Simulation and Design of a plastic injection Mold

A Joint mold for credit card and USB holder

Samson Teklehaimanot

Degree Thesis
PTE
2011

DEGREE THESIS	
Arcada	
DegreeProgramme:	Plastic Technology
Identificationnumber:	9639
Author:	Samson Teklehaimanot
Title:	Simulation and Design of a plastic injection mold
Supervisor (Arcada):	Marko Voho
Commissioned by:	

Abstract:

Injection molding is one of the most important processes in the plastic manufacturing industry. More than one-third of all plastic materials are injection molded, And the mold is one of the main components in the injection molding process. The aim of this engineering thesis is to show detailed steps on how to design a complete mold and using the simulation software to analyze the material flow and defects in the product. The product design for this project is a joint credit card and USB flash drive holder. What makes this product unique from others is that it contains both functions together whereas most of the products in the market today don’t contain the same dual application.

The design of this product and the mold were made by the designing analysis software Solid edge ST2 software, Which is then simulated by the use of Mold Flow.

Keywords:	Mold design, credit-card holder, USB holder, Mold flow simulation.
Numberof pages:	53
Language:	English
Date ofacceptance:	

Acknowledgements

I owe sincere and earnest thankfulness to my advisor *Marko Voho* for the support, guidance and patience he showed me throughout my Thesis writing.

I would also like to show my gratitude to *Mathew Vihtonen*, *Mariann Holmberg* and *Badal Durbo karis* for their support in the study years.

It is a great pleasure to thank all my friends and classmates who helped me write my thesis successfully by providing me great information resources.

I am sincerely and heartily grateful to my mother *Dr Wubayehu Berhane* and Brother *Fitsum* who boosted me morally. I am sure this would have not been possible without their help.

Table of Contents

List of Figures

List of Tables

1. Introduction

1.1.Background

The injection molding has seen steady growth since its beginnings in the late 1800's. The technique has evolved from the production of the simple things like combs and buttons to major consumer, industrial, medical, and aerospace products.

The invention of an injection molding machine was achieved by John Wesley who injected hot celluloid into a mold which resulted in billiard balls which were used as a replacement for ivory which was based on the pressure die casting technique for metals. [7] The industry progressed slowly over the years, producing products such as collar stays, buttons, and hair combs. The industry expanded rapidly in the 1940s because World War II created a huge demand for inexpensive, mass-produced products. In 1946, American inventor James Watson Hendry built the first screw injection molding machine, which allowed much more precise control over the speed of injection and the quality of articles produced. This machine also allowed material to be mixed before injection, so that colored or recycled plastic could be added to virgin material and mixed thoroughly before being injected. [20]

The main concept of plastic molding is placing a polymer in a molten state into the mold cavity So that the polymer can take the required shape with the help of varying temperature and pressure. There are different ways of molding a plastic some of them are blow molding, Injection molding, rotational molding and compression molding. Each technique has their own advantages in the manufacturing of specific item.[1]

1.2.Objective

These are objectives of the thesis

- To prepare a product design for ’’ a joint credit card & USB holder ’’ by using design analysis software
- To design and test the plastic injection mold for the specific product
- Using Mold Flow to simulate the polymer flow and finding out maximum clamp force and Fill time.

1.3 Abbreviation

1) LDPE - Low Density Polyethylene

2) HDPE - High Density Polyethylene

3) PP - Polypropylene

4) PS - Polysterene

5) ABS - Acrylonitrile-Butadiene-Styrene

6) PVC - Polyvinyl Chloride

7)PMMA - Polymethylmethacrylate

8)PVDF - Polyvinylidene Fluoride

9) PET - Polyethylene Terephtalate

10) PI - Polyimide

11) PC - Polycarbonate

12) PES–Polyethersulfone

13) Tg – Glass transition temprature

2. Literature Survey

2.1.The injection molding

Injection molding is a method of forming a plastic product from powdered thermoplastics by feeding the material through the machine component called the hopper to a heated chamber in order to make it soft and force the material into the mold by the use of the screw. In this whole process pressure should be constant till the material is hardened and is ready to be removed from the mold. This is the most common and preferable way of producing a plastic products with any complexity and size. [1]

Injection molding permits mass production netshape manufacturing of high precision, three-dimensional of plastic parts. [3]

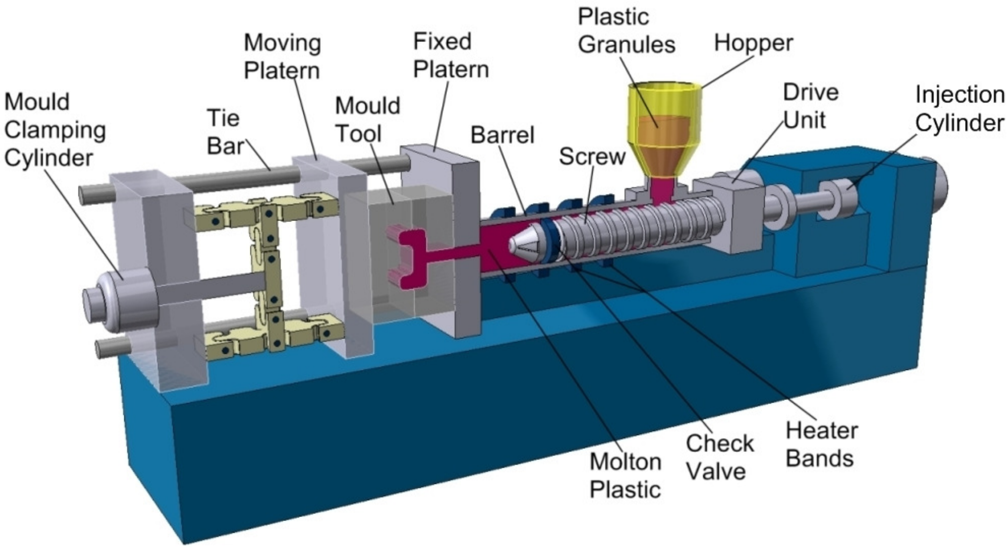


Figure 1 : Injection Molding Machine [4]

2.1.1. The injection molding process

The injection molding process stages starts with the feeding of a polymer through hopper to barrel which is then heated with the sufficient temprature to make it flow , then the molten plastic which was melted will be injected under high pressure into the mold the process is commonly known as Injection, After injection pressure will be applied to both platens of the injection molding machine(moving and fixed platens) inorder to hold the mold tool together afterwards the product is set to cool which helps it in the solidification process. After the product gets its shape the two platens will move away from eachother inorder to separate the mold tool which is known as mold opening and finally the molded product is ejected or removed from the mold. And the process will repeat itself.

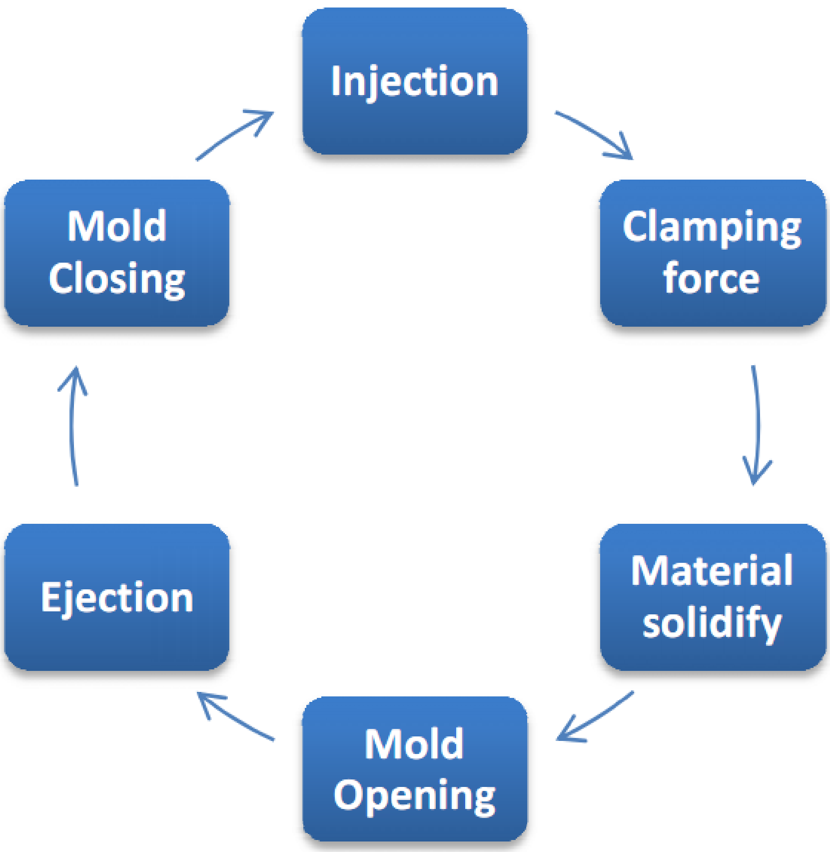


Figure 2 : Injection molding cycle [5]

The molding cycle starts with the retraction of the ejector plate, followed by closing of the mold. The injection unit melts the polymer resin and injects the polymer melt into the mold. The ram fed injection molding machine uses a hydraulically operated plunger to push the plastic through a heated region. The melt converges at a nozzle and is injected into the mold.

The melt is forced into the mold in two or three stages:

- **Stage 1:** Fill stage
  - During this stage, the mold cavities are filled with molten resin. As the material is forced forward, it passes over a spreader, or torpedo, within the barrel, which causes mixing. This stage is determined by an injection velocity (rate), a pressure, and a time. Injection velocity is the rate at which the plunger moves forward.
- **Stage 2:** Pack stage
  - As the melt enters the mold, it cools and introduces shrinkage. The pack stage is necessary to force more melt into the mold to compensate for shrinkage.
- **Stage 3:** Hold stage
  - When no more material can be forced into the mold, melt can still leak back through the gate. The hold stage applies forces against the material in the cavity until the gate freezes to prevent leaking of the melt. In some machines, pack and hold are combined into a single second or holding stage.

Each stage is governed by a particular pressure and time duration, as can be seen in Figure 4: Once the mold is filled and packed and the gate has cooled, the injection molding machine switches to the cooling stage. The amount of cooling is determined by the cooling time. After the cycle is complete and before the next cycle can be run, the machine must be purged per directions in the manual. [9]

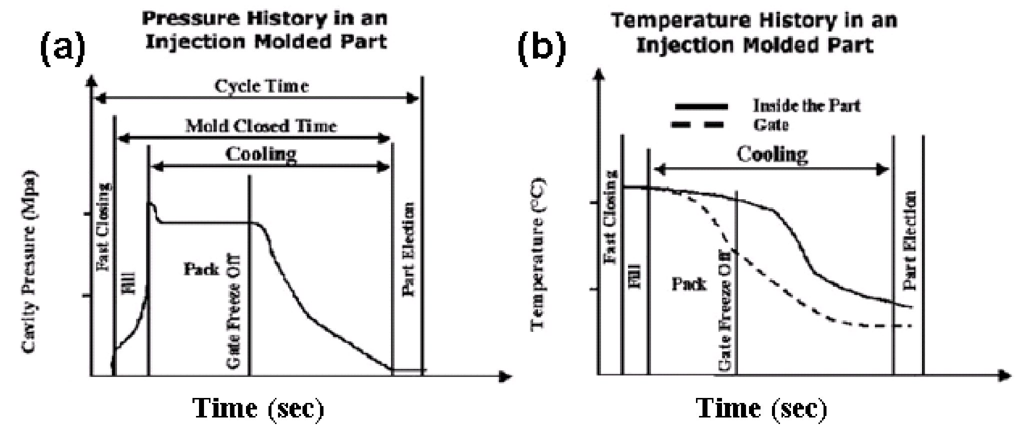


Figure 3 : Typical pressure and temperature cycle of injection molding [9]

2.2 Injection molding machine components

The injection molding machine consists of the hopper, screw, the barrel and the injection nozzle.

2.2.1 Hopper

In the molding process the plastic materials are supplied in the form of small pellets. The hopper functions as the holder of these pellets. The pellets are then gravity fed from the hopper to the barrel.

2.2.2 The barrel

The main use of the barrel is to give support for the screw. The Barrel consists of heater bands which function as a temperature recorder for each section of the barrel.

2.2.3 The Screw

Also known as the reciprocating screw is used in compressing, melting and conveying the plastic material. The Screw consists of three zones – The feeding zone,theTransition zone and the metering zone. In the feeding zone there will be no change to the plastic materials and they will remain pellets and will be transferred to the next zone which is the transition zone , In this zone melting of the pellets will occur and the molten plastics will be transferred to the next zone which is the metering zone , In this zone the molten material will be ready for injection.

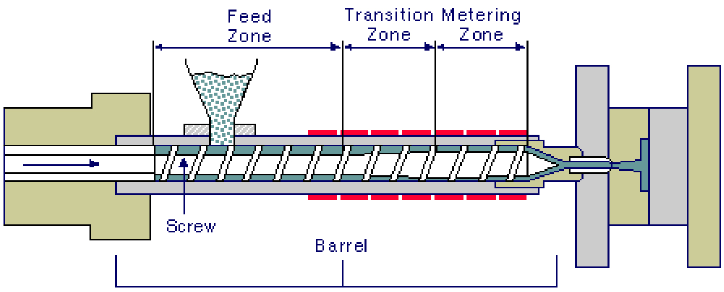


Figure 4 Different Zones of the Screw [21]

2.2.4 The Nozzle

The main function of the nozzle is in connecting the barrel to the sprue bushing which in turn forms a seal between the mold and the barrel. It's essential that the nozzle temperature should be set to the materials melt temperature.[21]

2.3 Material Analysis

Polymers play huge role in the injection molding process; most polymers are classified into three groups - thermoplastics, thermosets, and elastomers. The thermoplastics can be divided into two types - those that are crystalline and those that are amorphous.[25]

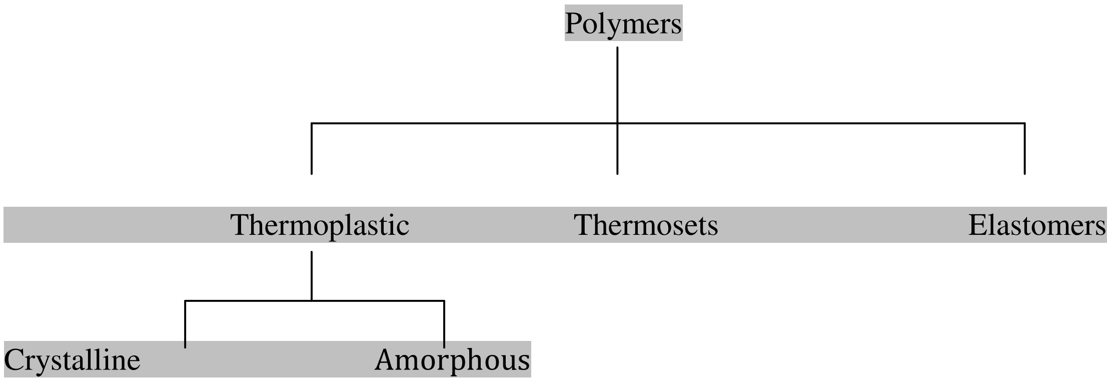


Figure 5 Classification of polymers

Polymers are the assembly of small organic repeating molecules called monomers. Properties of monomers include high resistivity to chemicals and can also be thermal and electrical insulators.

Thermoplastic materials generally soften when they are exposed to heat and return to their previous condition when cooled this is because there molecules are held by a weak intermolecular force. The property where they are repeatedly melted and cooled gives them a similar property as that of metals. The major thermoplastic groups are all produced by chain polymerization. Due to their recyclable property they are used in a wide range of applications such as Food packaging, insulation, automobile bumpers and credit card holders.

Table 1 Common thermoplastic groups

Themoplastic Groups	Common examples
Polyolefines	LDPE , HDPE, PP
Styrenics	PS, ABS
Vinyls	PVC
Acrylics	PMMA
Fluoropolymers	PVDF
Polyesters	PET
Polyamides (Nylons)	Nylon 6 , Nylon 66

Polyimides	PI
Polyethers	PC
Sulphur containing polymers	PES

Thermosetting plastics or simply known as thermosets solidifies irreversibily when heated. Thermosets cannot be reshaped by heating. Thermosets are usually three-dimensional networked polymers in which there is a high degree of cross-linking between polymer chains. The rigidity of the material is caused by cross-linking which restricts the motion of the chains. Thermosets are strong and durable. They primarily are used in automobiles and construction. They also are used to make toys, varnishes, boat hulls and glues.

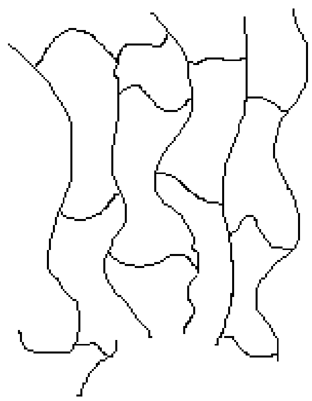


Figure 6 Thermoset Structure [23]

Elastomers are rubbery polymers that can be stretched easily to several times and which rapidly return to their original dimensions when the applied stress is released.Elastomers are cross-linked, but have a low cross-link density. The polymer chains still have some freedom to move, but are prevented from permanently moving relative to each other by the cross-links.[23] To stretch, the polymer chains must not be part of a rigid solid - either a glass or a crystal. An elastomer must be above its glass transition temperature, Tg, and have a low degree of crystallinity. Many elastic materials including rubber bands are made of elastomers.

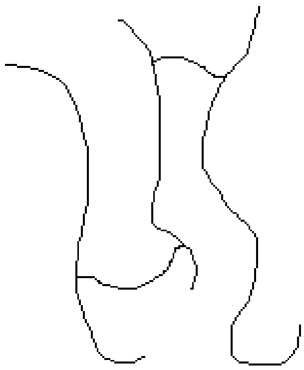


Figure 7 Elastomer Structure [23]

Highly crystalline polymers have properties like high rigidity, high melting point and are less affected by solvent penetration. The higher the crytallinity of polymers, the stronger the polymer but there impact resistance will be weakened. Small molecules and ions form a three-dimensional lattice with an extended regular structure that makes large crystals possible.

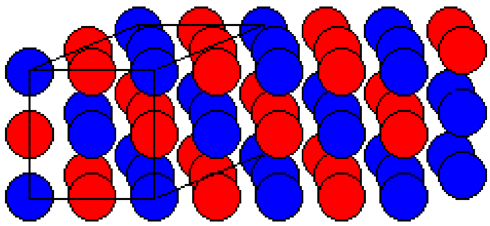


Figure 8 Crystalline structure [23]

Polymers which cannot pack together regularly to form crystals and polymer chains with irregular groups are known as Amorphous. These polymers are made of randomly coiled and entangled chains like spaghetti. Amorphous polymers are softer, have lower melting points, and are penetrated more by solvents than are their crystalline counterparts. [23]

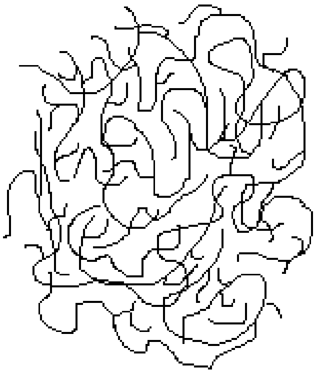


Figure 9 Amorphous structure [23]

2.4 Some problems associated with Injection molding

It's a common coincidence that defects in injection molding arise from the mold design imperfection and inappropriate material compounding.

Mold seal clearance, inappropriate clamping force and melting temperature along with non-uniform setting times play the major role in lowering the product quality. Besides, the usage of low grade polymer with inappropriate mass–mass ratio would also be a non-conformance in the production of a quality product.

The following below are some of the problems that might arise in the production line. These researches are done by ExxonMobil Chemical Company for the material Polypropylene (PP).This list will highlight problem areas of Injection Molding and a typical cause and method of resolving that problem.

Table 2 : Injection molding problems and solutions [6]

Problem	Cause	Possible solution
Sink marks	-Part is under filled	-increase shot size -increase cavity or hold pressure -Reduce fill rate -Open gates Improper gate location or design
Voids	-Part is under filled or has excessive shrinkage	-poor venting -improper gate location -Injection rate too high
Shrinkage	-Volume decreases as plastic cools -Insufficient cooling time	-increase hold time -improperly balanced cavity and core temperature -Runners or gates too small
Flash	-insufficient clamping force -	- increase clamp force -clean mold surfaces -change gate locations
Burning	-Compressed air in the mold degrades the resin	-decrease peak cavity pressure -clean vents , increase the size or number of vents -reduce melt temperature
Warp	-non uniform stress due to excessive orientation	-Part ejected too hot (so increase cycle time) -decrease injection fill rate -Flow too long , Insufficient gate -change gate location
Brittle parts	-Excessive orientation -improper design -degradation of resin	-increase injection fill rate -increase melt temperature -design properly , adequate radii at corners, notches or threads

2.5 Plastic Injection Molds

A mold is simply a machined steel plate with cavities into which plastic resin is injected to form a part. [13] A mold consists of two halves into which the impression of the part product to be molded is cut. The mating surfaces of the molded halves are accurately machined so that no leakage of plastic can occur at the split line. If leakage occurs it would be expensive to remove. [7]

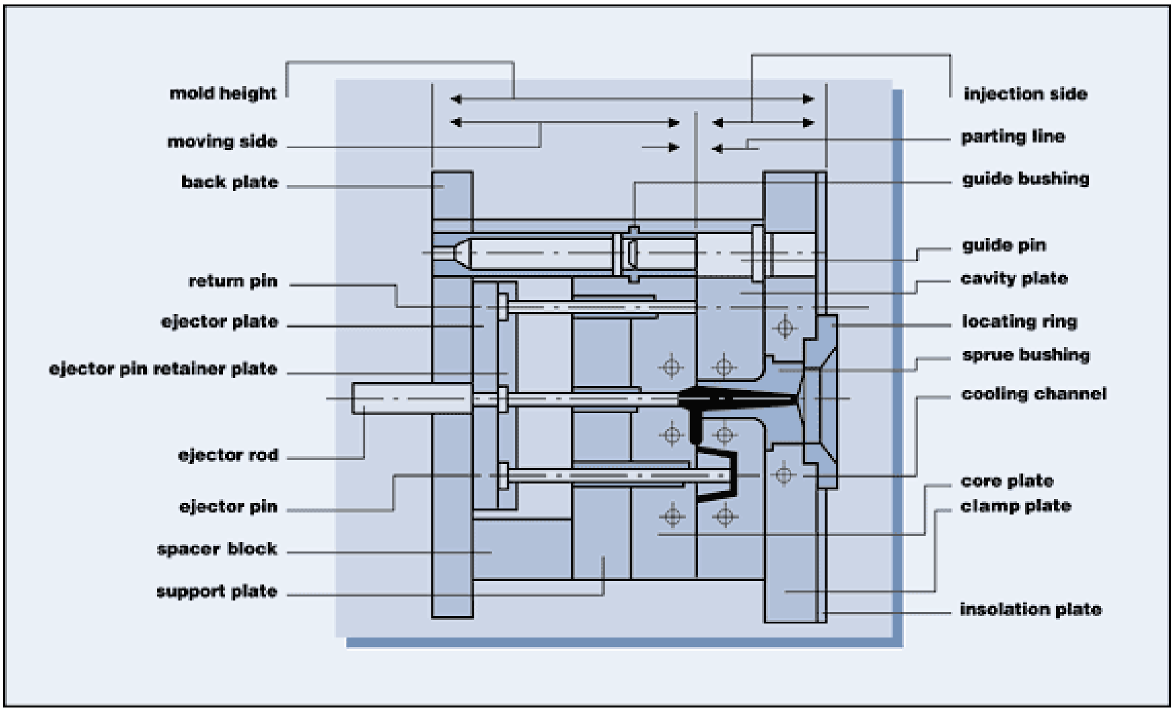


Figure 10 : Impression of a standard injection mold [8]

2.5.1Plastic Injection mold components & their function

The following table below gives a bird's eye view of a complete mold with each componentand their respective functions.

Table 3 : Mold components and their functions

Mold Component	Functions
Mold base	Hold cavities in a fixed position relative to machine nozzle
Guide Pins	Maintain proper alignments of two halves of the mold
Sprue bushing	Provides means of entry into mold interior
Gates	Control flow to cavities
Runners	Convey molten plastics from sprue into cavities
Cavity and Core	Control size, shape and surface texture of molded article
Water channels	Control temperature of mold surfaces to cool plastic to rigid state
Vents	Facilitates escaping of trapped air and gas
Ejector mechanism(pins, blades, stripper plate)	Eject rigid molded article from cavity or core
Ejector return pins	Return ejector pins to retracted position as mold closes for next cycle

2.5.2. Classification of molds

Wide range of molds can be classified into these 4categories:

1) The cold runner two plate molds
2) The cold runner three plate molds
3) The hot runner molds
4) The insulated runnermolds

The cold runner mold is the simplest type of mold which consists two plates, the cavity and the core. The main reason for being called two plate molds is that they consist of one parting line and the mold splits into two halves. This makes it ideal to place the runner system on the parting line.[9]

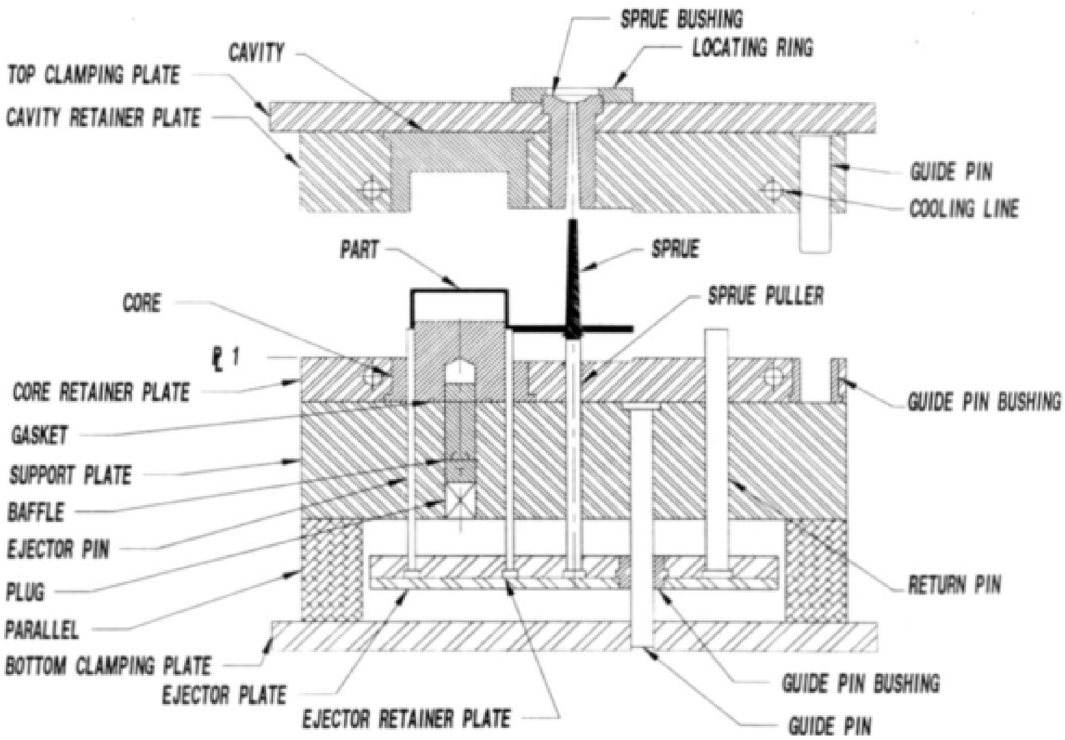


Figure 11 : Two plate cold runner mold [17]

The cold runner three plate molds consist of three plates - The stationary plate, The middle plate and The movable plate. The stationary plate also known as the runner plate is where the sprue and half of the runner are placed. The middle plate commonly known as the cavity plate as the figure below clearly indicates it consists of half of the runner and also the gate. And the remaining plate is the movable plate also named the force plate contains the product produced from the mold and the ejector system which is applicable in the removal of the molded part. The main reason for using this mold is because of their flexibility in gating locations.[9]

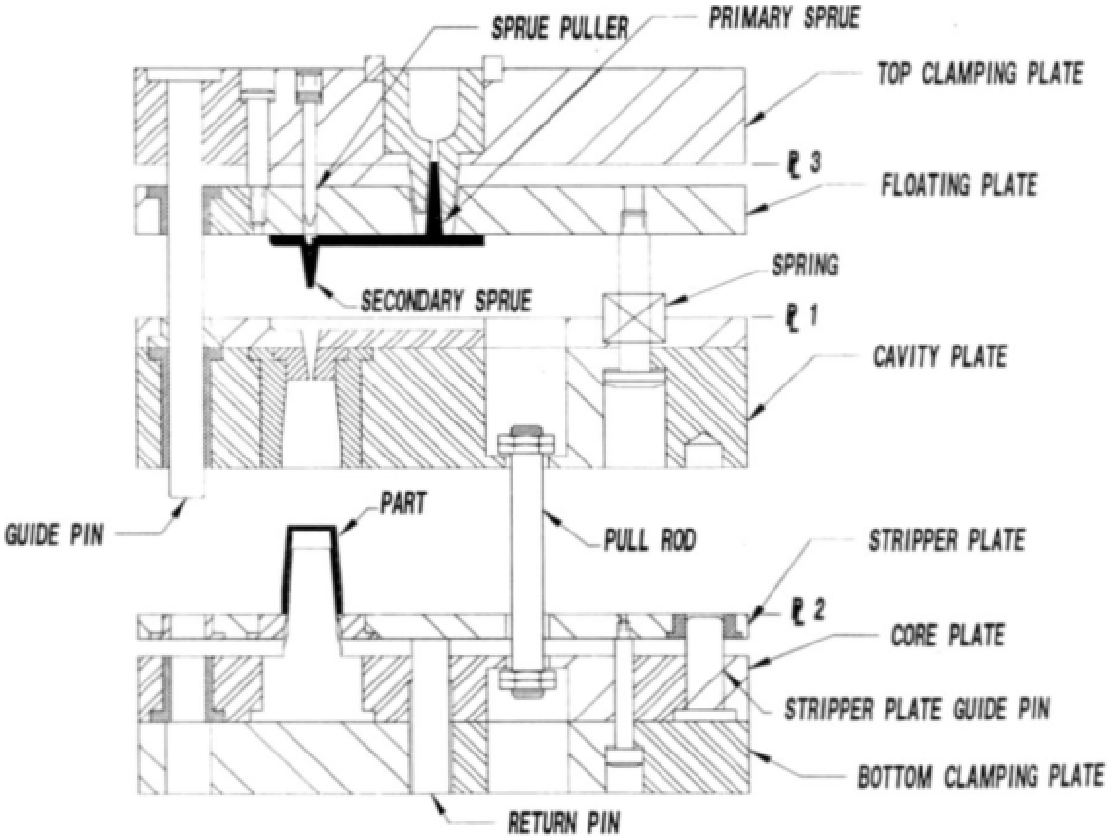


Figure 12 : Three plate cold runner mold [17]

The main difference between the two molds is that the three plate mold has two parting planes, and the mold splits sections every time the part is ejected.

In a hot runner mold, the runner is situated internally in the mold and kept a temperature above the melting point of the plastic. Runner scrap is reduced or eliminated. The major disadvantage of a hot runner is that it is much more expensive than a cold runner, it requires costly maintenance, and requires more skill to operate. Color changes with hot runner molds can be difficult, since it is virtually impossible to remove all of the plastic from an internal runner system.

Hot runners have many advantages to name a few: - They can completely eliminate runner scrap this is helpful so there are no runners to sort from the parts, and no runners to throw away or regrind. Hot runners are popular in high production parts, especially with a lot of cavities.

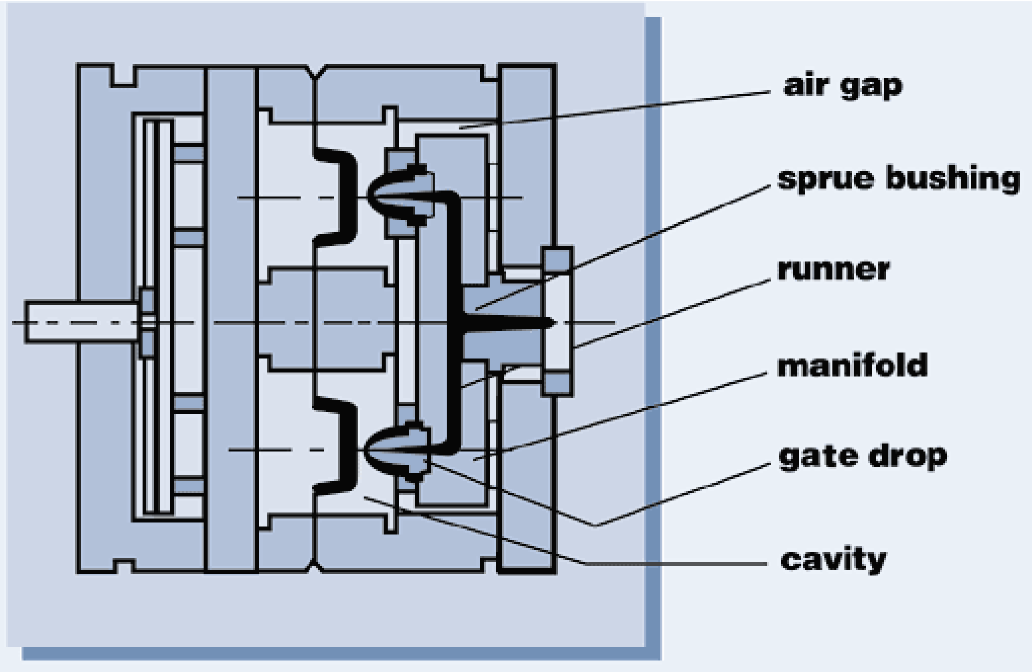


Figure 13 : Cross-section of a basic hot runner system [8]

Insulated runner molds have oversized passages formed in the mold plate. The passages are of sufficient size that, under conditions of operation, the insulated effect of the plastic (frozen on the runner wall) combined with the heat applied with each shot maintains an open, molten flow path. The insulated runner system should be designed so that while the runner volume does not exceed the cavity volume all of the molten material in the runner is injected into the cavity during each shot. This helps prevent excessive build-up of the insulating skin and minimizes any drop in melt temperature.

Advantages and Disadvantages of Insulated runner molds compared to cold runner mold can be found in the table below.

Table 4 : Comparison between insulated runner and cold runner mold

Advantages	Disadvantages
Reduction in material shear	More complex tool design
Faster cycle times	Higher tool cost
Elimination of runner scrap	Higher maintenance cost
Decreased tool wear	Startup procedure is more difficult
Improved part finish	possible thermal degradation of material
Shorter Cycle times	Color change is considered difficult

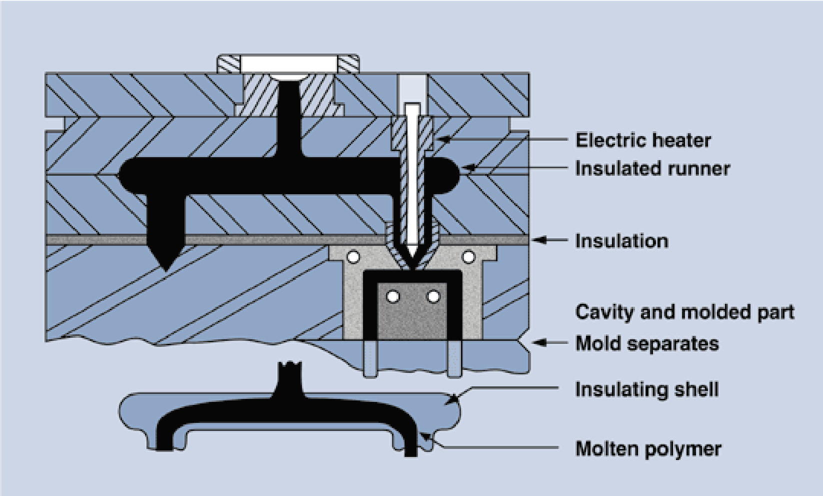


Figure 14 : Insulated runner mold [8]

2.6 Gates

Gates are a transition zone between the runner system and the cavity. The location of gates is of great importance for the properties and appearance of the finished part. The melt should fill the entire cavity quickly and evenly. For gate design the following points should be considered:

- Locate the gate at the thickest section
- Note gate marks for aesthetic reasons
- Avoid jetting by modifying gate dimensions or position
- Balance flow paths to ensure uniform filling and packing
- Prevent weld lines or direct to less critical sections
- Minimize entrapped air to eliminate burn marks
- Avoid areas subject to impact or mechanical stress
- Place for ease of de-gating

Commonly Used gate types include Sprue gate, Edge gate, Tab gate and Fan gate

Sprue gate is recommended for single cavity molds or for parts requiring symmetrical filling. This type of gate is suitable for thick sections because holding pressure is more effective. A short sprue is favored, enabling rapid mold filling and low-pressure losses. A cold slug well should be included opposite the gate. The disadvantage of using this type of gate is the gate mark left on the part surface after the runner (or sprue) is trimmed off. Freeze-off is controlled by the part thickness rather than determined the gate thickness. Typically, the part shrinkage near the sprue gate will be low; shrinkage in the sprue gate will be high. This results in high tensile stresses near the gate.

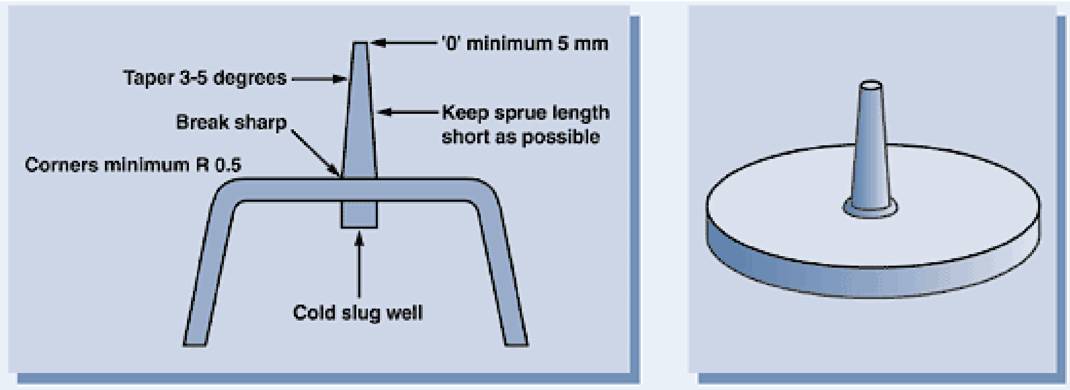


Figure 15 : Sprue gate [8]

The edge or side gate is suitable for medium and thick sections and can be used on multi-cavity two plate tools. The gate is located on the parting line and the part fills from the side, top or bottom.

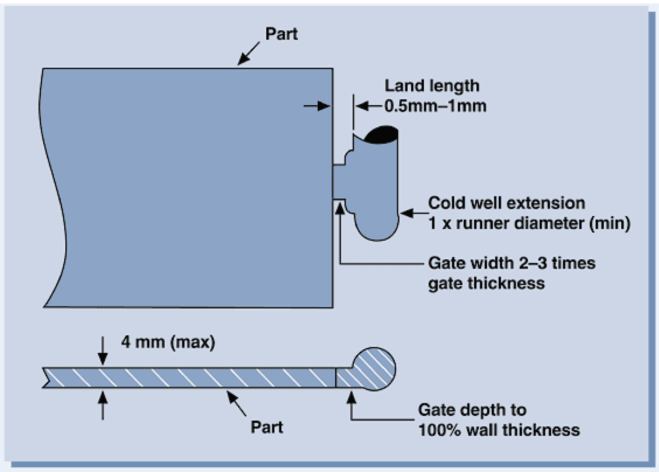


Figure 16 : Edge gate [8]

A tab gate is typically employed for flat and thin parts, to reduce the shear stress in the cavity.The high shear stress generated around the gate is confined to the auxiliary tab, which is trimmed off after molding.

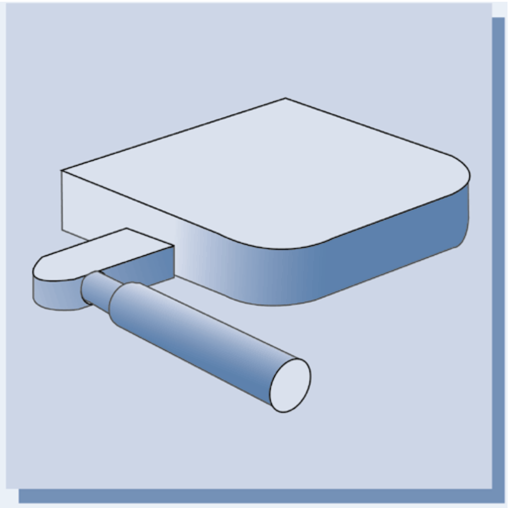


Figure 17 : Tab gate[8]

A fan gate is a wide edge gate with variable thickness. This type is often used for thick-sectioned moldings and enables slow injection without freeze-off, which is favored for low stress moldings or where warpage and dimensional stability are main concerns. The gate should taper in both width and thickness, to maintain a constant cross sectional area. This ensures that the melt velocity will be constant, entire width is being used for the flow and the pressure is the dam across the entire width.

2.7 Sprue

Sprue is the way the molten plastic flows into the mould by injecting the molten material from the nozzle of the machine into runner system and then the mold. To allow smooth material flow the design of sprue should be round, smooth and tapered. It’s not wise to make the sprue extremely long since it’s going to be a waste. [20]

2.8 Runner systems

The runner system is a manifold for distribution of thermoplastic melt from the machine nozzle to the cavities. The sprue bushing and runners should be as short as possible to ensure limited pressure losses in the mold.

Correctly designing a runner system will benefit in the following areas

- Deliver melt to the cavities
- Balance filling of multiple cavities
- Balance filling of multi-gate cavities
- Minimize scrap
- Eject easily
- Maximize efficiency in energy consumption
- Control the filling/packing/cycle time.

2.9 Ejection System

The method of ejection has to be adapted to the shape of the molding to prevent damage. In general, mold release is hindered by shrinkage of the part on the mold cores. Large ejection areas uniformly distributed over the molding are advised to avoid deformations.

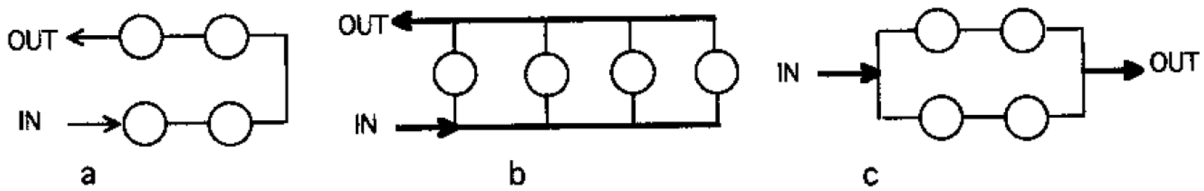
Several ejector systems can be used like Ejector pins or sleeve, Blades , Air valve and stripper plate. When no special ejection problems are expected, the standard ejector pin will perform well. In case of cylindrical parts like bosses a sleeve ejector is used to provide uniform ejection around the core pin. Blades are poor ejectors for a number of reasons: they often damage parts; they are prone to damage and require a lot of maintenance. Blade ejectors are most commonly used with ribbed parts.

2.10 Cooling System

Cooling is very important to remove heat efficiently and dissipate the heat of the molding quickly and uniformly. For efficient molding, the temperature of the mold should be controlled and this is done by passing a fluid through a suitably arranged channel in the mold.Adequate mold temperature control is important consistent molding. Setting the right mold temperature will help achieve optimal properties of engineering plastics. Some of the effects of mold temperature occur on the Mechanical properties , Shrinkage behavior, warpage, surface quality and cycle time.

2.10.1 Cooling Layout

There is no precise rule on which layout on the mold as long as the flow is uniform. But in many multicavity molds the cooling channel layout are partly in parallel and partly series.



a) Series cooling b) Parallel cooling c) Series-parallel cooling

Figure 18 : Different types of cooling layouts

2.11. Venting

It is the process of removing trapped air from the closed mold and volatile gases from the processed molten plastic.

Inadequate venting can cause corrosion of the mold, Discoloration due to compression of trapped air as plastic tries to force out, poor weld line strength and volatile gases will be absorbed by the plastic causing voids, blisters, bubbles and variety of other defects.

Below is the dimension of vent for common materials.

Table 5 : Vent depth of some materials

MATERIAL	VENT DEPTH (in.)
ABS	0.002
Acetal	0.0007
Acrylic	0.002
Cellulose Acetate	0.001
Ionomer	0.0007
Nylon 6/6	0.0005
Polycarbonate	0.002
Polyethylene	0.001
Polypropylene	0.001
Polystyrene	0.001

2.12 Injection mold steel selection

The most common steel types that are used for the fabrication Injection molds are

1) pre-hardened mold and holder steels
2) Through-hardening mold steels
3) Corrosion resistant mold steels.

1) The Pre-hardened mold and holder steels

These steels are mostly used for large molds, molds with low demands on wear resistance and high strength holder plates

The Surface hardness can be increased by flame hardening or nitriding. Pre-hardened mold steels are used for large molds and molds with moderate production runs. This steels are delivered in the hardened and tempered condition, Usually within the 270-350 Brinell range. No heat treatment is necessary before is put into use.[12]

2) Through hardened Mold Steels

These steels are mostly used for long production runs, to resist abrasion from certain molding materials and to counter high closing or injection pressures

These steels are usually rough machined, hardened and tempered to the required hardness and often polished or photoetched.

Better wear resistance is especially important when filled or reinforced plastic materials are used. Resistance to deformation and indentation in the cavity, gate areas and parting lines helps to maintain part quality.

Better polishability is important when high surface finishing is required on the molded part.

3) Corrosion Resistant Mold steels

If a mold is likely to be exposed to corrosion risk, then stainless steel is strongly recommended. The increased initial cost of this steel is often less than the cost involved in a single repolishing or replating operation of a mold from conventional steel.[12]

Plastic Molds can be affected by corrosion in several ways:-

- Plastic materials can produce corrosive by-products
- Reduction of cooling efficiency when water channels become corroded

3. Methods and Result

In this Part the main details that are discussed are the appropriate designing methods to accomplish the mold design for the required product.

The appropriate steps in designing a mold are

1) Product design
2) Mold design

3.1 Software assistance

Many software's can be used in designing the required product with accurate calculations and minimized time spam.

- 2D and 3D modeling, assembly and part designing: AutoCAD, Solid edge
- Analysis and simulation of the phases of injection under varying temperature and pressure: Mold flow
- Location of injection point for the required product which in turn gives adequate filling of cavity : Mold flow
- Manufacturing, simulation and finding the appropriate machine time : MasterCAM

3.2 Product Design

The mold that is designed in this this project is for a wallet that can hold a credit card aswell as USB. In todays industry we mostly see products as similar as this but almost all of them are made independantly(as a credit card holder or USB holder) , But this contains both in one place which is more convinient and solves the common problem of losing once own credit card or USB.

The Figure below shows the product design for which a mold is designed , The product is made for Universally accepted dimension of Credit card which is (85.60 mm X 53.98 mm) and with a thickness of 0.762mm which is common for banking cards (ATM cards, credit cards and debit cards).

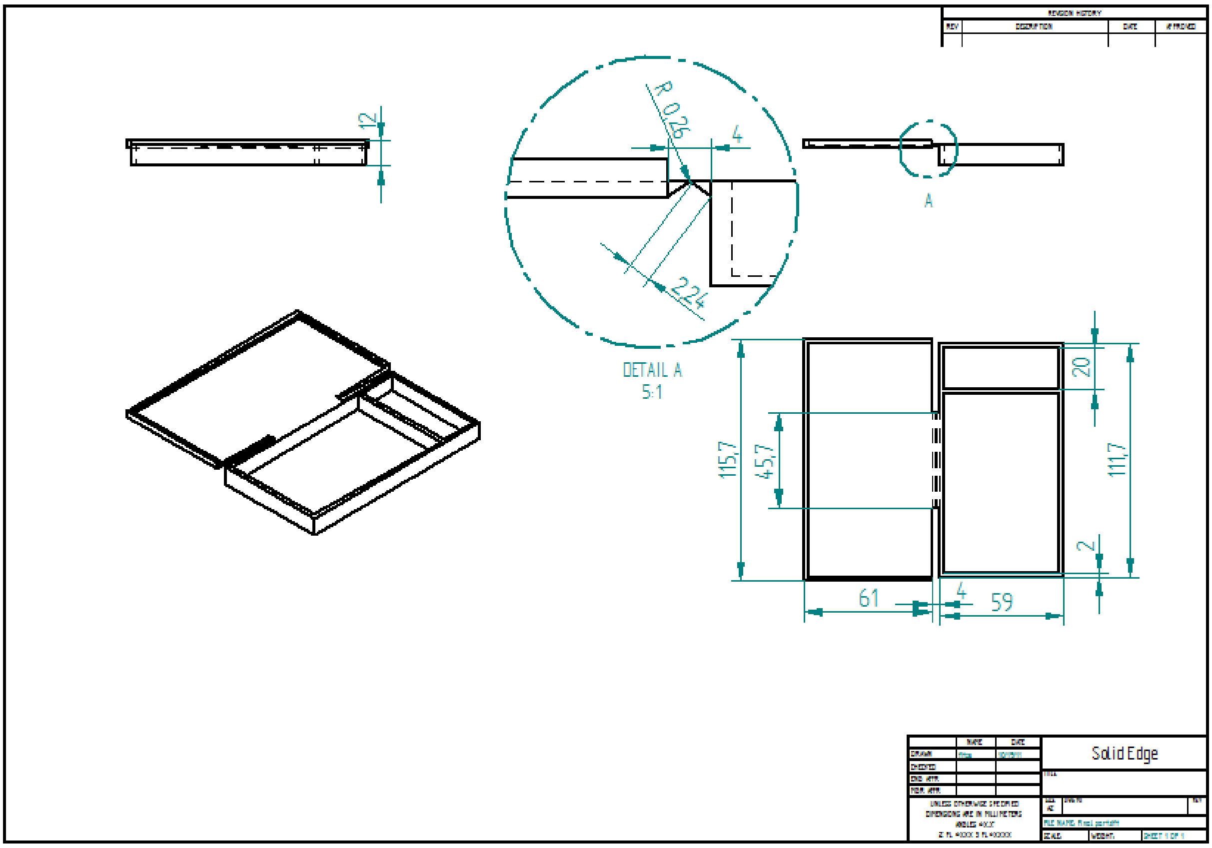


Figure 19 : Credit card and USB holder product design

3.3 Mold Flow analysis

Analysis of the product using Autodesk Mold flow (Simulation tool) software helps us validate and optimize plastic parts, injection molds, and the injection molding process. This software is essential for designing and mold making through simulation setup and results interpretation to show how changes to wall thickness, gate location, material, and geometry affect manufacturability. And also experiments with “what-if” scenarios before finalizing a design.

The first step in analyzing is importing the design into the moldflow software using IGES Format. After importing the design the next step is to mesh the product design using moldflow software.

3.3.1 Meshing

In order to analyze fluid flows, flow domains are split into smaller subdomains (made up of geometric primitives like hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D) that are collectively known as meshing. In the mold flow software there are three different ways of meshing a product design. Those are Midplane Mesh, Dual Domain mesh and 3D Technology.

The mesh type applied for this project is Dual Domain Mesh ,This type of mesh is considered as a surface mesh along with Midplane type. It provides the basis for the Dual Domain analysis, and it contains of a mixture of different types including regions with traditional Midplane elements and surface shell elements. This mesh type is appropriate when there are many thin regions in the part design. The reason for choosing this mesh type is it doesn't require too much time to obtain the analysis.

3.3.2. Meshing Steps

1) Open Moldflow software and import the part that is going to be analyzed with the Dual Domain option

2) After Import, Click on Create mesh and then generate mesh

3) In the mesh options it's advisable to leave the Global Edge lengths on default But if there is a need to increase to a finer mesh its recommended to shorten the edge length by half.

4) Finally click Mesh now

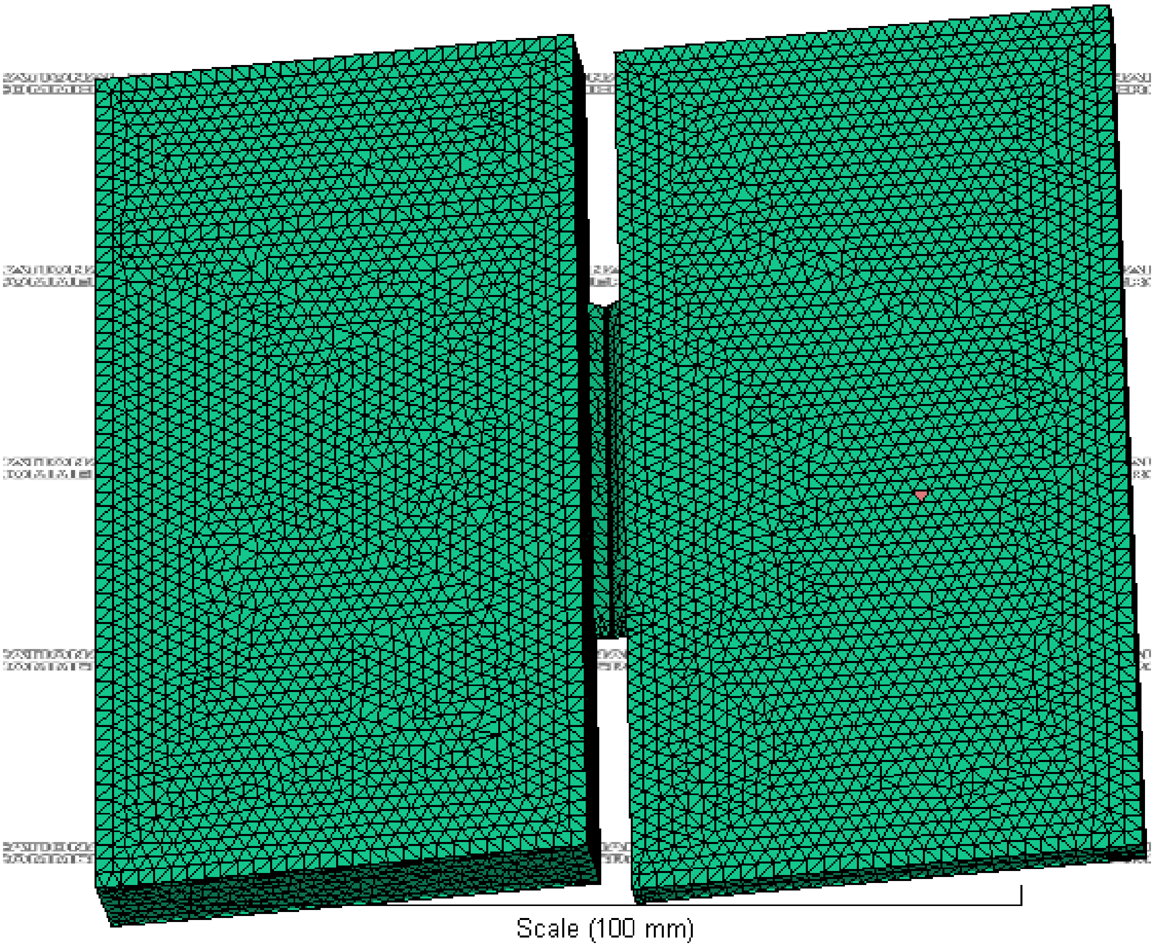


Figure 20 – Results of the mesh analysis

3.3.3. Mesh Statistics

After Mesh is done, it's almost compulsory to check the mesh statistics to check if there are problems that needs fixing making sure that the mesh has no flaws.

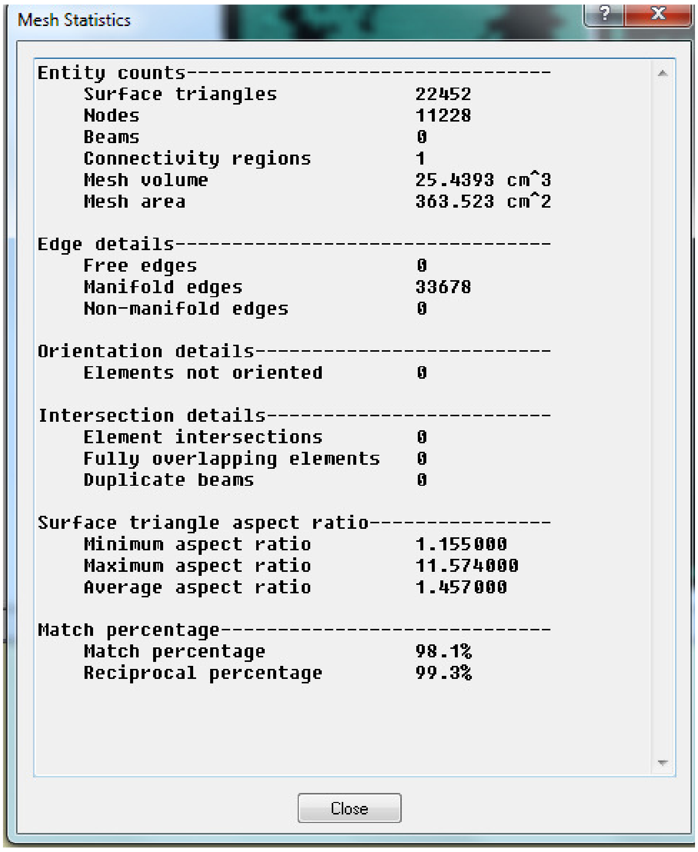


Figure 21 – Mesh Statistics for analysis

The first analysis about the mesh is the Connectivity Region. It represents how the region is connected as a whole design. It should always be connected and represented as one region, therefore if the value is more than two it means that there is a disconnected region somewhere in the design.

The second analysis will be the Edge Details; it is where the surface edge is checked theFree Edges has to be zero; meaning that all the edges in the elements are connected. A Manifold Edge is a mesh edge that has two elements attached to it. A Non-Manifold Edge means the edge that has more than two elements attached to it. This is not acceptable in the Dual Domain; therefore the value has to be zero as well.

Intersections Details shows how the shared surfaces are reported. Since intersections and overlapping elements are not allowed the value for that should also be zero.

3.3.4. Fill Analysis

The main benefit of doing fill analysis are predicting Fill pattern this will help us understand some of the flaws if any in the product. Some other benefits include Reduce scrap, Balance filling and pressure distribution, Material selection, Determining the clamp force, Identifying weld line and gas trap locations and also identify shear stress levels.[19]

This analysis can help predict short shots. Short shots are a legitimate concern for those involved in creating plastic parts. If you have a component with variable wall thickness, it is important to run an analysis to make sure these areas will fill out. [18]

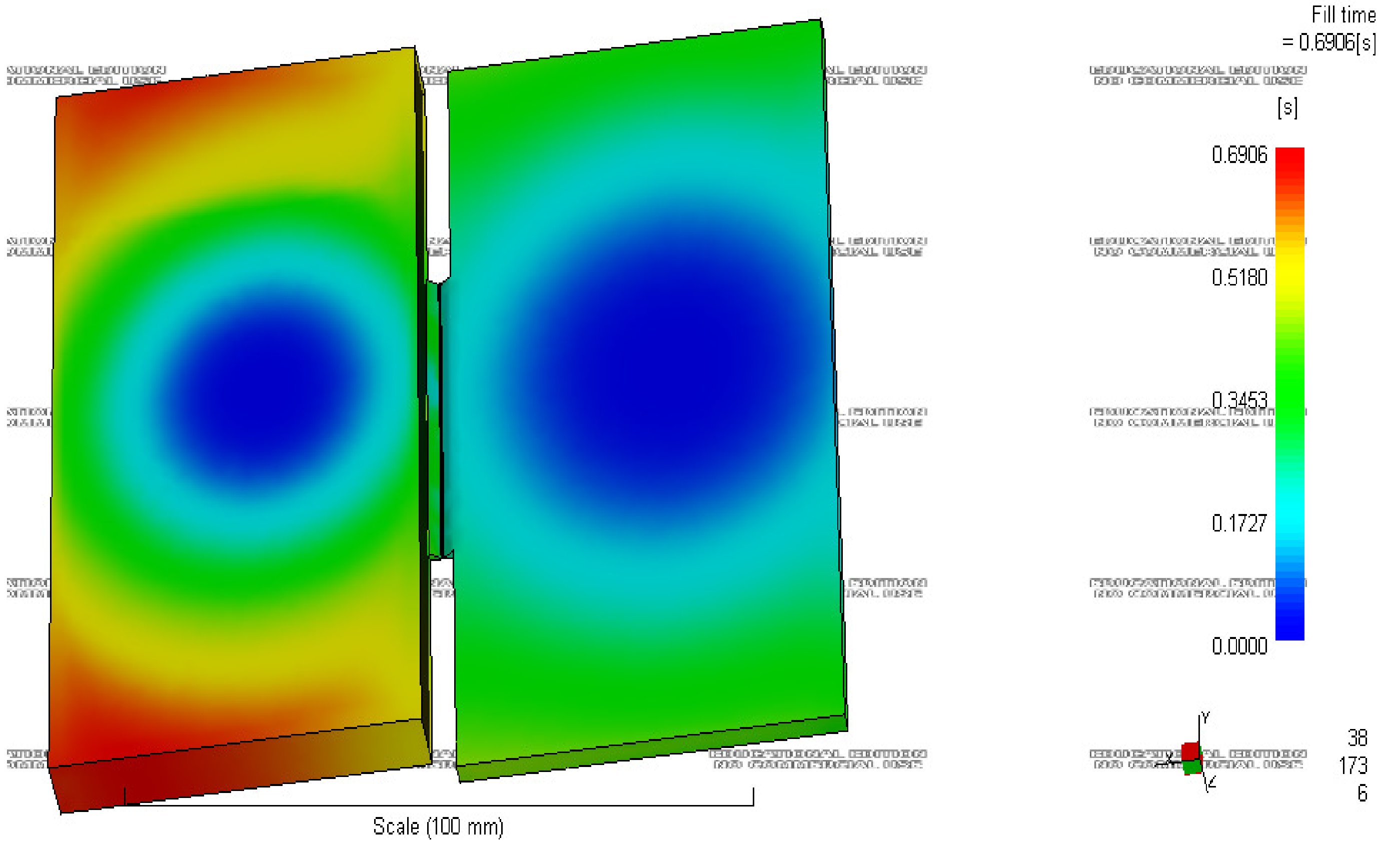


Figure 22- Fill analysis

In this Fill analysis it's recommended to have to two injection points to make the fill time faster and more efficient. This analysis is made for a single cavity mold and the table below shows the summarized fill analysis for the design.

Table 6. Fill analysis Results

Parameters	Results
Mesh Type	Dual Domain
Fill Time	0.6906 Secs
Maximum Clamp force	16.9692 tonnes
Maximum Injection Pressure	19.1341 tonnes

3.4 Mold Design

When making a mold design, the following parts should be included Back plate for cavity and core side, Ejector plate, retaining plate, guiding pillars, spacer blocks, cavity plate and core plate. Designing a mold is a very complicated and important part of the injection molding process. When designing a mold, the designer needs to take many factors into account in addition to the actual shape of the mold. Warpage, shrinking, venting, residual stress and runner size are just a few of the factors that must be weighed while in the design stage.

The Full mold in the figure below has a property of steel and includes a back plate that holds the movable side of the mold like spacer block, support plate, cavity plate and ejector mechanism to the movable platen of the injection machine.The design of the back plate for the mold has a dimension of (156mm x 156mm) and a depth of 17mm.Then it's the ejector plate often referred to as the ejector covet plate which provides backup for pins set into the ejector-retaining plate. The retaining comes on top of the ejector plate and functions in holding ejector pins. The travel space for the ejector plates and ejector pins is provided by the spacer block.

To give the plates strength and rigidity it's smart to add a support plate. The support plate is followed by the core plate which holds the core element which is the mating half of the cavity. The core plate is inserted to the cavity which is where the plastic material is formed. To secure the stationary side of the mold to the machine it's necessary to have the clamping plate or also known as back plate for the core.

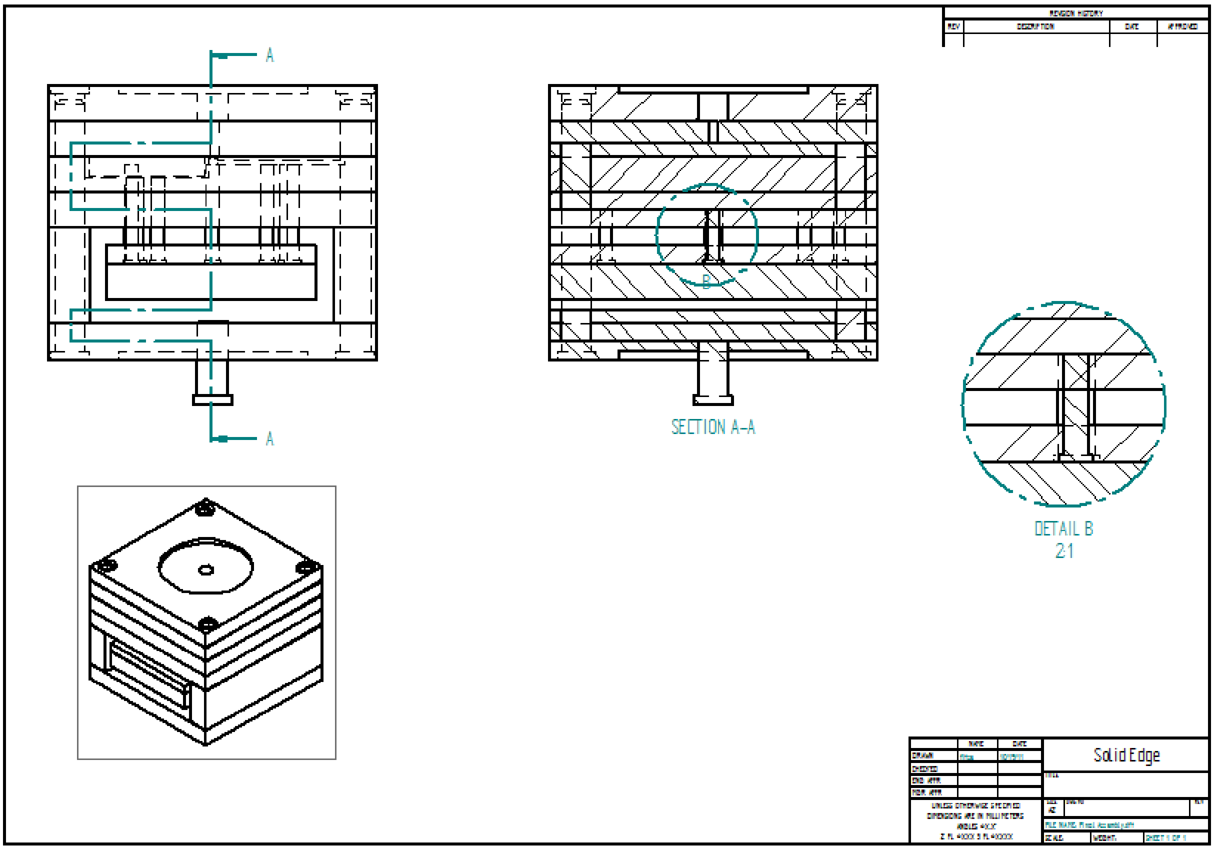


Figure 23 Section view of the mold

The Section view helps improve the visualization of the mold designs, clarify multiview drawings and facilitate the dimensioning of drawings. For the drawing above section views are used to reveal interior features of an object that are not easily represented using hidden lines.

The section view below will clearly reveal where every part list lies.

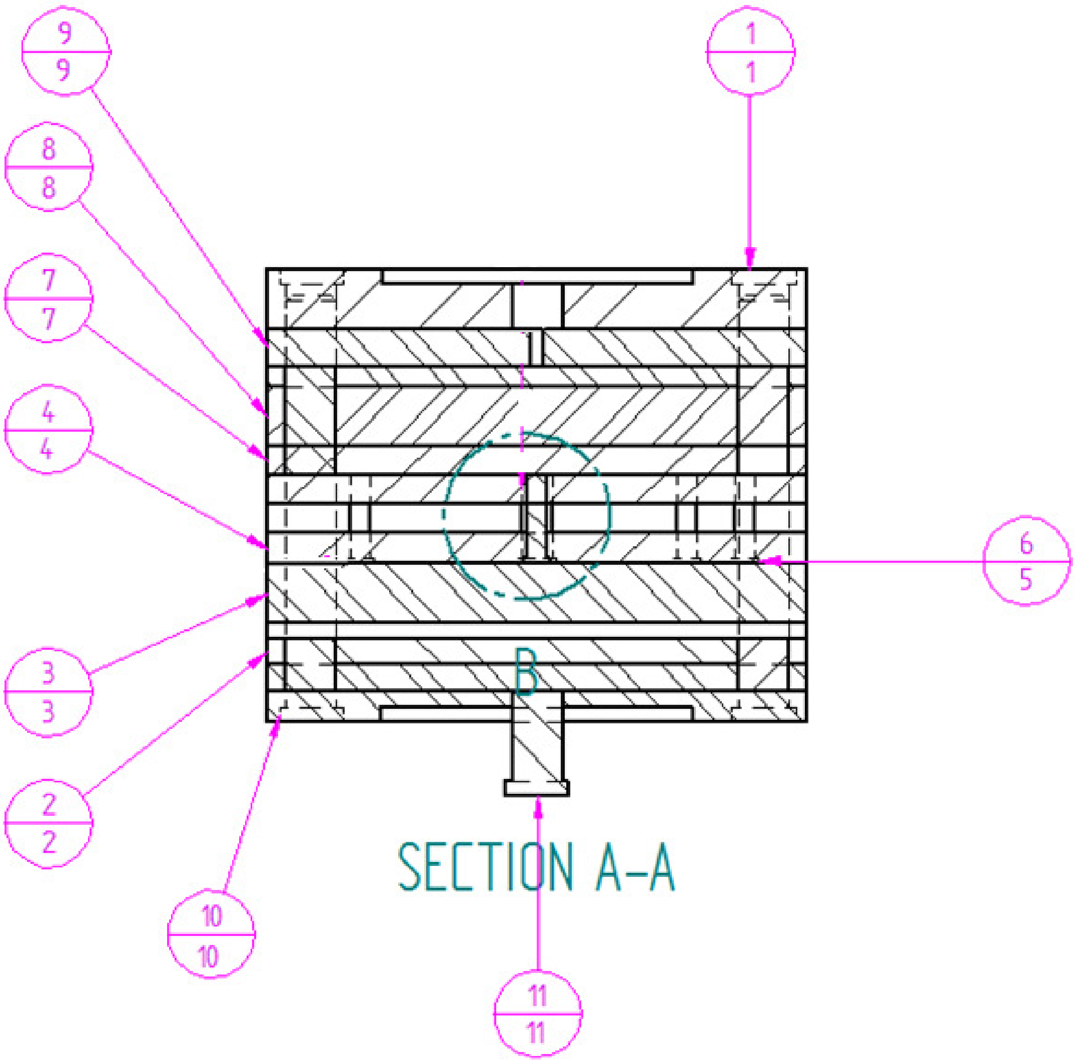


Figure 24 Mold detailed part list

Item Number	Title	Material
1	Back Plate for Core (156 mm x 156 mm)	Steel
2	Spacer block (156 mm x 20 mm)	Steel
3	Ejector Plate (156 mm x 100 mm)	Steel
4	Retaining plate (156 mm x 100 mm)	Steel
5	Ejector Pin (Ø 10,6 mm)	Steel
6	Retaining pin (Ø 8 mm)	Steel
7	Support plate (156 mm x 156 mm)	Steel
8	cavity plate (156 mm x 156 mm)	Steel
9	core plate (156 mm x 156 mm)	Steel
10	Guiding pillar (Ø 18,5 mm)	Steel
11	Ejector pin for ejection (Ø 18,5 mm)	Steel

3.4.1 Cavity plate and Core plate

A mold system is an assembly of platens and molding plates typically made of tool steel. The mold system shapes the plastics inside the mold cavity and ejects the molded part. The cavity plate is generally mounted on the stationary platen. The core plate moves with the moving platen guided by the tie bars. Cavity plates are plates that the actual form of your desired plastic component is cut into. The desired plastic product comes from the combination of the core and cavity plate. To achieve the plastic product in Figure 19, the cavity and the core plate should look like the figures below.

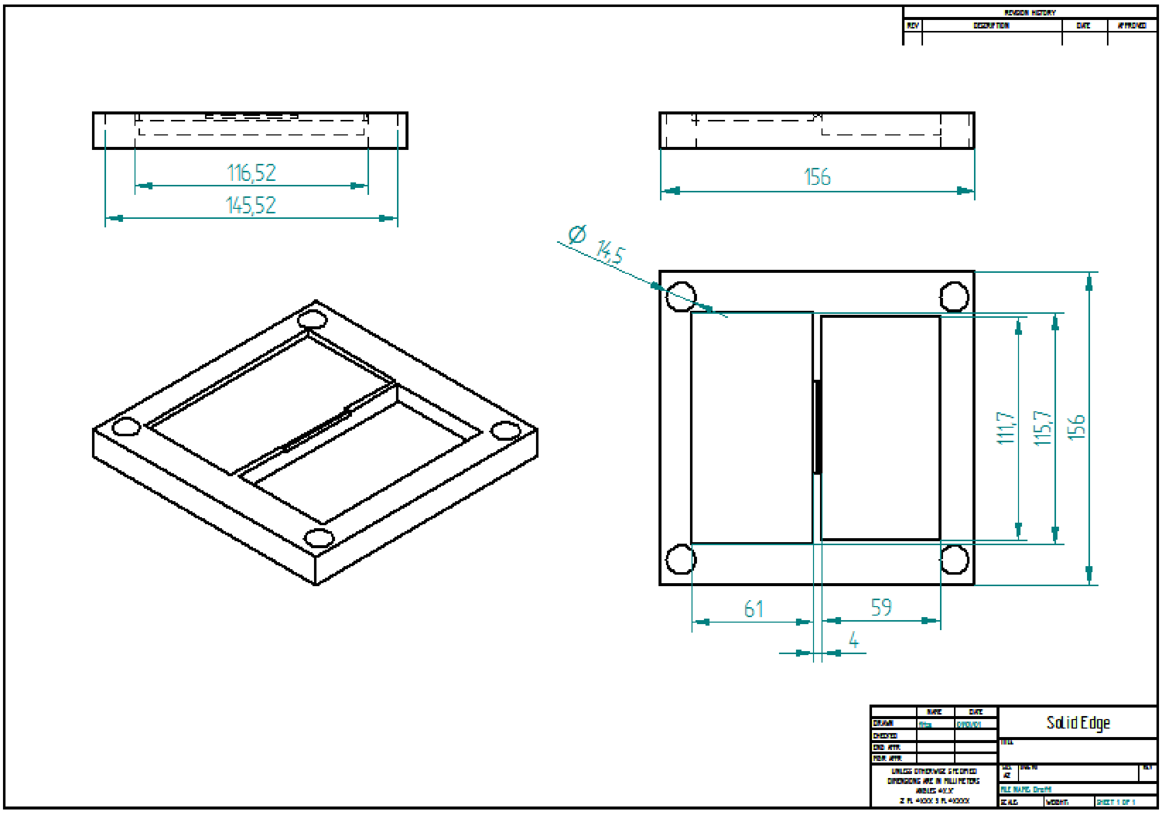


Figure 25 Technical drawing of the cavity plate

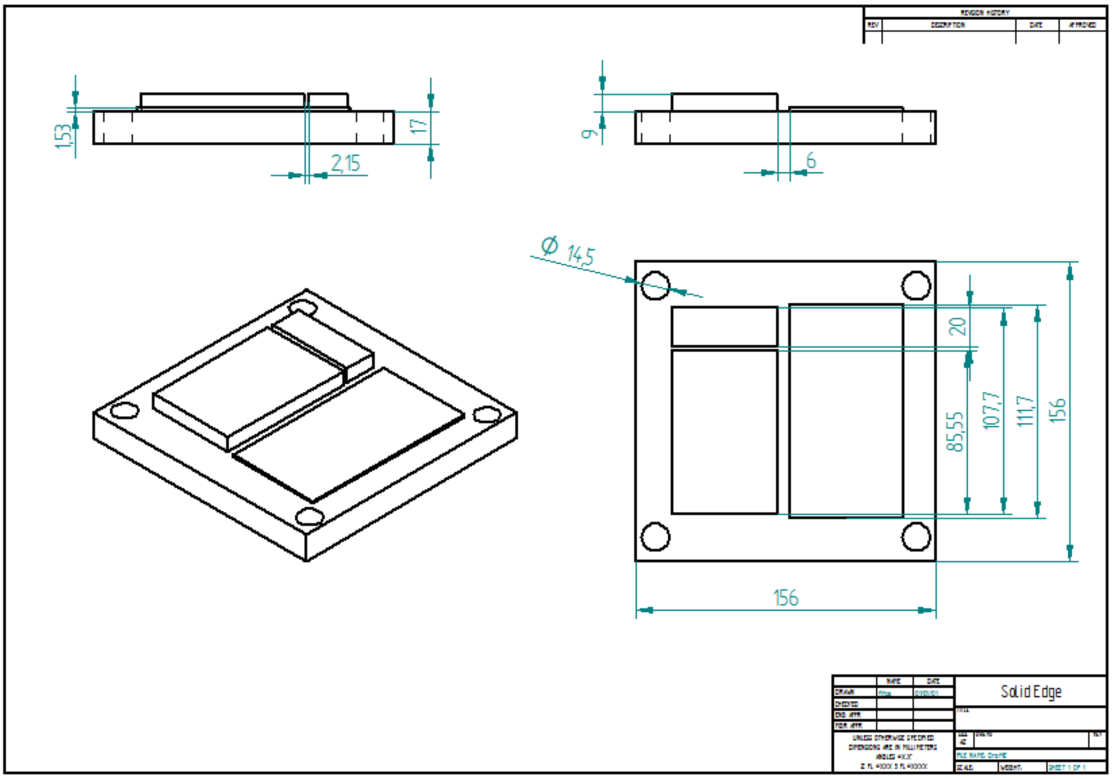


Figure 26 Technical drawing of the Core plate

An exploded view of the mold shows relationship of assembly of various parts. It shows the components of an object slightly separated by distance, or suspended in surrounding space in the case of a three-dimensional exploded mold. And also how those mold parts fit together.

The reason for the figure below to being a bit diagonal is that the projection of an exploded view is normally shown from above and slightly in diagonal from the left or right side of the drawing.

The figure below shows detailed parts of the full mold which is designed by Solid edge.

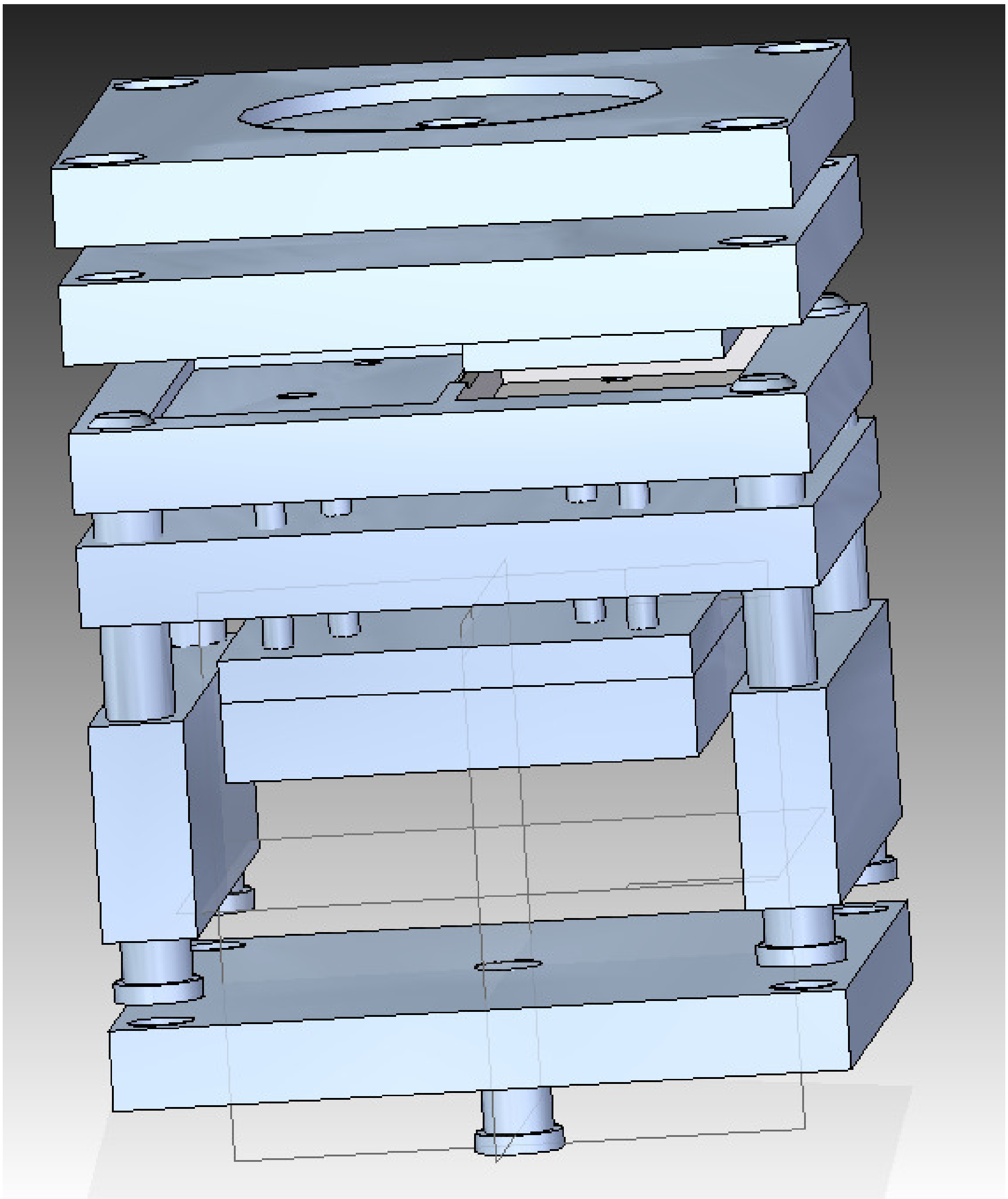


Figure 27 Exploded view of the mold

.

4. Conclusions

In this Engineering thesis work a design of a mold for a wallet with dual application was made. The applications were credit card and USB holding capability. The Software Solid Edge was crucial in designing all the necessary parts of the mold. With the manufacturing process in mind it was necessary to choose a material that can coup with properties of real time mold making and Steel was the perfect fit.

In making the mold it was necessary to have the best possible product design So that it won't complicate the mold designing process. With all the necessary dimensions and by the help of Solid Edge product design was achieved. In this phase there were lots of ups and downs in trying to figure out what the best closing system for the wallet might be, a lot of designs were drawn due to the fact that wallet closing system complication complicates the mold design which intern complicates the manufacturing process of the mold,

In this thesis it was crucial to find out if there were any defects in the product design and also finding out some important values like material selection, Fill time, Fill pattern and Clamping force. By using the simulation and analysis software Mold flow the above values have been achieved and there were no defects found on the product design ..

Further researches can be done on the above designs stage to put the layout of a Cooling system for the mold. The function of the cooling system of a plastic injection mold is to provide thermal regulation in the injection molding process. As the cooling phase generally accounts for about two-thirds of the total cycle time of the injection molding process, efficient cooling is very important to the productivity of the process.

5. Reference

1) All about plastic moulding. 2009. *The History of Plastic Moulding*. [ONLINE] Available at: http://www.plasticmoulding.ca/history.htm#Plastic. [Accessed 10 October 11].
2) Autronic plastics INC. 2008. Glossary of Plastic Injection Molding Terms. [ONLINE] Available at: http://www.apisolution.com/images/glossary_plastic_injection_molding_engineering_manufacturing.pdf [Accessed 18 August 11]
3) Pmold industrial limited. 2010. *The injection molding process*. [ONLINE] Available at: http://www.pmolds.com/news/4/15/The-injection-molding-process/. [Accessed 18 August 2011]
4) RUTLAND plastic Limited. 2007. *PLASTIC INJECTION MOULDING MACHINE*. [ONLINE] Available at: http://www.rutlandplastics.co.uk/moulding_machine.shtml . [Accessed 22 August 2011]
5) Murti, B, 2010. *Simulation and analysis of Injection Moulding product and rapid prototyping*. Bachelor. Finland: Arcada University.
6) ExonMobil Chemical. 2002. *Injection molding problems & solutions*. [ONLINE] Available at: http://www.scribd.com/doc/6200927/Injection-Molding-Problems-and-Solutions . [Accessed 25 August 11]
7) Crawford, R, 1998. *Plastic Engineering*. 3rd ed. Oxford: Butterworth – Heine-mann.
8) DSM engineering plastics. 2005. *Design guide*. [ONLINE] Available at: http://www.dsm.com/en_US/downloads/dep/designbroch05USweb.pdf . [Accessed 31 August 11].
9) Rosato, D;Donald,D;Marlene,D, 2000. *Injection molding handbook*. 3rd ed. Norwell massachusetts 02061: Kluwer Academic Publishers.
10) Herbert, R, 2001. *Understanding Injection Mold Design*. 1st ed. Cincinnati, USA: hanser Gardner Publications.
11) Plastic Troubleshooter. 2007. *VENTING THE MOLD*. [ONLINE] Available at: http://www.plastictroubleshooter.com/ThePlasticTroubleshooter/mold_venting.htm. [Accessed 1 September 11].

12) UddeholmMoulding. 2004. *Steels for moulds*. [ONLINE] Available at: http://www.uddeholm.co.jp/pdf/mould-english.pdf. [Accessed 5September 11].

13) DRM associates. 2007. *INJECTION MOLDING GLOSSARY*. [ONLINE] Available at: http://www.npd-solutions.com/injectmoldglos.html. [Accessed 10September 11].

14) Injection Mold Design Tutorial, Technology and Engineering. 2010. *Basic Functions of MoldBaseParts.*[ONLINE]Available at: http://mould-technology.blogspot.com/2008/02/basic-functions-of-mold-base-parts.html. [Accessed 10 October 11].

15) PlasticsOne. 2009. *Plastic Part Design*. [ONLINE] Available at: http://www.plastics1.com/Articles/CIM/62.php. [Accessed 12 October 11].

16) Rosato, D.V. Rosato, M.G. &Rosato, D.V., 2000. *Concise Encyclopedia of Plastics.* Springer: Kluwer Academic Publishers

17) KenPlas . 2011. *Platics molds*. [ONLINE] Available at: http://www.kenplas.com/mold/injectionmold/moldtype.aspx. [Accessed 17October 11].

18) Detriot Design Engineering. 2005. *Mold Filling Analysis*. [ONLINE] Available at: http://www.detroitdesign.net/moldfilling.htm. [Accessed 18 October 11].

19) EPS Flo Tek. 2007. *Filling Analysis*. [ONLINE] Available at: http://www.epsflotek.com/default.asp?upage_id=9414&apage_id=1153. [Accessed 18 October 11].

20) Injection mold making and molding. 2009. *Injection mold sprue and runner*. [ONLINE] Available at: http://imoldmaking.com/mold-making-2/mold-design/runner-and-gate/injection-mold-sprue-and-runner/. [Accessed 21October 11].

21) Machine components. 2009. *Injection Systems*. [ONLINE] Available at: http://www.dc.engr.scu.edu/cmdoc/dg_doc/develop/process/control/b1000002.htm. [Accessed24October 11].

22) Kamal, Musa R., 2009. Injection molding Technology and Fundamentals. 1st ed. Munchen, Germany: Hanser Publications.

23) Polymer chemistry. 2007. *Classification of polymers*. [ONLINE] Available at: http://faculty.uscupstate.edu/llever/Polymer%20Resources/Classification.htm. [Accessed27October 11].

24) elmhurst Chemical book. 2008. *Polymers*. [ONLINE] Available at: http://www.elmhurst.edu/~chm/vchembook/400polymers.html. [Accessed29October 11].
25) Tim A, Osswald, 2008. *Injection molding handbook*. 2nd ed. Munichen , Germany: Hanser Publications.
26) SubsTech Substances and Technologies. 2008. *Thermoplastics*. [ONLINE] Available at: http://www.substech.com/dokuwiki/doku.php?id=thermoplastics&DokuWiki=5c3eeba6c7368144fe7aa5d8c298d11b. [Accessed29October 11].

APPENDIX

- **Back plate –** It's the plate used as a support for the mold cavity block, guide pins, bushings, etc.
- **Brinell** - A test which measures the hardness or indentation of a material using a shaped point of specified weight.
- **Bubbles** -Air pockets that have formed in the material of the component.
- **Cavity** -The space inside a mold into which material is injected.
- **Clamping Plate** - A plate fitted to a mold and used to fasten the mold to a platen.
- **Cooling channels** - Channels located within the body of a mold through which a cooling medium is circulated to control the mold surface temperature.
- **Ejector Pins** - Pins that are pushed into a mold cavity from the rear as the mold opens to force the finished part out of the mold.
- **Glass transition temperature –** is when an amorphous polymer is heated, the temperature at which it changes from a glass to the rubbery form
- **Molding Cycle** - The period of time occupied by the complete sequence of operations on a molding press requisite for the production of one set of molded articles.
- **Nitriding**- is a heat treating process that diffuses nitrogen into the surface of a metal to create a case hardened surface. It is predominantly used on steels, titanium and aluminum.
- **non-conformance** - a condition of product in which any characteristics do not conform to specifications required and/or stated. This may include failures, deficiencies, defects, malfunctions.
- **Parting Line –**It's the line that shows where the two mold halves met when they were closed.
- **Pellets** - Resins or mixtures for resins with compounding additives in the shape of similar sized tables and granules that have been extruded or chopped into short segments to prepare them for molding operations.
- **Polishing** - are finishing processes for smoothing a workpiece's surface using an abrasive and a work wheel

- **Sprue** - The feed opening provided in injection molding between the nozzle and cavity or runner system.
- **Sprue Bushing** - A hardened-steel insert in the mold that accepts the plasticator nozzle and provides an opening for transferring the melt.
- **Sprue gate** - A passageway through which melts flows from the nozzle to the mold cavity.
- **USB (Universal Serial Bus)** - a connection technology for attaching peripheral devices to a computer, providing fast data exchange.
- **Warpage –** it is a form of distortion that can occur in some materials and can be defined as a dimensional distortion in a molded product after it is ejected from the mold at the end of the injection molding process.