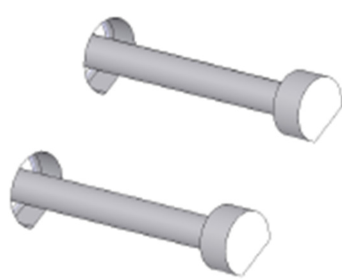


Figure 26 - Fixed Position Ejectors (DS, Standard Components, 2008)

Sprue Pullers: The sprue pullers are specially designed ejectors. In Figure 27(a), we can see the mould with a sprue puller sleeve. In Figure 27(b), a sprue puller sleeve structure, a cross-section of part of runner forms inside the sprue puller sleeve. Figure 27(c) shows the ejected part with the sprue puller.

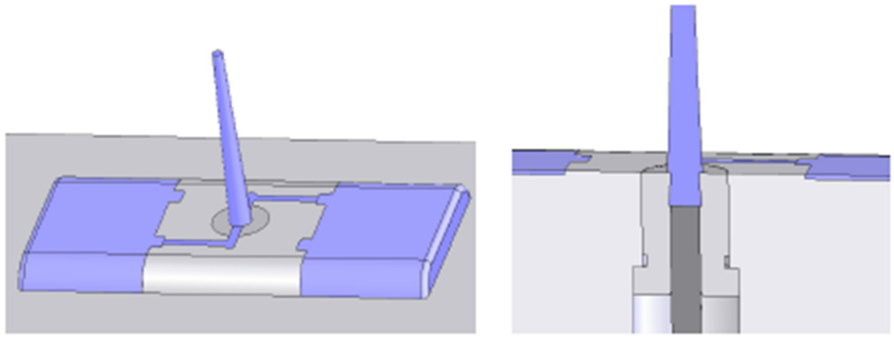


Figure 27 - (a) Mould with sprue puller; (b) Sprue puller with runner (DS, Standard Components, 2008)

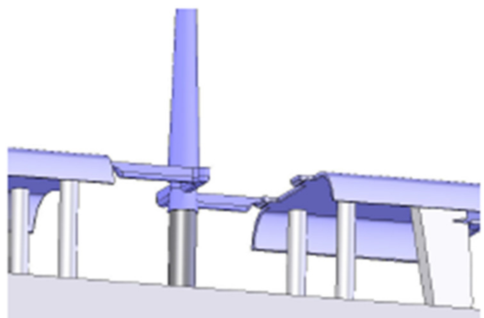


Figure 27 - (c) Ejected Part with Sprue puller (DS, Standard Components, 2008)

Tilting Ejectors: The tilting ejectors are used in areas of mould where there is a small negative draft on the part like in places of snap fit and small slots (Figure 28).

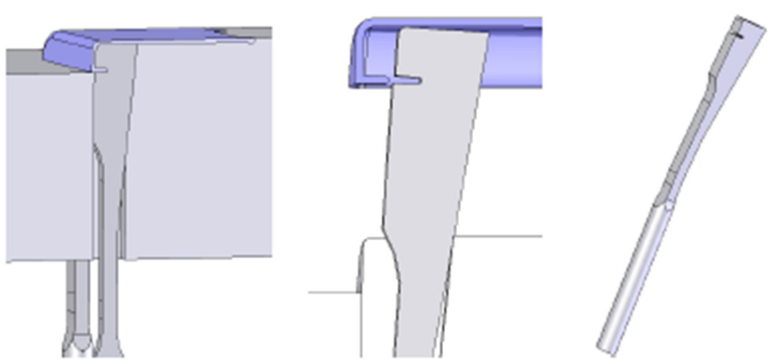


Figure 28 - Tilting Ejector (DS, Standard Components, 2008)

5.3 Ejector Set Guiding Elements

The mould ejector set is fixed to the moulding machine with an element in the centre of the mould. Without guides, the ejector pins are the sole elements which support these ejector plates along with the fixture. The ejector set is under a lot of stress and strain while moving front and back during the ejection of the part. The ejection pins are not stiff and strong enough to resist this amount of loading. Even the smallest imbalance can produce bends and, in the worst case, break the pins. Hence, the ejector set is fixed to the moulding machine with the support of four ejection bars. Four bars give a rather good strength, but it is still possible that the ejector set succumbs to high stress and loading. These ejector set plates work normally on hydraulics. The ejector plate is sent to its initial position with the cylinder back position movement. In some moulds the backward motion of the ejector plate is assisted by the action of a spring.

Returning Pins: Returning pins are the thick ejector pins which are placed outside the cavity and extended to the mould parting surface. If the ejector set returning system does not work properly, these pins secure the ejector set to the initial position (Figure 29).

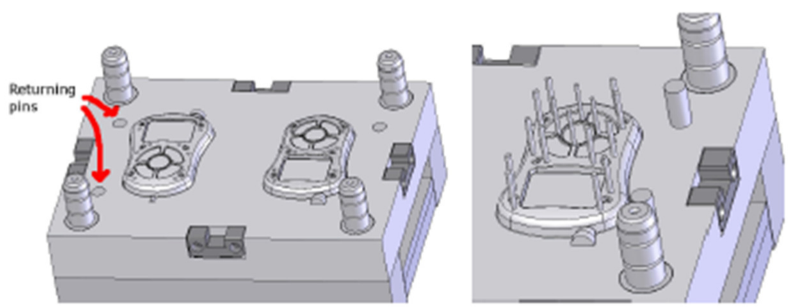


Figure 29 - Return pins in back and front position (DS, Standard Components, 2008)

Ejector Guiding System: If it is necessary to guide the ejector set more reliably and accurately, guide pillars and sleeves are used with or without bearings to assist the movement. (Figure 30).

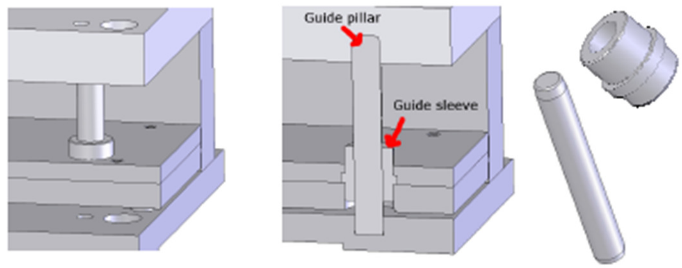


Figure 30 - Ejector Guiding System (DS, Standard Components, 2008)

5.4 Cavity Support Element

Usually the ejection side mould cavity is supported with supporting pillars. The fixed cavity plate has enough support from the moulding machine base, but the ejection side cavity plate faces an open ejection box construction. The supporting pillar is placed between the clamping plate and the cavity back plate under the cavity (Figure 31).

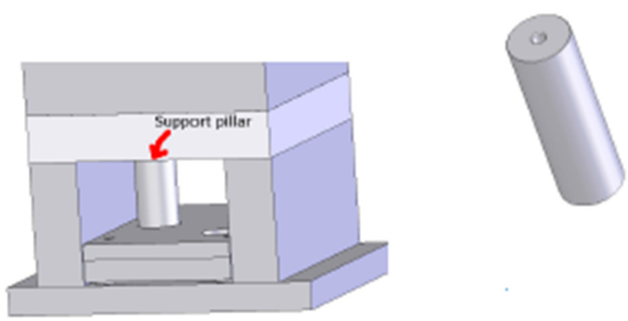


Figure 31 - Cavity Support Pillar (DS, Standard Components, 2008)

5.5 Core Pins

Core pins are similar to the ejector pins, but the material is different. The core pins are placed to the mould cavity to shape deep and narrow holes in the part (Figure 32).

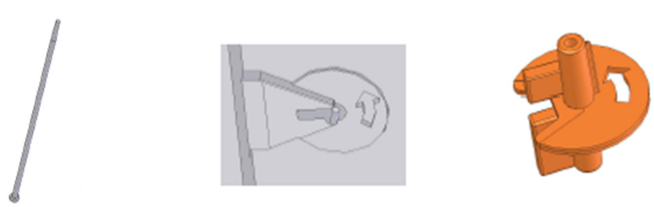


Figure 32 - (a) Core pin; (b) Core pin in mould cavity; (c) Moulded part (DS, Standard Components, 2008)

5.6 Core Moving Elements

Core moving elements may be powered by different hydraulic, pneumatic or electric equipment or slide mechanisms, which move the moving core out of the mould cavity to enable part ejection.

Slide Mechanism: The slide mechanism is purely mechanical, driven by the mould opening movement. The main parts in the slide mechanism (Figure 33 and Figure 34) are angle pin, slide locking parts, slide guiding and slide. Some standard manufacturers sell complete slide sets with all the necessary parts in them.

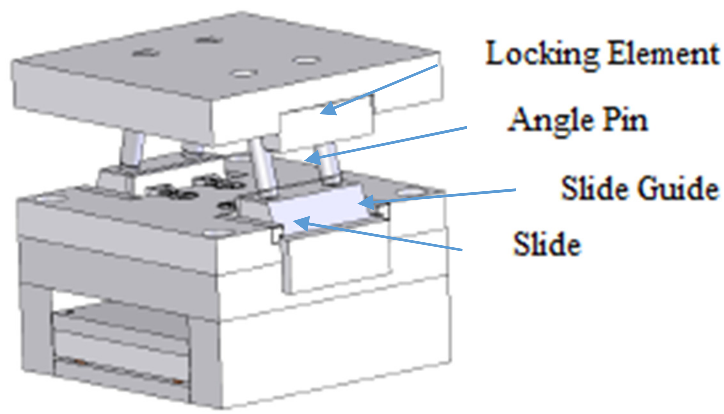


Figure 33 - Slide Mechanism (DS, Standard Components, 2008)

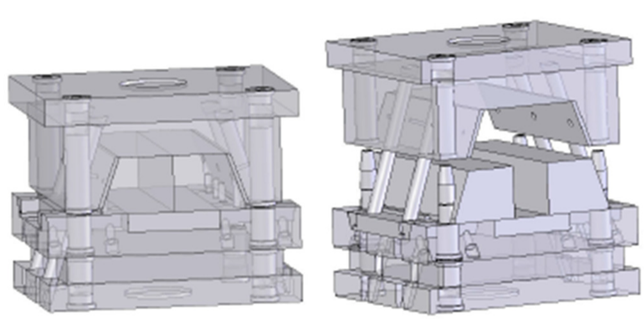


Figure 34 - Slide Mechanism in action (DS, Standard Components, 2008)

Core Pulling Cylinders: If the moving core is relatively long, it is more practical to use core pulling cylinders because normally in mechanical systems, the side opening movement is restricted by the angle pin length. In average size moulds, the typical core movement is less than 50 mm. Core pulling cylinder strokes vary between 100-250 mm, even 300 mm is sometimes required. Figure 35 shows one type of core pulling cylinder.

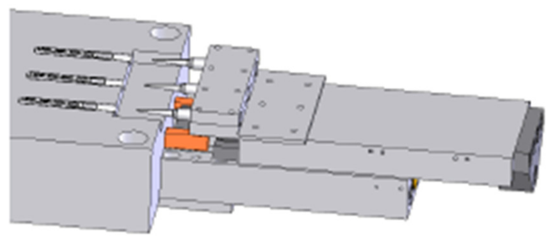


Figure 35 - Core pulling cylinder (DS, Standard Components, 2008)

5.7 Runner System Components

Runner System: The purpose of the runner system is to provide the melted plastic a route from the injection machine nozzle to the mould cavity. There are basically three types of runner systems: cold runner, hot runner and combined runner systems. When using the cold runner systems, the cold runner is removed with the part from the mould because the plastic solidifies inside the cold runner system. Whereas, the hot runner system keeps the plastic in liquid form and only the part comes out.

Sprue Bushing and Centring Ring: A cold runner system consists of a central gate (sprue bushing), runners, gates and ingates. Runners, gates and ingates are mainly machined channels in the mould plates. The most common standard cold runner system parts are sprue bushings and rings for centring the injection machine nozzle to the sprue head (Figure 36 and Figure 37).

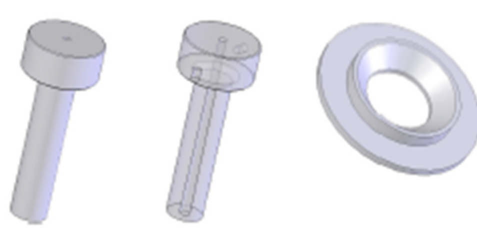


Figure 36 - Sprue bushing and Centring Ring (DS, Standard Components, 2008)

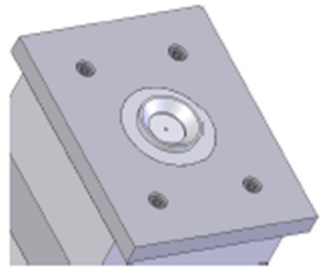


Figure 37 - Assembly of Centring ring and sprue bushing with mould (DS, Standard Components, 2008)

Hot runner system consists of a sprue bushing, manifold, nozzles and gate bushings. There are basically three hot runner system types: externally heated system, internally heated system and combined system. In Figure 38, Figure 39 and Figure 40, there are some typical components of an externally heated system.

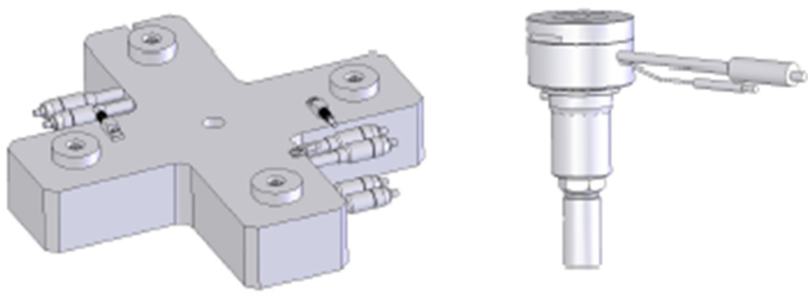


Figure 38 - Manifold and heated nozzle (DS, Standard Components, 2008)

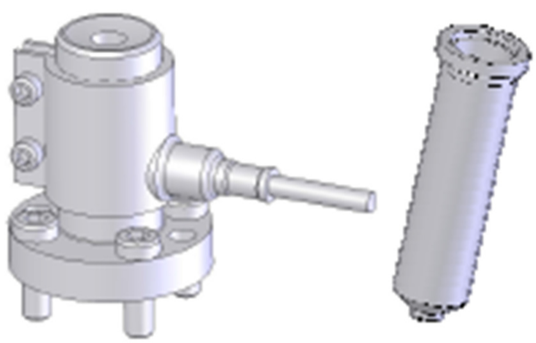


Figure 39 - Heated sprue bushing and Gate bushing (DS, Standard Components, 2008)

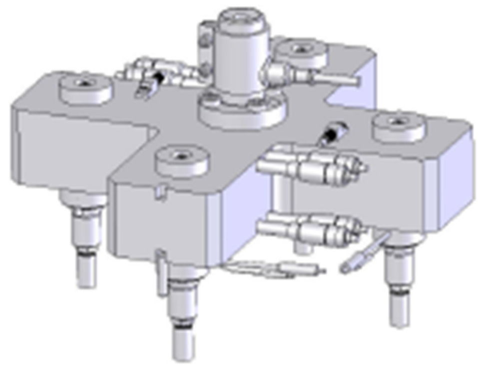


Figure 40 - Hot runner system assembly (DS, Standard Components, 2008)

An internally heated system is constructed with similar components. The difference is in the heating the channels.

5.8 Feeding System of the Mould

The feeding system consists of a series of channels generally machined in one or more plates of the mould through which the melt is transferred from the nozzle to each moulding zone. The entry of the molten plastic in the moulding area is done through **gates**.

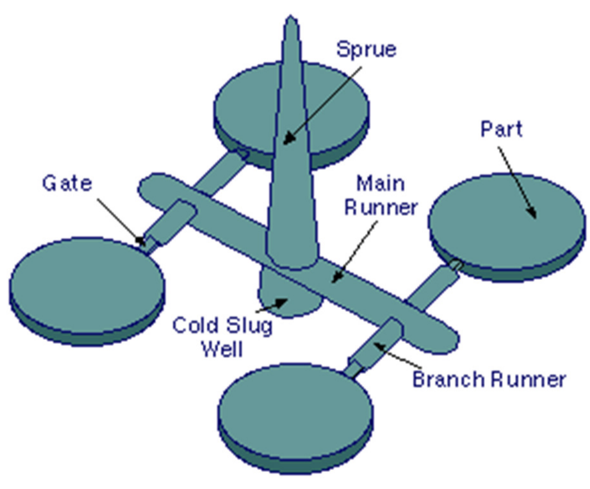


Figure 41 - Parts of feeding system (Capetronics, 2012)

The **sprue** is a conical channel which connects the injection nozzle feeder to the moulding area. The sprue usually has an opening angle of 2 to 5°. This taper is necessary to facilitate its extraction. Typically, this component is not machined directly in the mould plate but is accompanied with an ejector.

5.8.1 Types of Runners

The runners can be classified according to their cross-sections (Figure 42): Full Round Runner, Trapezoidal Runner, Modified Trapezoidal Runner, Half Round Runner and Rectangular Runner.

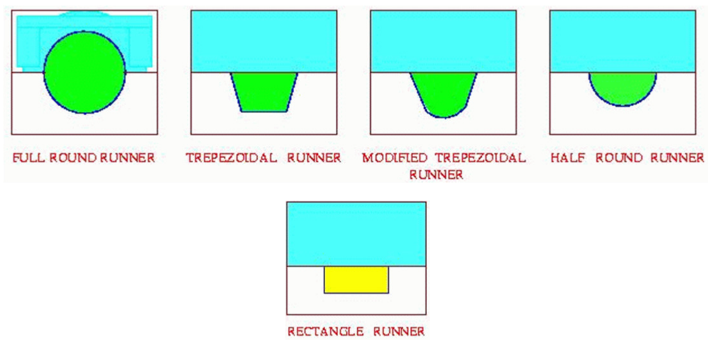


Figure 42 - Types of Runners (Prasad, 1999)

The Full Round Runners are the most efficient type of channels. The flow resistance of this type of channels is relatively smaller as compared to others. The melt temperature drop is also smaller. The only disadvantage is that these types of runners have to be machined in both mould halves symmetrically. On the other hand, the Modified Trapezoidal Runner are almost as good as the Full Round Runners but have an advantage of being machined on only one half of the mould. Thus, this is the most widely used type of runner as these have the best cost-benefit relation. The channels with Half Round and Rectangular Runners should be avoided wherever possible.

5.8.2 Types of Gates

Direct Gate: This type of gate is used for single cavity moulds. The injection pressure and the feeding time required is less when using this gate. But the direct gates are difficult to remove and some big gates leave marks on the parts.

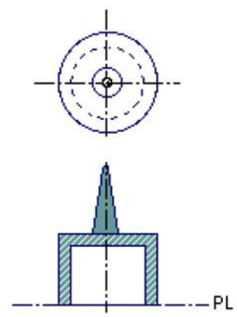


Figure 43 - Direct Gate (Injection Molding Gate Types)

Tab Gate: Tab gate is used on the narrow side of long plastic parts and have a thickness of 3/4th of wall thickness. This gate design reduces the shear stress generated around the gate.

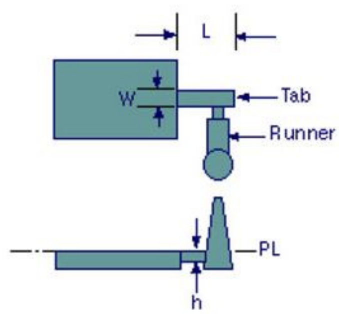


Figure 44 - Tab Gate (Injection Molding Gate Types)

Fan Gate: This gate permits the flow of melt plastic into the cavity through a wide opening and is used to create stable flow into wide parts. This gate helps to maintain stability and to avoid deformation.

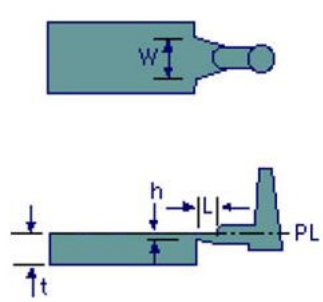


Figure 45 - Fan Type Gate (Injection Molding Gate Types)

Pin Gate: This is used specifically in 3-plate moulds. The runner channel is located in a different plate and the mould flow is divided into several directions, leading into the cavity by several gate locations. The gate point is designed to be very small and is allowed to be trimmed off by the action of mould opening.

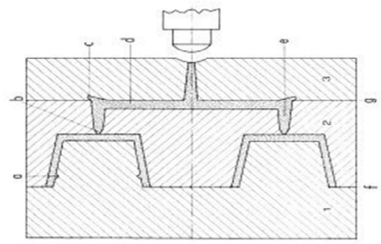


Figure 46 - Pin Gate (Design of Gates)

Sub (Tunnel) Gate: This is used for 2-plate moulds. The gate point is small and the area of gate is place at an uncritical area. This gate can also be easily trimmed off during mould opening action.

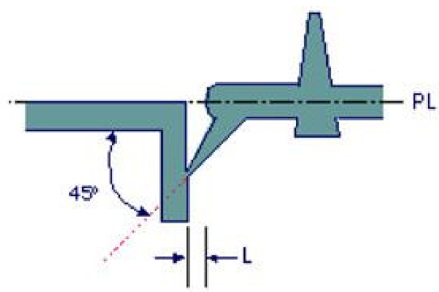


Figure 47 - Sub (Tunnel) Gate (Injection Molding Gate Types)]

5.9 Tempering Channel Fixtures and Other Components

Mould tempering is typically done with straight/conformal cooling channels. The cooling liquid is water. Sometimes oil is used to enable the operation of the mould at extremely high pressure.

Mould manufacturers have different systems and parts for building the tempering channels. These parts include fittings for connecting the tempering machine to the channels with pies and also for connecting channels to each other (Figure 48).

Figure 48 - Fitting for connecting tempering pipes to each other and to machine (DS, Standard Components, 2008)

There are also standard parts and fittings to cascade the tempering liquid to strengthen the cooling effect. These fittings include cascading liquid junctions, cooling baffles and cooling cores (Figure 49). Typically, these systems are placed inside narrow cores.

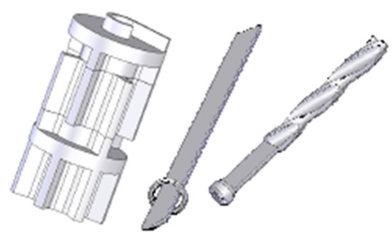


Figure 49 - Cooling core, straight baffle and spiral baffle. (DS, Standard Components, 2008)

5.10 Fastening and Closing Elements, Elements for Lifting

Moulds are fastened to the machine platens with clamps and screws. The injection machine plates are typically equipped with T-slots and with threaded holes.

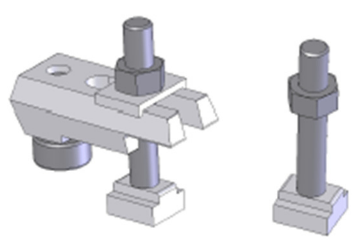


Figure 50 - Clamping assembly on Injection Machine (DS, Standard Components, 2008)

Some manufacturers have different quick-clamping systems like magnetic and hydraulic clamping. Moulds are generally lifted to the machine with a crane. To make short work of the lifter, the mould are equipped with at least two pair of eyebolts (Figure 51).

Figure 51 - Eyebolt for lifting mould (DS, Standard Components, 2008)

Ejector Clamps: Injection machines typically have several different ways of clamping the mould ejection plates to the machine ejection system. There is only one point of attachment, but the parts for attachment are numerous. The simplest clamping system is one ejector rod (Figure 52). But there are also different couplings for pneumatic or mechanical quick clamping devices (Figure 53).

Figure 52 - Ejector Rod (DS, Standard Components, 2008)

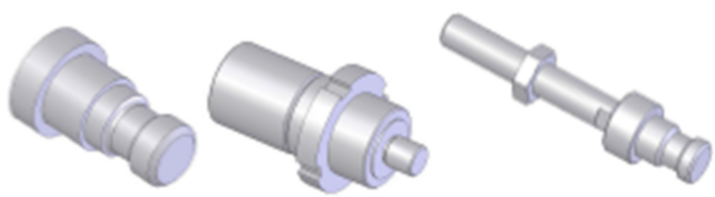


Figure 53 - Couplings for different ejector clamping systems (DS, Standard Components, 2008)

6. Factors Influencing Plastic Injection Moulding

The injection process should ensure the proper development of the product with smooth material filling and good mechanical properties. To accomplish this, the important parameters to be controlled are temperature (melt temperature, mould temperature), pressure (plasticizing/back pressure, injection pressure, holding pressure) and time (clamping time, injection time, holding time, cooling time). (Tayu, 2011)

6.1 Temperature

Temperature has a significant influence on the final properties of the material regardless of part design. The two most important process conditions which have a huge impact on the behaviour of polymer are melt temperature and mould temperature.

Melt Temperature - Melt temperature is the temperature of the polymer as it exits the injection machine nozzle and enters the mould. The correct choice of melt temperature is related to the quality and type of the plastic. The heating temperature of plastic should always be higher than the plastic flow temperature (melting point), but below its decomposition temperature.

Mould Temperature – Mould temperature is controlled by the coolant flowing through the channels in the mould. This determines the cooling rate of the injected plastic.

Importance of Melt and Mould Temperature (Sepe, 2011)

It is generally considered that the melt temperature has an influence on the viscosity. But melt temperature also has an effect on the final molecular weight of the polymer in the moulded part. For example, parts moulded at higher melt temperatures tend to have more molecular weight. This translates to better impact resistance if the parts are moulded in shorter cycle times.

Mould temperature has a more profound effect on final parts. In amorphous polymers like ABS (Acrylonitrile butadiene styrene) and polycarbonates higher mould temperature produce lower levels of stresses in the part and, consequently, better impact resistance, crack resistance and fatigue performance.

In semi-crystalline materials, mould temperature is an important factor in determining the crystallinity of the moulded part. The crystallinity governs many parameters like creep resistance, wear resistance and dimensional stability at elevated temperatures. Crystals can only form if the melt temperature is below the melting point and above the glass transition temperature of the polymer.

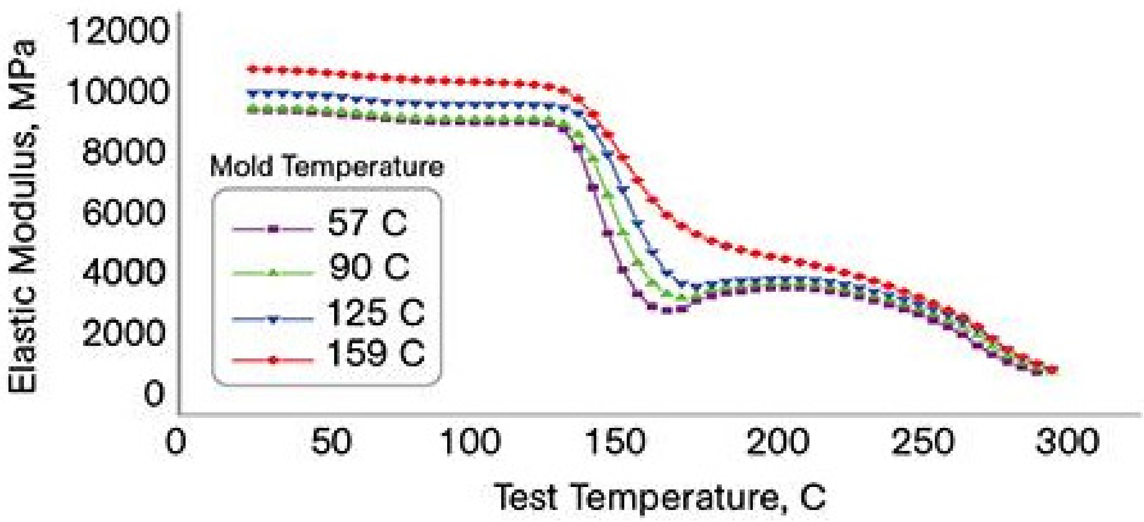


Figure 54 - Effect of Mould Temperature on Elastic Modulus vs Temperature behaviour of PPA (Sepe, 2011)

Figure 54 compares the behaviour of high temperature nylon (PPA) when moulded at a proper mould temperature and lower temperatures. It is evident from the plot that the modulus of the material is a function of the temperature. As the mould temperature increases, the stiffness of the material at room temperature also increases. As the material approaches the glass transition temperature at 130°C to 140°C, the modulus begins to decline at lower temperatures and falls faster at much lower temperatures.

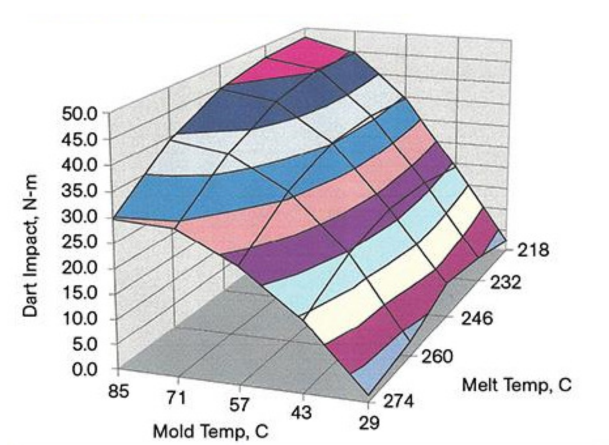


Figure 55 - Effect of Mould and Melt Temperature on Impact Performance of ABS (Sepe, 2011)

Figure 55 shows the interaction of mould and melt temperatures in determining the impact performance of ABS. It is surprising that the dart impact resistance ranges from 2Nm to almost 50Nm over a small variations in mould and melt temperatures.

The mould temperature is a dominating factor. However, the best results can be obtained when higher mould temperatures are combined with lower melt temperatures.

6.2 Pressure

Plasticizing / Back Pressure - The resistance of the molten plastic to flow forward is known as back pressure. Typically, the back pressure is around 7-10 bar, thought it may be as low as 3.5 bar. Sometimes, back pressure is increased as much as 28 bars to improve melt uniformity, increase melt temperature or to eliminate air traps.

Injection Pressure –Injection pressure is the pressure given to the molten plastic by the screw when the screw is moving forward to push the plastic in the mould. If the injection pressure is high, the flowability of the plastic is increased and if the injection pressure is low, the flowability is decreased and problems like bubbles and voids will arise.

Holding Pressure – Even when the cavity is full, more pressure is given to melt plastic in order to pack the space completely. This pressure is called holding Pressure.

6.3 Time

The time needed for one injection cycle is called cycle time. The cycle time includes clamping time, injection time, holding time and cooling time.

Clamping Time – Clamping time is the time taken to clamp the mould in the injection moulding machine. If the clamping time is too long then the mould temperature will be too low while the plastic stays in the nozzle for a long time. If the clamping time is too short, mould temperature will be too high.

Injection time – Injection time is the time from when the plastic starts to melt to the time the plastic is filled in the cavity of the mould. Normally, the injection time for small parts is around 3-5 seconds and for large parts is around 10 seconds. Injection time always has inverse effect on the injection speed.

Holding Time – Holding time refers to time for continuing pressure on the plastic product. If the holding time is too short, the plastic product will not be tight enough and this may cause dents and unstable size. If the holding time is too long, stresses in the product will be increased.

Cooling Time – Cooling time runs from the finish of the holding pressure to the opening of the mould. Cooling time depends on the thickness of the product, thermal properties of the plastic and crystallization characteristics.

Gate Seal Experiment (Bozzelli, 2010)

A major part of optimizing any moulding part is determining the second stage injection, or pack and hold parameters, i.e., if we need to 'hold' until the gate freezes or not. This is a critical variable. The decision of how much time to hold depends largely on the way the molecules of plastic behave. Usually at higher temperatures the molecules of the material are undisturbed and the part works perfectly. But, at lower temperatures the part begins to shrink and eventually develops cracks and can even break. This happens because the 'hold time' for the part was too long and the gate froze quickly resulting in the molecules being packed very tightly. When the temperature changes, the molecules gain or lose energy, thereby enabling them to move from one place to another. Even though this process occurs at a microscopic level, the unavailability of space for the movement of molecules produces internal stresses.

The best way to find the optimum hold time for a particular part is to test that part with and without the gate sealed. First, we need to find the pressure at which the part looks okay. Then we change the second stage time and add and subtract the difference to/from the cooling or mould close timing to keep the same cycle time. Then we have to weigh the part to four significant figures. We repeat the steps until the weight of the parts stop increasing. This time for which the weight of the parts is constant will be the optimum holding time for that part.

For example, John Bozzelli (Bozzelli, 2010) carried out an experiment with a specimen and the results are shown in Figure 56.

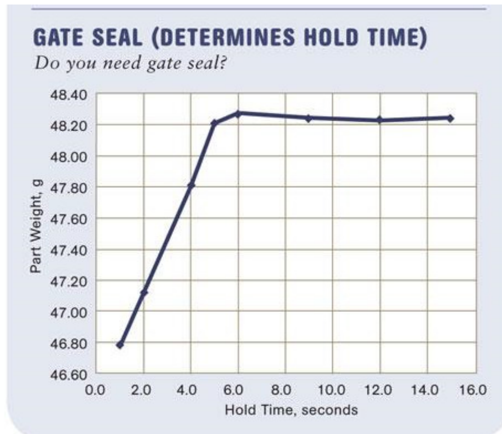


Figure 56 - Experimental graph (Bozzelli, 2010)

As we can see, the weight of the part is constant after a 9 seconds, so the holding time would be 10 seconds, one second is added to ensure that the gate freezes completely.

7. Quality Control in Injection Moulded Parts

Parts made in a plastic injection moulding process can have their own unique set of possible defects. The following is the listing of the most common defects associated with the plastic injection process:-

Sink Marks: Sinking is caused by the outer skin of plastic solidifying while the material inside is still molten and viscous. As it cools and solidifies, the material compacts. The best way to avoid these dimples is to design the part with a consistent wall thickness. However, many times the problem of sinking can be taken care of by adjusting the process parameters only.

Figure 57 - Sink Marks (Rebling Plastics)

Gas Marks: When molten plastic is injected into a mould cavity, the air that is inside needs a place to escape. If the hot compressed air cannot escape, it may leave burn mark on the part. The best way to avoid gas marks is to strategically locate air vents of the proper depth within the mould. Adjustments to process parameters such as injection speed and screw speed can be altered to reduce or eliminate burning.

Figure 58 - Gas Marks (Rebling Plastics)

Flash: When the two halves of a mould come together, the sides of the cavities must press together tight enough so that no molten plastic leaks out of the cavity, resulting in a thin layer of plastic called flash protruding from the part. The plastic injection moulding press must have enough force, and the mould must be properly built and maintained to avoid this problem.

However, some materials flash more easily than others. In fact, most thermoset materials will flash regardless of the press and mould.

Figure 59 - Flash (Rebling Plastics)

Knit Lines (Weld Lines): If a part has a hole or insert, the molten plastic will flow around the opening and meet on the other side. Where the two sides meet, the part will be weaker, and a knit line may be visible. Often this effect can be minimized by adjusting processing parameters such as injection speed and mould temperature.

Figure 60 - Knit Lines and Short Shot (Rebling Plastics)

Short Shots: If too little material flows into the mould, the cavity will not fill properly, which we refer to as a short shot. This can be corrected by changing the plastic injection moulding parameters.

Flow Marks: If molten plastic does not properly flow as it fills the cavity, flow marks may result. This can often be fixed by changing moulding parameters or adjusting the mould by changing the gate location or size.

Splay: Bubbles may flow along the part surface during the plastic injection moulding process, which can cause marks on the part surface. Proper mould design and processing parameters can prevent splay.

Figure 61 - Splay (Rebling Plastics)

PART II

STUDY CONCERNING THE THREATS AND OPPORTUNITIES POSED BY ADDITIVE MANUFACTURING TO MOLDING INDUSTRY

8. Additive Manufacturing

Additive manufacturing (AM) is the process of joining materials to make objects directly from a three dimensional model data, layer by layer. Unlike traditional processes, in which material is removed from a larger rough form causing wastage of material, additive manufacturing reduces the wastage of materials to a large extent. Commonly known as "3D printing"; AM's synonyms include rapid prototyping, additive fabrication, additive process, additive techniques, additive layer manufacturing, layer manufacturing, freeform fabrication, solid freeform fabrication and direct digital manufacturing. AM provides a cost-effective and time efficient way to produce low-volume, customized products with complicated geometries.

The parts produced by AM techniques have dimensional accuracy and good cosmetic value. But the question is, can AM replace injection moulding or can AM be used in collaboration with injection moulding? This section of the report reviews this question by looking into the recent trends and developments in AM techniques.

Since its inception in the mid-1980s, AM has evolved into a large group of processes, including Stereolithography (SLA), Fused Deposition Modelling (FDM), 3D Printing (3DP), Laminated Object Manufacturing (LOM), Selective Laser Sintering (SLS), Selective Laser Melting (SLM), Laser Metal Deposition (LMD), and others. Recent innovations in materials and processes are transforming 3D printing from rapid prototyping method to rapid manufacturing. AM is driving innovation because it can be used to manufacture complex geometries quickly and without any difficulty. By eliminating many constraints imposed by conventional machining, AM may bring a shift in ideology from Design for Manufacturing (DFM) to Manufacturing for Design (MFD).

9. Comparative Overview and Challenges of Additive Manufacturing

<u>Comparative Overview of AM</u> (Gao, 2015)

The distinguished features of AM are usually presented in comparison to conventional machining. The unique capabilities of AM are noted below:-

Design Flexibility: AM has an ability to produce parts of almost all kinds of complex geometrical shapes. Unlike subtractive manufacturing where we have design constraints, need of fixtures, diverse cooling, etc. AM imposes very few constraints which enable the designers to place the material in selective areas to have the desired functionality.

Cost of Geometric Complexity: AM is providing ease of designing in terms of part complexity. With AM, part complexity comes with no additional cost, as there is no need for additional tooling. The complexity can also be achieved with traditional methods like injection moulding, but the complexity of the part comes along with higher mould cost.

Dimensional Accuracy: The dimensional accuracy is a difference in qualitative and quantitative measurements of finished products to the original digital model of the product. In all kinds of traditional processes, general and specific geometrical tolerances have been established for designs to be manufactured. It will be necessary in the near future to define tolerances for parts to be manufactured by AM processes.

Need for Assemblage: With AM, it is possible to produce 'single-part assemblies' having integrated mechanisms. If produced conventionally, the complex parts are made by the assembly of different smaller parts. In AM, these parts are joined and printed and are suspended by supporting material. Later the supporting material can be removed in post-processing operations.

Time and Cost Efficiency in Production Run: Some traditional manufacturing processes like injection moulding are time and cost efficient during mass production even though the start-up cost is high. AM is significantly slower than injection moulding in terms of producing parts, but AM can be advantageous in the area of prototyping and small batch manufacturing.

Challenges of Additive Manufacturing (Gao, 2015)

Personal Fabrication vs Mass Production: AM is being largely employed in the field of aerospace, bio-medical and high end automotive industries as they concentrate on manufacturing of end-products. Even though the use of AM technique is not cost effective and time saving for batch production as compared to conventional methods of production, in some cases cost and cycle time are not significant as much as material saving, demand of product and geometrical complexity.

Building Scalability vs Layer Resolution: To have a good surface finish, the layer resolution must be high, which means small layer thickness. Decreasing the thickness of layers drastically affects the speed of printing. For this reason, the layer resolution for most commercially available printers is from ~0.1mm to ~2.5mm.

Material Heterogeneity and Structural Reliability: The use of different types of materials is a limitation of AM techniques. A large variety of AM printers allow the usage of only one material at a time. Multilayer AM systems enables us to use polymers and metals in the same part. However, their use is still not common due to uncertain behaviour at material interfaces.

AM Standardization and Intellectual Property: In order to have good part quality and characteristics, most of the AM printers have standardised set of process parameters and limited material selection options. From an intellectual property standpoint, the emergence of 3D printing market place and open source downloading has challenged the current legal issues and social safety.

10. Additive Manufacturing Technologies

AM translates virtual solid model data into a physical object. The modelled data is usually available in Standard Tessellation Language (STL) format which is broken down into a series of 2D cross-sections. These cross-sections are then fed to the AM machine that adds up material layer by layer to fabricate the physical part. From an historical perspective, AM has its roots since almost 35 years. The first successful AM process was proposed by Ciraud in 1972 (David, 2009) which was a powder deposition method using an energy beam.

Intensive research in the field of materials, processes, software, equipment and integration has brought the use of AM directly and indirectly to make prototype parts for evaluating and testing, as well as for making tools and moulds. The AM processes can be classified in the following seven categories:-

10.1 **VAT Photopolymerisation** (Stereolithography)

VAT Photopolymerisation (VP) processes makes use of liquid and light curable resins as their primary materials. When radiation is focused on the resin, it undergoes photopolymerisation, thus solidifying. A pictorial description of this process can be seen in Figure 62. A movable table (A) is placed initially at a position just below the VAT (B) filled with liquid photopolymer resin (C). The laser beam is moved over the surface of the liquid photopolymer to trace the geometry of cross-section of the object. This causes the resin to harden in the areas where the laser strikes. The laser is moved in the X and Y direction by the scanner system (D). After a layer is completely traced and hardened by the laser beam, the table is lowered into the vat to a distance equal to the distance of one layer.

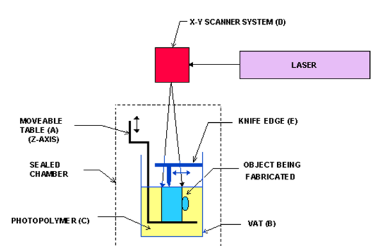


Figure 62 - VAT Photopolymerisation (Stereolithography)

Advantages of VP include part accuracy, surface finish and process flexibility. The main drawback is their usage of photopolymers which generally do not have the impact strength of durability of good quality injection moulded thermoplastics. Stereolithography (SLA) is the process of this category of AM.

10.2 **Material Jetting** (Material Jetting)

The first generation of material jetting (MJ) machines were commercialized in the 1980s. These machines relied on heated wax thermoplastics deposited by inkjet print heads. However, the recent focus has been on deposition of acrylate photopolymers. Multiple jet printer heads deposits droplets of material simultaneously to create a layer, and the UV light is then used to cure the layers (Figure 63).

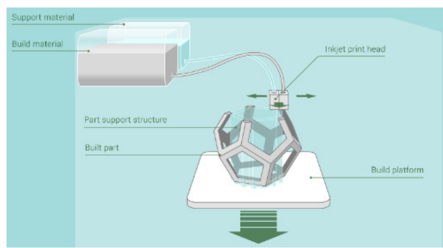


Figure 63 - Material Jetting (Material Jetting)

The process benefits from high accuracy of deposition of droplets and therefore low waste. The major disadvantage is that the material to be used is limited to wax and polymer.

10.3 **Binder Jetting** (DREAMS)

Binder Jetting is an AM technology that creates parts through the deposition of binder into a powder bed of raw material (Figure 64). Once a layer has been printed, the powder feed piston raises, the build platform lowers, and the counter rotating roller spreads a new layer of powder on top of the previous layer. The subsequent layer is then printed and is stitched to the previous layer by jet binder.

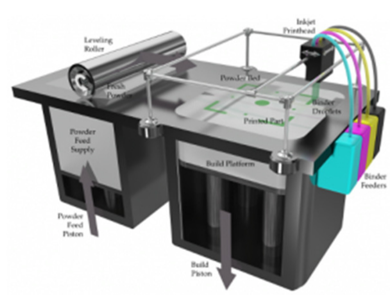


Figure 64 - Binder Jetting Machine (DREAMS)

Speed of printing is a major advantage for this printer. Other advantages are large printing volume, material election flexibility and recyclability of leftover powder. This technique is not suitable for structural parts due to use of binder.

10.4 **Material Extrusion** (About Additive Manufacturing)

Fused Deposition Modelling (FDM) is a common material extrusion process and is trademarked by the company Stratasys. In Figure 65, the material is drawn through the nozzle, where it is heated and is then deposited layer by layer. The nozzle can move horizontally and the platform moves up and down vertically after each new layer is deposited. The material layers can be bounded by temperature control or by external chemical agents.

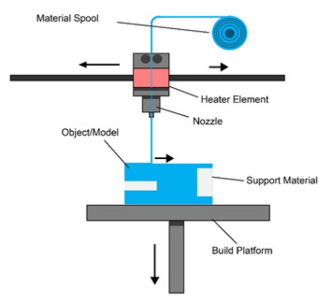


Figure 65 - Material Extrusion (About Additive Manufacturing)

The process is an inexpensive process and ABS plastic can be used which has good structural properties and is easily accessible. The plastic is melted and extruded from a nozzle which has a limited radius. This hampers the final quality. The speed of the process is low as compared to other processes.

10.5 **Powder Bed Fusion** (LUXEXCEL)

The Powder Bed Fusion (PBF) process includes the following processes: Direct metal laser sintering (DMLS), Electron beam melting (EBM), Selective heat sintering (SHS), Selective laser melting (SLM) and Selective laser sintering (SLS).

PBF methods use either a laser or electron beam to melt and fuse material powder together. EBM methods require a vacuum but can be used with metals and alloys in the creation of functional parts.

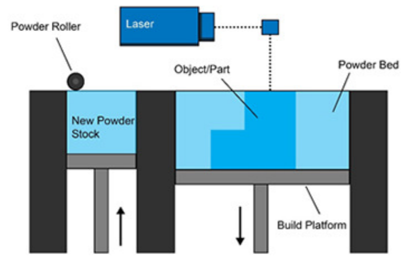


Figure 66 - Powder Bed Fusion (LUXEXCEL)

A hopper or a reservoir below or aside the bed provides fresh material supply. DMLS is the same as SLS, but with the use of metal instead of plastic. The process sinters powder layer by layer. As the layers are added with a roller in between the fusion of layers, the platform lowers the model accordingly.

This process is suitable for visual models and prototypes and a large range of material options benefits this process. But the size of the parts is limited and finish depends upon the powder grain size.

10.6 **Sheet Lamination** (About Additive Manufacturing)

Sheet lamination processes includes ultrasonic additive manufacturing (UAM) and laminated object manufacturing (LOM). UAM uses sheets or ribbons of metal, which are bound together using ultrasonic welding. The process does require additional CNC machining and removal of unbound metal. LOM uses a similar layer by layer approach but uses paper as material and adhesive instead of welding. UAM uses aluminium, copper, stainless steel and titanium. The process takes place at low temperatures and allows for internal geometries to be created.

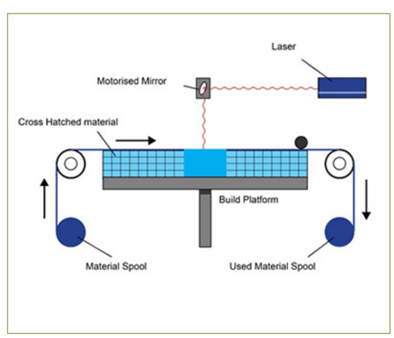


Figure 67 - Sheet Lamination (About Additive Manufacturing)

Advantages of this process include speed, low cost and ease of material handling. Limited use of material adds to the disadvantages of this process.

10.7 Directed Energy Deposition

Directed Energy Deposition (DED) covers a range of terminologies: Laser engineered net shaping (LENS), Direct metal deposition (DMD), Directed light fabrication (DLF) and 3D laser cladding (3DLC). It is a more complex printing process commonly used to repair or to add additional material to existing components.

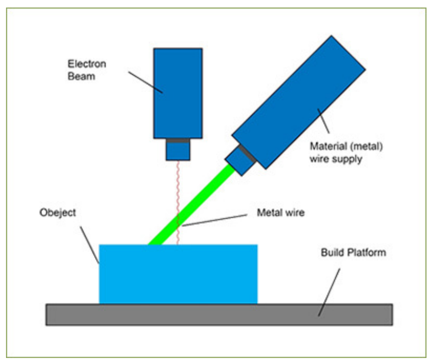


Figure 68 - Direct Energy Deposition

A typical DED machine consists of a nozzle mounted on a multi axis arm, which deposits melted material onto a specified surface, where it solidifies. The process is similar in principle to material extrusion, but the nozzle can move in multiple directions and is not fixed to a specific axis. The process can be used with polymers and ceramics but is typically used with metals, in the form of either powder or wire. This process can control the grain structure to a high degree which leads to good quality finish.

Summary of AM technologies:-

As we have seen above, all these processes differ from each other in terms of the techniques used to deposit layers and the ways in which the deposited layers are bound together. Table 1 summarizes the above information and also depicts the materials that can be used for each process.

Table 1: AM processes and materials

Type	**Process/Technology**	**Material**
VAT photopolymerisation	SLA (Stereolithography)	UV curable resins
		Waxes
		Ceramics
Material Jetting	MJM (Multi-Jet Modelling)	UV curable resins
		Waxes
Binder Jetting	3DP (3D printing)	Composites
		Polymers, Ceramics
		Metals
Material Extrusion	FDM (Fused Deposition Modelling)	Thermoplastics
		Waxes
Powder Bed Fusion	SLS (Selective Laser Sintering)	Thermoplastics
		Metals
	SLM (Selective Laser Melting)	Metals
	DMLS (Direct Metal Laser Sintering)	Metals
	EBM (Electron Beam Melting)	Metals
Sheet Lamination	LOM (Laminated Object Modelling)	Paper
		Metals
		Thermoplastics
Directed Energy Deposition	DMD/DLF/LENS	Metals

11. Impacts of AM Industry

Commercial View: The improvements in 3D printing is focused on time and cost efficiency of the industrial manufacturing. This area is being explored by the industries in the field of aerospace, defence, power generation and medical facilities. For example, Optomec has developed the Laser Engineered Net Shaping (LENS) systems; 3D Systems has developed the ProJet series of Printers; e-Manufacturing Solutions (EOS) has developed the EOSINT series for laser-sintering systems; Z-Corporation (acquired by 3D Systems in 2012) has developed the ZPrinter series; ExOne has developed the M-Print systems. Now a days the AM systems are finding their use in small companies, supportive communities and individual designers.

Intellectual Property Considerations: Ross Housholder (1981) holds the first patent in the area of plastics of AM. The invention of Stereolithography by Charles Hull was done in 1984 and revolutionized the field of 3D printing. In the 1980s, selective laser sintering, sheet lamination, material extrusion and 3D printing were patented with the support of the National Science Foundation. 3D Systems and Stratasys own the largest number of patents in the area of AM. The cost of 3D printers is bound to get less for the consumers as a large number of patents in this field are expiring and many companies are entering this business.

Educational View: As AM is being adopted by many industries, it is important for the workforce to be educated on how to employ these technologies. As outlined in "2009 roadmaps for additive manufacturing" (Huang, 2014), unfamiliarity with AM technologies is seen as barrier to adoption. The roadmaps suggests the development of university courses and programs for educating the general population to enhance the interest in AM applications and generate some social awareness for the technologies. On this basis, a National Science Foundation workshop on AM education was held, wherein the discussions about the educational needs and opportunities for AM engineering workforce were conducted. Hence it was decided that the workforce need to have knowledge about (1) AM processes and process/material relationships, (2) engineering fundamentals with an emphasis on material science and manufacturing, (3) professional skills for problem solving and critical thinking, (4) design practices and tools, and (5) cross-functional teaming and ideation techniques to nurture creativity. AM courses at the undergraduate and graduate levels are emerging but their limited quantity does not match the recent interests and importance of the technology.

12. AM and Injection Moulding

In contrast to the injection moulding processes which require costly moulds, AM showcases relatively low fixed costs. Since AM does not require expensive tooling, it is particularly cost effective for small production runs.

In comparison to subtractive technologies, there is less waste material with AM. Building objects layer by layer, instead of traditional machining methods, can reduce material needs and costs by up to 90% (The Printed World: Three-Dimensional Printing from digital designs, 2011). In addition, 95-98% of waste material can be recycled in AM.

AM technologies are also able to produce initial parts much quicker than injection moulding as no setup time is required. Furthermore, considerable time is conserved when producing revised designs.

Advantages of AM over Injection Moulding:-

- No need for costly tools.
- No geometrical constraints. Virtually any complex shape can be produced without need for design constraints.
- No setup time needed. Direct production of parts can be initiated.
- Speed and ease of designing and modifying products.
- No scrap, milling or sanding required.

Limitations of AM:-

- Higher costs for large production runs relative to injection moulding and other technologies.
- Reduced choice of materials and surface finish.
- Lower precision as compared to other technologies.
- Limited strength, resistance to heat and moisture.

These limitations of additive manufacturing are the main reasons due to which injection moulding cannot be replaced by any AM technologies at this point of time. Today, injection moulding can provide parts with unit costs and production rates with which AM cannot compete. Even in part designs where the total cycle time (including setup time) is lower in AM, the unit costs of the process are still not competitive.

A lot of research is being made to combine additive and subtractive machining processes so as to optimize material usage and also the cost of the process. The system using both additive and subtractive manufacturing processes is called as hybrid system of manufacturing.

13. Rapid Tooling with Hybrid System of Prototyping

Traditional injection moulding is less expensive for manufacturing plastic products in high quantities; in contrast, RP processes are faster and less expensive when producing relatively small quantities of parts. However, there exists a small area where neither the use of injection moulding or traditional rapid prototyping process can be justified.

Rapid tooling (RT) techniques, an extension of rapid prototyping, allows the manufacture of production tools rather than the actual part.

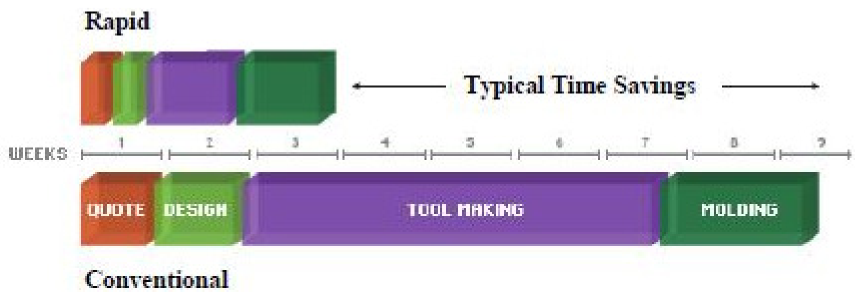


Figure 69 - Rapid vs conventional injection moulding process (Kumar, 2010)

There are several existing choices for rapid tooling available from purely additive, purely subtractive and hybrid systems.

Case 1: A hybrid system of friction stir welding (FSW) and CNC machining was developed for rapid tooling. The STL model of the part was developed. From the STL file, the boundary wall dimensions, tool path of FSW and CNC machine was derived. The number, location and sequence of spot welds were calculated from boundary wall dimension and the tool path was generated.

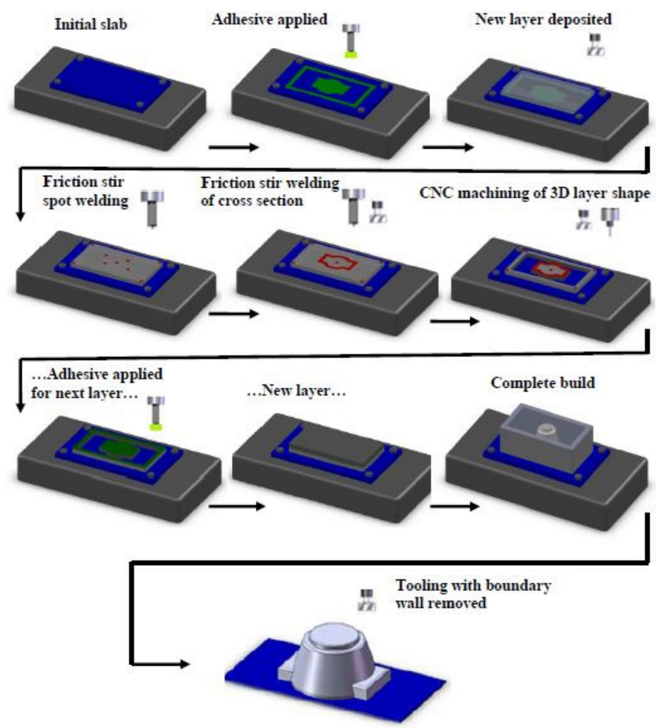


Figure 70 - Detailed process steps of rapid tooling system (Kumar, 2010)

An adhesive is applied on the boundary where the cross-section of the part lies. The adhesively bonded plate is then spot welded using friction stir spot welding. This is because the strength of adhesives alone is assumed to be insufficient to withstand the forces of friction stir welding process. A new layer is deposited by use of friction stir welding over the cross-section and then the CNC machining of 3D layer shape is done. This process is repeated till the built is complete. Later the boundary walls are removed if needed.

Case 2: Hybrid manufacturing can also be used for making conformal cooling systems in inserts and other mould parts. The conformal cooling system is a term for cooling channels which conforms to the contours of the inserts or cavity of the injection moulding tool. The difference between conventional and conformal cooling systems is presented in Figure 71:

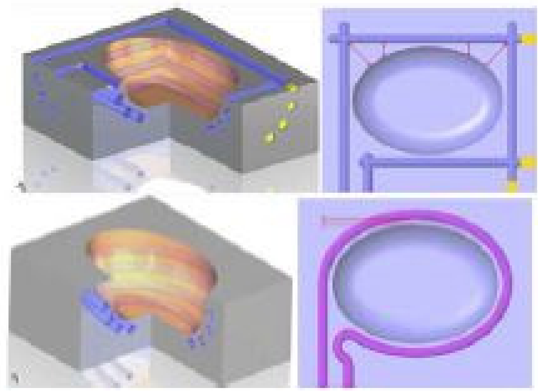


Figure 71 - (Above) Conventional cooling system. (Below) *Conformal cooling system (Homar, 2012)*

With conformal cooling systems, it is possible to get better dimensional accuracy of the moulded part, better mechanical properties of the part and a reduced cycle time of injection moulding up to 40%. Conformal cooling systems are not possible with conventional machining systems due to design constraints but, with hybrid manufacturing, conformal cooling channels are a possibility. In Figure 72, we can see the example of a tool insert with conformal cooling system.

Figure 72 - Tool insert with conformal cooling channels

The hybrid production enables the production of one work piece with two different processes. This way, it is possible to manufacture products in any shape with minimal production expenses.

The above explained process is one example of a hybrid system of rapid tooling/prototyping. As we know that the process of subtractive prototyping causes a lot of material wastage and additive prototyping is very expensive. This kind of hybrid processes are cost effective and also time saving.

14. Wohlers Report

In 2013, the Wohlers Report, from the 3D printing industry analyst company Wohlers Associates, was published. It is not shocking that the growth of 3D printing is expected to increase. However the more surprising fact is that the report suggests that “growth of low cost personal 3D printers” will be more significant. Here are some of the facts and figures and excerpts (Park, 2013) from the press release that Wohlers Associates had issued:-

“Wohlers Report 2013 provides an in-depth look at market forces, competitive products and services, and industry growth. The market for 3D printing in 2012, consisting of all products and services worldwide, grew 28.6% to $2.204 billion. This is up from $1.714 billion in 2011, when it grew 29.4%. Growth was 24.1% in 2010. The average annual growth of the industry over the past 25 years is an impressive 25.4%. The CAGR (Compound Annual Growth Rate) is 27.4% over the past three years (2010–2012)”

Growth of the low-cost (under $5,000) “personal” 3D printer market segment averaged 346% each year from 2008 through 2011. In 2012, the increase cooled significantly to an estimated 46.3%, according to research by Wohlers Associates. Most of these machines are being sold to hobbyists, do-it-yourselves, engineering students and educational institutions.

The 3D printing industry is expected to continue strong double-digit growth over the next several years. By 2017, Wohlers Associates believes that the sale of 3D-printing products and services will approach $6 billion worldwide. By 2021, Wohlers Associates forecasts the industry to reach $10.8 billion. It took the 3D printing industry 20 years to reach $1 billion in size. In five additional years, the industry generated its second $1 billion. It is expected to double again, to $4 billion, in 2015.”

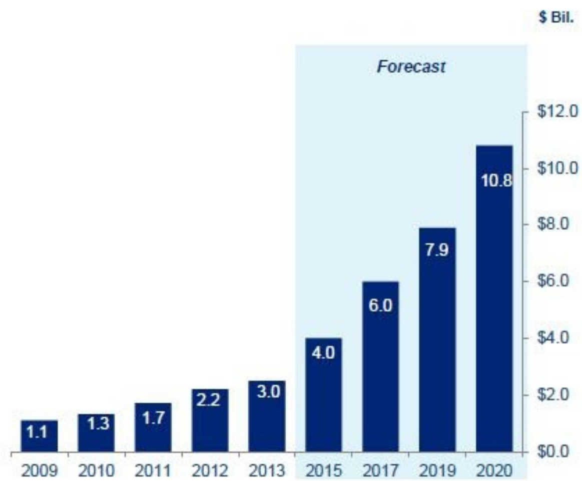


Figure 73 - Wohlers Data on Market Size and forecast of 3D printing market (Cotteleey, 2014)

As shown in Figure 73, the global additive manufacturing market, reached sales of $3 billion in 2013, an annualized growth of 35% over sales of $2.2 billion in 2012. AM industry growth over the last 25 years has been 25% and 29% in the last three years.

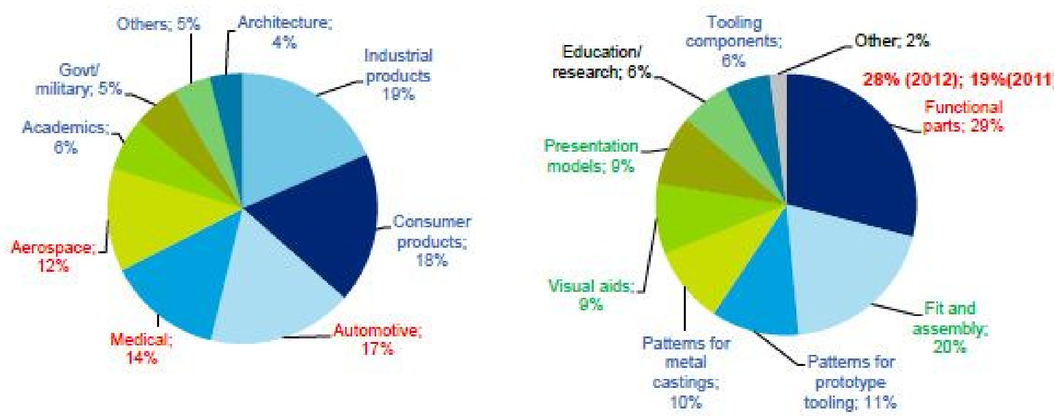


Figure 74 - (left) AM system sales revenue to various sectors; (right) AM systems deployments by application (Cotteleey, 2014)

As shown in Figure 74, AM systems are sold into a wide range of sectors. The following points are noteworthy when looking into the market approach towards AM.

- Multiple industry verticals contributed to double digit sales of AM systems in 2013.
- Automotive, Medical and Aerospace lead (43%) among targeted sectors.
- Functional part production is growing faster than the rest of market.

- Companies are typically spending 10 times more on production than prototyping – an imperative for AM users and providers is to look beyond prototyping and focus on end-parts production.

According to the Wohlers Report 2015 (Wohler, Terry, 2015) there were a total of 79,602 industrial installation of 3D printers between 1988 and 2014. During that time period, the US had 38.1% of world market for additive manufacturing for machines costing $5000 and above. Japan had 9.3% of these units, China had 9.2% and Germany had 8.7%, while other countries lagged behind.

Between 2007 and 2014, the annual desktop printer sales saw a growth from just 66 units to 139,584 units. The Wohlers Report of 2015 believes that the largest growth for desktop 3D printer sales will be for companies and educational institutes and not for home use. As for future growth, the Wohlers Report 2015 suggests that total global sales in 2015 will increase to $7.3 billion, while in 2018 it will be $12.7 billion and in 2020 the market is expected to reach $21.2 billion.

It is very interesting to note that in the Wohlers Report of 2013, it was forecasted that the total market sale of AM systems in 2020 would be around $10.2 billion. But now in their 2015 report, the predicted forecast of market sale more than doubles those figures. This shows that the demand of AM systems has soared to a different level and in the near future AM has the ability to superimpose the conventional part making processes.

15. Future Trends of AM

AM will find its market based on the custom products that are made when ordered, such as dental and medical devices, and low turnover replacement parts. These goods are typically ordered in unique configuration and in very small quantities. A significant advantage of AM is the ability to quickly and cost effectively supply low-demanding products without the risk of having an inventory of finished goods. With AM, the risk of poor design is based on wasted design time rather than production investment.

A number of issues relating to cost, accuracy and strength of AM products needs to be solved before this technology achieves widespread adoption. For instance, in terms of cost, AM process cost per part are 10 to 100 times more expensive than the corresponding injection moulding process for large batch production. Material choice, colours and surface finish is more limited than in typical mass-production processes. There are also some issues relating to interlayer bonding that can lead to delamination and breakage under stress, the material's strength, viscosity, dimensional stability, resistance to heat and moisture and colour stability need careful evaluation. As the prices of materials fall and material quality gets better, the use of AM technology will expand beyond its current scope.

A secondary promise of AM lies in the separation of product design from product manufacturing. As AM evolves, consumers will be able to purchase designs online and then build products at home. Appliance companies can also contact out the manufacturing of spare parts to third parties, which will then build parts based on CAD software provided by the appliance manufacturer. Since no inventory will be kept, the cost of parts will be less.

In the long term, the range of industrial 3D printing applications will increase drastically as new AM machines will be available to accommodate larger products and achieve great levels of precision. There should be significant decline in materials and machinery cost as more individuals and firms adopt this technology. Additionally, the usage of AM will expand due to the availability of materials with greater strength and resistance to heat and moisture.

When the price of AM machines is reduced, perhaps to 500 euros or so, the number of machines used for home applications will drastically expand from current levels. At this price levels, the market penetration of home-based 3D printers will begin to resemble that of small laser printers. The future holds great promise for AM as a technology and for end users as a result.

16. Conclusions

The moulding industry is a place where accuracy and quality are one of the most important factors for future development. During the internship in Moldegama S.A., I was able to go through various sections and improve greatly my understanding of the process of mould making. It was also important to gain more knowledge on some processes of quality control and problem detection about which I had never heard of before. In terms of problem solving and quality control, Moldegama S.A. is a relatively well-organised company. But, in my opinion, the process can be planned in a much better way so as to reduce the areas of repetitive errors in designing as well as machining. To conclude, I would like to say that this internship has been a great tool to understand the problem solving methods being employed in the industry and to figure out ways to face the competitive and the demanding market of the world.

When it comes to additive manufacturing, I believe we are poised and on the brink of "a third industrial revolution" where emerging companies are rethinking about how traditional manufacturing will be transformed. Some of the driving factors that are already bringing about this change, or are likely to do so in the next few decades, are discussed above along with some of the challenges they bring.

Additive manufacturing processes are making way into the world at rather a greater pace as thought of which can be estimated by the review of the Wohlers Report. This is because most of the patents in this field are expiring soon, paving the way for researchers to look into the technology and improve it. Even though we have a lot of printers in the market, their usage is not in abundance, due to high cost and low awareness among people. Currently, the direct fabrication of functional end-use products is becoming the main trend of AM technology, and it is being increasingly implemented to manufacture parts in small or medium quantities. The segment of market to use products from AM will be the high end automotive industries and medical industries. This is because of the fact that high end automotive industries such as Lamborghini, Ferrari, etc. do not make large quantities of the same products as they believe in uniqueness. In the same way, medical industries also do not produce products like tooth, prosthetics, etc., in large numbers. This is because each person has unique body parameters and generalisation is not something that reputed medical industries can afford. In the near future, AM will make products move closer to end users rather than the producers.

Also the development of hybrid processes of rapid prototyping and tooling is making an impact on the way injection moulds are being made. The replacement of moulds with 3D printers is not a solution for the coming few years, in terms of mass production, but a hybrid process of traditional mould making and manufacturing inserts for moulds with AM processes is definitely a good deal.

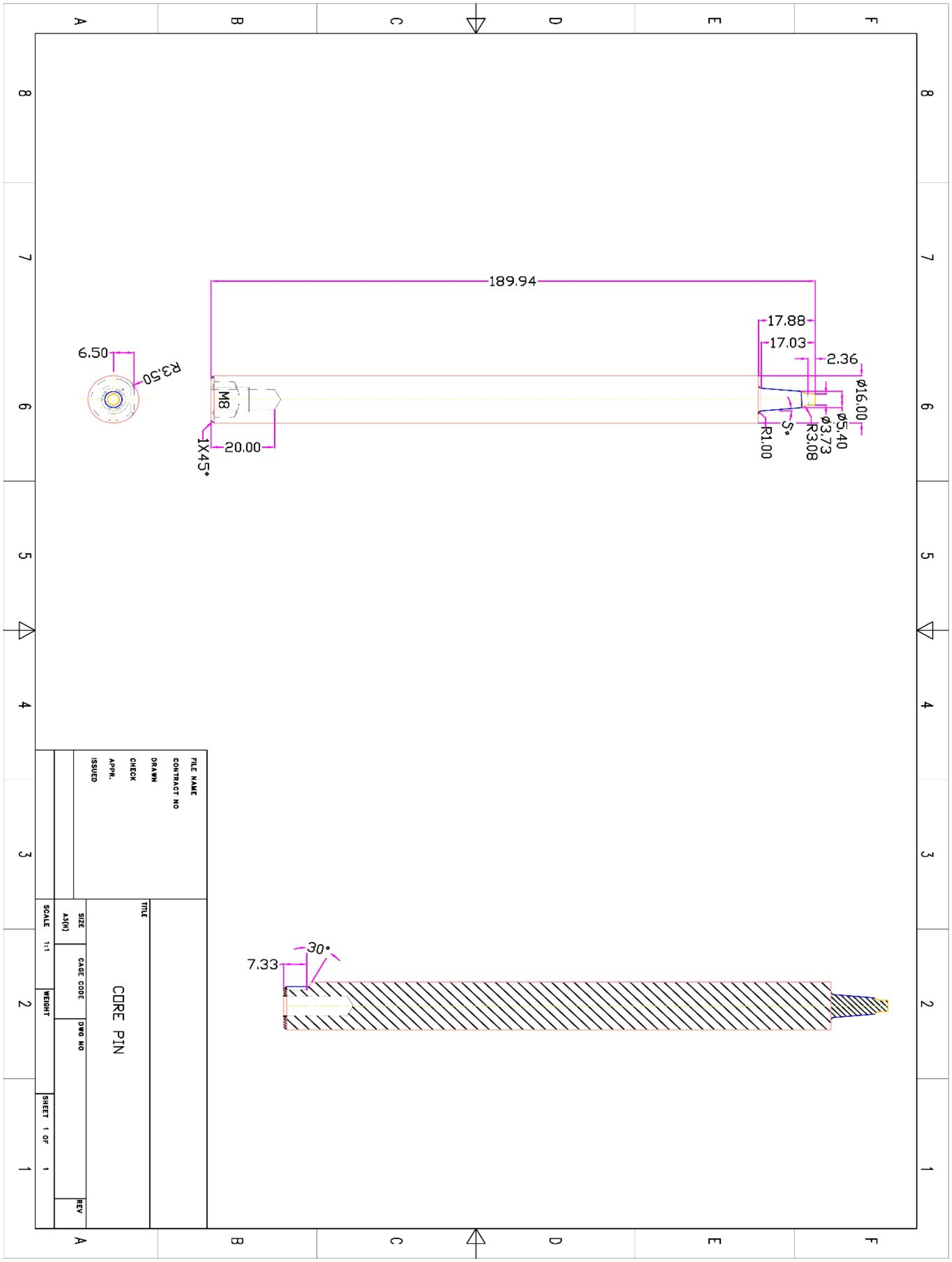
17. Bibliography

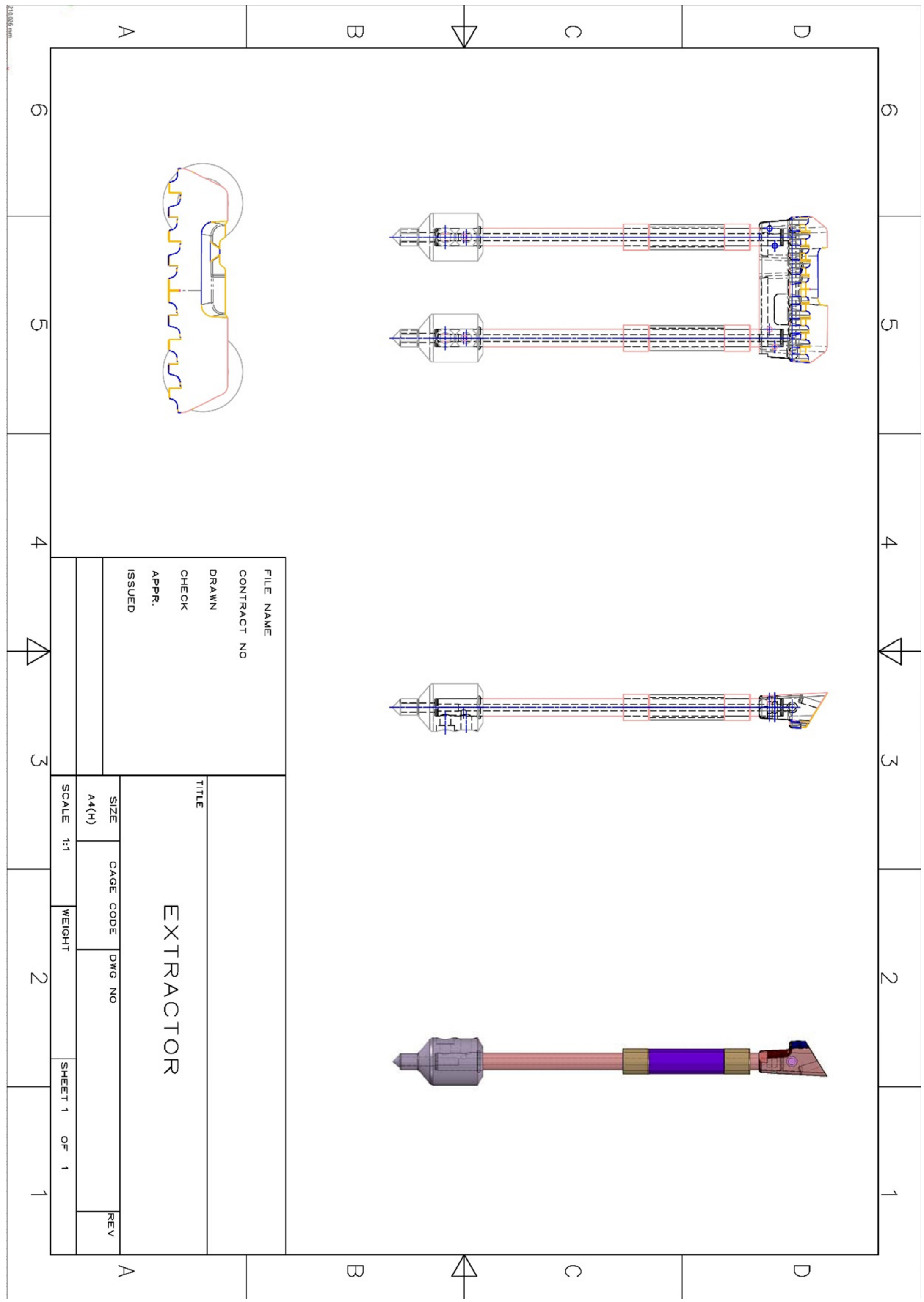
- Retrieved from http://www.xcentricmold.com/resource/images/injection-mold-tooling.jpg
- *About Additive Manufacturing*. Retrieved from http://www.lboro.ac.uk/research/amrg/about/the7categoriesofadditivemanufacturing/material extrusion/
- Aleck, G. (2015, June 25). Growing West Michigan CNC Machining Business for Sale!
- Bozzelli, J. (2010). *Injection Molding: Why and How do Gate-Seal Experiments.* Retrieved from http://www.ptonline.com/columns/injection-molding-why-how-to-do-gate-seal-experiments
- Bryce, D. M. (1996). *Plastic Injection Molding.* Michigan.
- Capetronics. (2012, September 06). *Basic Injection Molding Machine*. Retrieved from http://www.capetronics.com/info/injmachine.htm
- *CN Moulding*. Retrieved from http://www.chinamoldmaker.org/Cooling-design.html
- Cotteleey, D. M. (2014). 3D opportunity: Additive manufacturing paths to performance, innovation and growth.
- David, L. (2009). *A Brief History of Additive Manufacturing.* Retrieved from https://iweb.tntech.edu/rrpl/rapidtech2009/bourell.pdf
- *Design of Gates.* Retrieved from http://files.hanser.de/hanser/docs/20040401_244515438-3573_3-446-21256-6.pdf
- *DREAMS*. Retrieved from http://www.me.vt.edu/dreams/binder-jetting/
- DS, C. (2008). *Deep Hole Drilling.* Retrieved from Cast Products and Mold Designer Skills at the European Context: http://webhotel2.tut.fi/projects/caeds/tekstit/frame_materials.html

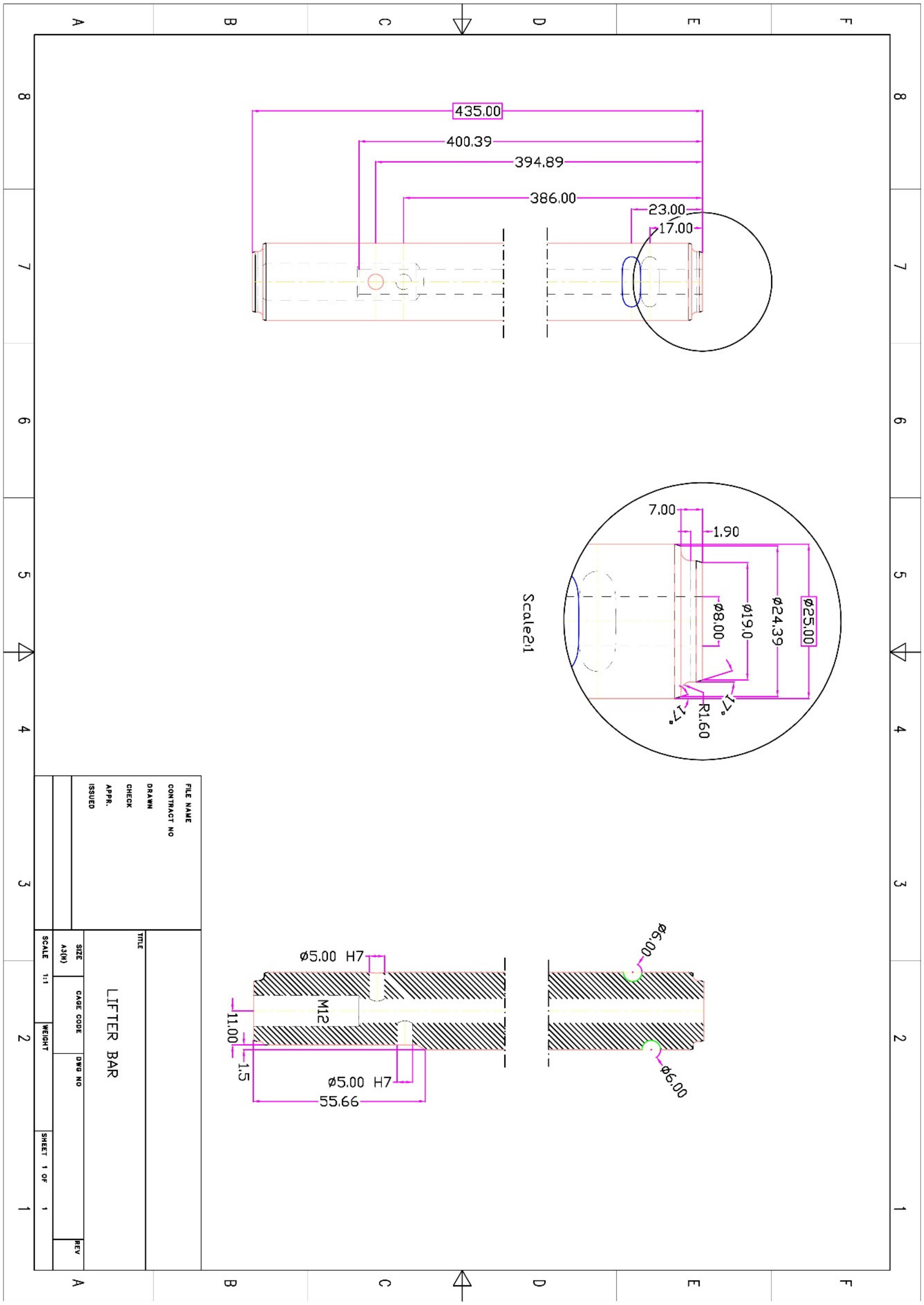
- DS, C. (2008). *Standard Components.* Retrieved from Cast Products and Mold Designer Skills: http://webhotel2.tut.fi/projects/caeds/tekstit/frame_materials.html
- Gao, W. (2015). *The status, challanges and future of additive manufacturing in engineering.* Retrieved from http://dx.dio.org/10.1016/j.cad.2015.04.001
- *Gestion De Compras*. Retrieved from http://www.gestiondecompras.com/en/products
- Herbert, R. In *Understanding Product Design for Injection Molding* (p. 14). Munich: Hanser Publication.
- Herbert, R. Understanding Product Design for Injection Molding. Munich: Hanser Publications.
- Herbert, R. Understanding Product Design for Injection Molding. Munich: Hanser Publications.
- Hill, M. *Plastic Injection Mold Designs.* Retrieved from http://faculty.ferris.edu/hillm/myweb7/index.htm
- Homar. (2012). *Additive Manufacturing and high speed cutting included in hybrid manufacturing.*
- *Hotech Polerteknik*. Retrieved from http://www.polerteknik.dk/index-english.html
- Huang, Y. (2014). NSF Additive Manufacturing Workshop. *Frontiers of Additive Manufacturing Research and Education*.
- *Injection Mold Tech Basics.* (2006). Retrieved from http://www.mapeng.net/news/mechanical_English_article/2014/4/mapeng_1446836342846.html
- *Injection Molding Gate Types*. Retrieved from ACO MOLD: http://www.acomold.com/injection-molding-gate-types.html
- Kumar, R. (2010). Process planning for rapid manufacturing of plastic injection mold for short run production. Ames, Iowa: Iowa State University.
- LUXEXCEL. Powderbed Fusion. Retrieved from http://blog.luxexcel.com/3d-printing/powderbed-fusion/

- Maffei, K. (2013). Press Release. Munich, Germany. Retrieved from http://www.kraussmaffei.com/en/press-releases/d/K2013_IMM_energy.html
- *Material Jetting*. Retrieved from https://www.additively.com/en/learn-about/material-jetting
- Park, R. (2013). Unsurprisingly, Wohler Report 2013 reveals continued growth in 3D printing. Retrieved from http://3dprintingindustry.com/2013/05/24/unsurprisingly-wohlers-report-2013-reveals-continued-growth-in-3d-printing/
- Prasad, K. (1999, September). *Plastic Mold Design using C-Mold.* Retrieved from http://www.cse.iitk.ac.in/users/amit/courses/751/99/kprasad/
- *Rebling Plastics.* Retrieved from Quality: http://www.reblingplastics.com/quality.htm
- S.A., M. (2015). Retrieved from http://www.moldegama.com/
- Sepe, M. (2011). *The Importance of Melt and Mold Temperature.* Retrieved from http://www.ptonline.com/columns/the-importance-of-melt-mold-temperature
- *Stereolithography*. Retrieved from http://www.additive3d.com/sl.htm
- Tayu. (2011). *Factors affecting the Injection Process.* Retrieved from http://injection-molding-machine.tayu.cn/injection-machine/209.html
- *The Printed World: Three-Dimensional Printing from digital designs.* (2011, Februaru 10). Retrieved from The Economist: www.economist.com/ node/18114221
- Wang, J. (2012). PVT Properties of Polymers for Injection Molding.
- Wohler, Terry. (2015). The Next Frontier in 3D Printing. Colorado, USA.
- XCENTRIC. Retrieved from http://www.xcentricmold.com/

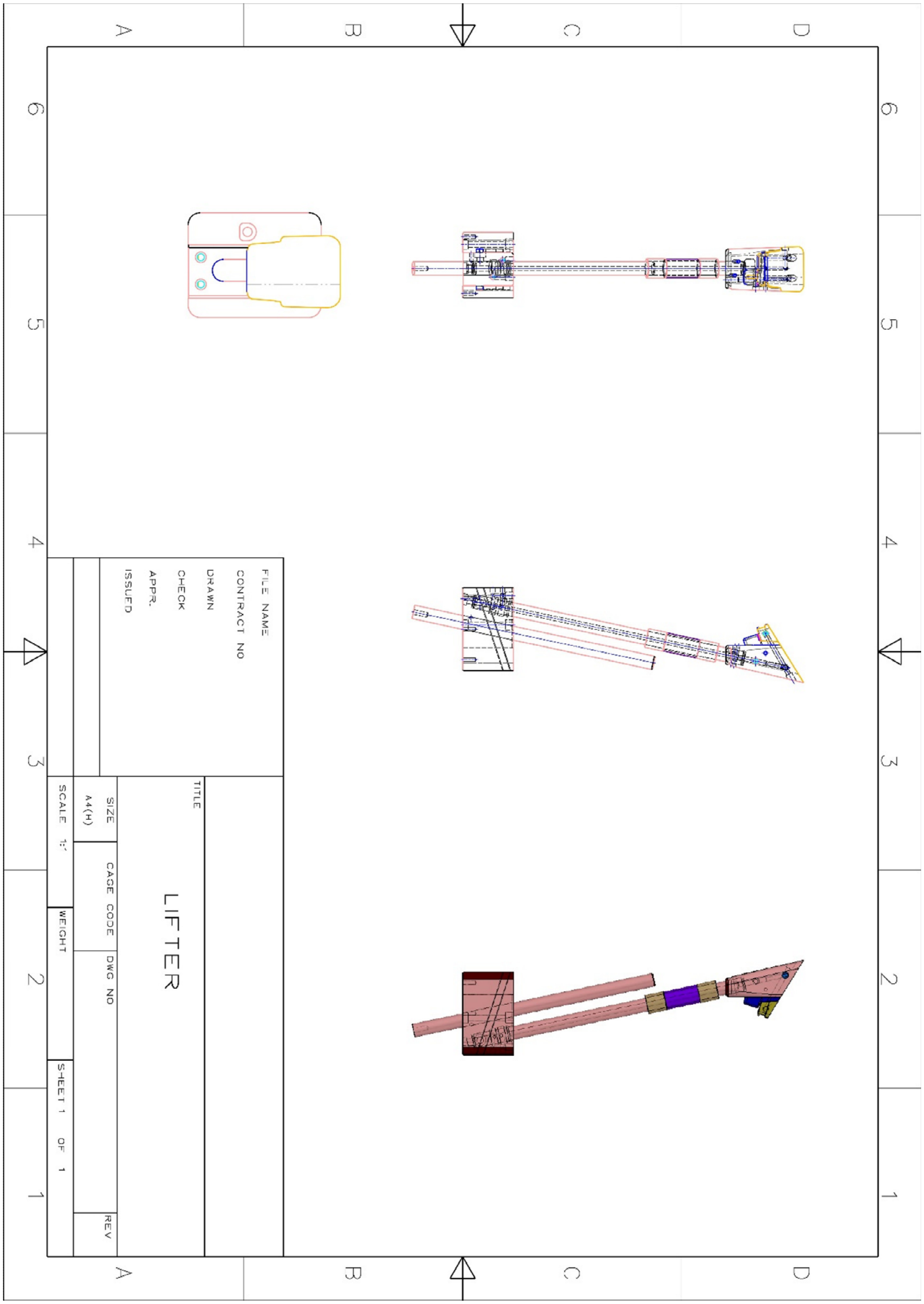
ANNEXES

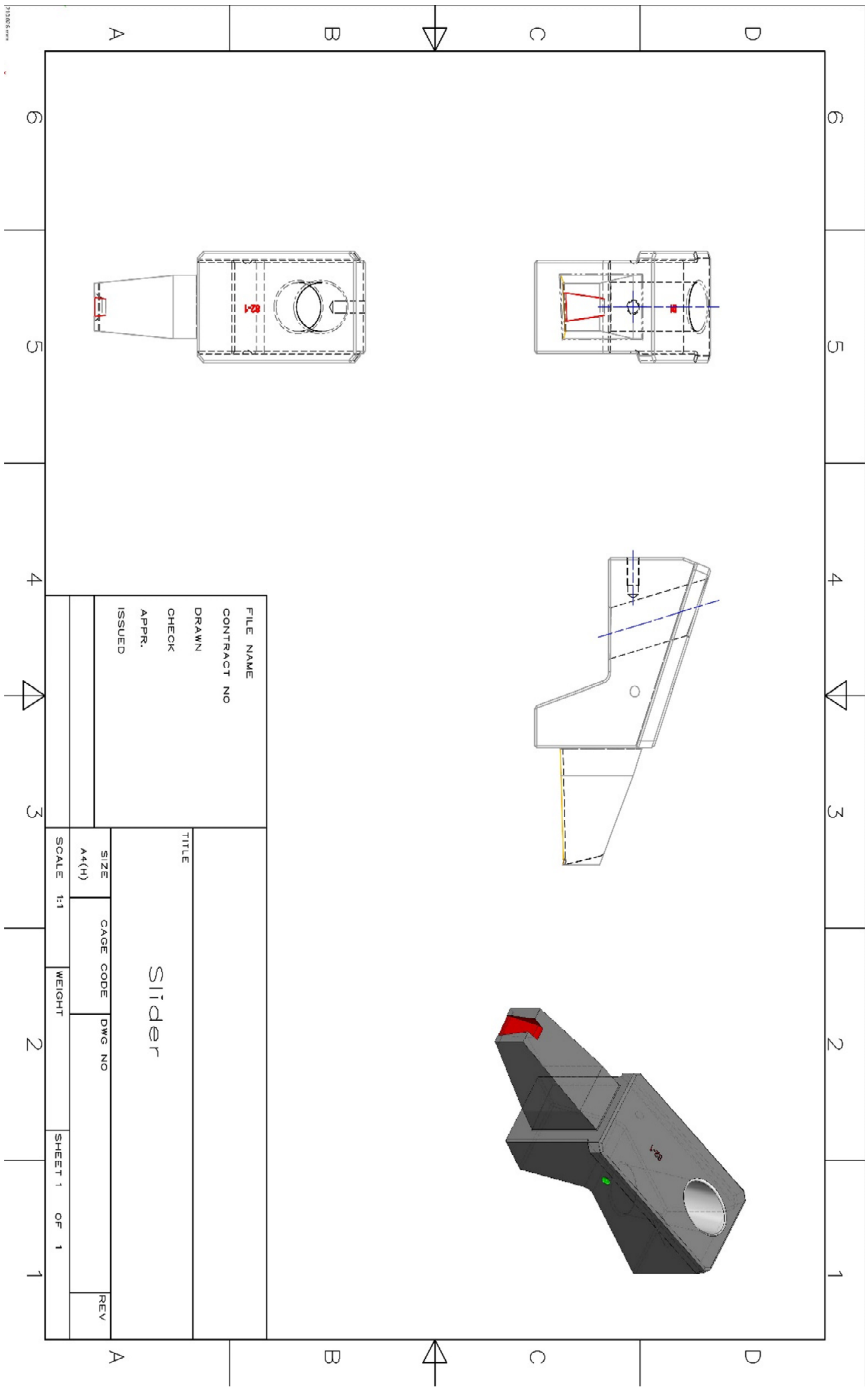

CORE PIN
189.94
17.88
17.03
2.36
ø16.00
ø5.40
ø3.73
R3.08
5°
R1.00
M8
20.00
1X45°
6.50
R3.50
7.33
30°
FILE NAME
CONTRACT NO
DRAWN
CHECK
APPR.
ISSUED
TITLE
SIZE
A3(H)
CAGE CODE
DWG NO
REV
SCALE 1:1
WEIGHT
SHEET 1 OF 1


FILE NAME
CONTRACT NO
DRAWN
CHECK
APPR.
ISSUED
TITLE
EXTRACTOR
SIZE
A4(H)
CAGE CODE
DWG NO
REV
SCALE 1:1
WEIGHT
SHEET 1 OF 1



LIFTER BAR
TITLE
FILE NAME
CONTRACT NO
DRAWN
CHECK
APPR.
ISSUED
SIZE
A3(H)
CAGE CODE
DWG NO
REV
SCALE 1:1
WEIGHT
SHEET 1 OF 1
435.00
400.39
394.89
386.00
23.00
17.00
7.00
1.90
Ø25.00
Ø24.39
Ø19.0
Ø8.00
R1.60
17°
17°
Scale2:1
Ø5.00 H7
M12
11.00
1.5
Ø5.00 H7
55.66
Ø6.00
Ø6.00


LIFTER
TITLE
FILE NAME
CONTRACT NO
DRAWN
CHECK
APPR.
ISSUED
SIZE
A4(H)
SCALE 1:'
CAGE CODE
DWG NO
WEIGHT
SHEET 1 OF 1
REV


FILE NAME
CONTRACT NO
DRAWN
CHECK
APPR.
ISSUED
TITLE
Slider
SIZE
A4(H)
CAGE CODE
DWG NO
REV
SCALE 1:1
WEIGHT
SHEET 1 OF 1