Design Enhancement and Optimization of Plastic Injection Moulding

D.Venkatesa Prabu[1] A.Sathishkumar[2] R.Nagaraj[3] R.Pugazhenthi[4] N.Rangeesh[5]
[1,2]Assistant Professor [3,4,5]UGScholar
[1,2,3,4,5]Department of Mechatronics Engineering
[1,2,3,4,5]Mahendra College of Engineering, Salem, Tamilnadu, India.

Abstract— This project focus on plastic injection moulding process is a near net shape manufacturing method allowing the production of small to moderate size complex shaped components in design the casting industry, using the plastic and wax material. In this project four process parameters namely, barrel temperature, mould temperature, holding pressure and injection speed were considered. The casting industry has traditionally been regarded heavily craft based in various stages and jewellery manufacturing. Jewellery design and model making are time consuming tasks. The enhancement and jewellery making in less time and using less number of components to improve die design is presented parts conduct mold flow analysis to determine the feasibility. The die gating system and the pressure-assisted injection moulding process are optimized. Reliability and predictability of the jewellery making technology, after moulding the binder must be removed from moulded specimens. Various debinding techniques have been developed for this purpose the main techniques can be classified into thermal and solvent debindings, therefore the main focus of this contribution has been laid on jewellery making like Gold, Silver, Platinum, copper in injection moulding. The optimized design scheme of feed system is determined in view of the particular structure that can be analyses through plastic part, and then the simulation flow analysis of experimental system in injection molding is carried execution. Injection molding parts conduct mold flow analysis to determine the feasibility, and the die gating system used by gas-assisted injection molding process are optimized. Thus, high projection quality products are obtained. Practice has proved that CAE technology for plastic mould design and plastic moulding has a strong guiding role.

Key words: component; formatting; style; styling; insert.

I. Introduction

Injection molding is the most commonly used manufacturing process for fabrication of plastic parts. A wide variety of products are manufactured using injection molding, which vary greatly in the size, complexity, and application. The injection molding process requires the use of an injection molding machine, raw plastic material, and a mold. The plastic is melted in the injection molding machine and then injected into the mold, where it cools and solidify into the final part. The steps in this process are described in greater detail in the next section Injection molding and overview Injection molding is used to produce thin-walled plastic parts for a wide variety of applications, one of the most common being plastic housings. Plastic is a thin-walled enclosure, often requiring many ribs and bosses in the interior. These housings are used in a variety of products including household appliances, consumer electronics, power tools and automotive dashboards. Other common thin-walled products include different types of open containers, such as buckets. Injection molding is also used to produce several items such as toothbrushes and small plastic toys. Many medical devices, including valves and syringes, are manufactured using injection molding as well.

A. *Heating Coil:*

A heating element converts electricity into heat through the process of resistive of Joule heating. Electric current passing through the element encounters resistance and resulting in heating of the element. Unlike the Peltier effect of this process is independent of the direction of current flow figure 1.1.

Fig. 1.1 Heating coil

B. *Thermocouple:*

A thermocouple is an electrical device consisting of different conductors forming electrical junctions. A thermocouple produces a temperature dependent voltage as a result of thermoelectric effect and this voltage can be interpreted to measure temperature. Thermocouple is a widely used type of temperature sensor. Commercial thermocouple is inexpensive, interchangeable and they are supplied with standard connectors and they measure a wide range of temperatures. In contrast to other methods of temperature measurement and thermocouples are self powered and require no external form from the excitation. The main limitation of thermocouples contains accuracy. System errors of less than one degree Celsius (°C) have been difficult to achieve. They are widely used in science and industry. These applications include temperature measurement for kilns, gas turbine exhaust, diesel engines and other industrial processes. Thermocouples are also used in homes, office and business as the temperature sensors in thermostats and also as flame sensors in safety devices shown in Figure 1.2.

Fig. 1.2: Thermocouple

Product Type	Temperature control
Control input (thermocouple)	Yes
Control output 1	Relay: SPDT, 16 A at 250 VAC
Display	3-digit LED, red, 1/2" H
Power (VAC)	230

Table 1: Thermocouple Specification

C. Pressure Relief Valve:

The pressure relief valve is a valve used to control the pressure in a system or vessel which can build up for a process or equipment failure. The pressure can be relieved by allowing the pressurized fluid to flow an auxiliary passage out of the system. The relief valve is designed or set to open at a predetermined set pressure to protect pressure vessels and other equipment that exceed their design limits. When the set pressure is exceeded to the relief valve becomes "path of least resistance" from the valve is forced open and portion of the fluid is diverted through auxiliary route. As the fluid can be diverted means the pressure inside the vessel will stop rising. Once it reaches to the valve's reseating pressure then the valve will close. The blow down can be usually stated as a percentage from set pressure and refers to the pressure needs to drop before the valve resets. The blow down can vary from 2–20% and these valves have adjustable blow downs. In high pressure gas it is recommended that the outlet of the relief valve is in open air. In systems the outlet is connected to piping the opening of a relief valve will give a pressure build up in the piping system of the relief valve. This often the relief valve will not re seat once the set pressure is reached. For these system often so called differential relief valves are used. This means that pressure is only working on the area that is smaller than the opening area of the valve. If the valve is opened and the pressure has to decrease before the valve closes and also outlet pressure of the valve can be easily keep the valve open. Another consideration of other relief valves are connected to the outlet system that they may open as the pressure in exhaust of the auxiliary route. The diverted fluid is usually routed through a piping system or relief header to a central or elevated gas 5 flare where it is usually burned and the resulting combustion of gases are released to the atmosphere in fig 1.3.

Fig. 1.3: Pressure relief valve

D. Aluminium Die:

A die was specialized tool used in manufacturing industries to cut or shape material mostly using a press. Like molds, dies are generally customized to the item they are used to rated. The die is a metal block that is used for forming materials. For the vacuum forming of plastic sheet only a single form is used, typically to form transparent plastic container for merchandise. For the forming of sheet metals such as automobile body parts two parts may be used one, called the punch, another stretching, bending and blanking operation, while another part called the die block, securely clamps the work piece and provides similar stretching, bending and blanking operation. The work piece pass through several stages using different tools or operations to obtain the final form. In the case of an automotive component there will usually be shearing operation after the main forming is done and then additional crimping operations to ensure that all sharp edges are hidden and to add rigidity to the panel in fig 1.4.

Fig. 1.4: Aluminum die

E. Die Components:

The main components for die tool sets are:

- Die block - This is main part that all the other parts are attached.
- Punch plate - This part hold and then supports to the different punches in place.
- Blank punch - This part along with blank die and then produces the blanked part.
- Pierce punch - This part along with the pierce die remove parts from the blanked finish part.
- Stripper plate - This can be used to hold the material on the blank die and strip the material off the punches.
- Pilot - This will be helped to place the sheet accurately for the next stage of operation.
- Setting block - This part is used to control the depths of the punch going into the die.
- Blanking dies - See blank punches.
- Pierce die - See pierce punches.
- Shank - used to hold in the presses. It should be aligned and situate at the center of gravity of the plate.

F. Processes:

Blanking: A blanking die produce a flat piece of material by cutting the desire shape in an operation. The finished part is refer to a blank. Generally a blanking die only cut the outside contour of a part, often used for parts with no internal features.

Accuracy: A properly sharpen die with the correct amount of clearance between punch and die will produce a part that hold close to the dimensional of the tolerance relationship to the part edges.

Appearance: Since the part is blank in one operation, the finish edge of the part produce a uniform appearance may be opposed to varies degrees of burnish from multiple operations.

Flatness: Due to the even compression of the blanking process to the end results is a flat part that can retain a specific level of flatness for additional manufacturing operations.

Broaching: This process of removing of material through use of multiple cutting teeth, with each tooth cutting behind the other. A broaching die is often used to remove material from parts are too thick for shaving.

Bulging: A bulging die expand the closed end of tube through use to two types of bulging die. Similar to the way a chef hat bulges out at the top from the cylindrical band around the chef head.

Bulging fluid dies: Use of water or oil, vehicle to expand the part.

Bulging rubber dies: Uses a rubber pad or block under pressure to move the wall of work piece.

Coining: A similar to forming with the main difference between coining die may form completely different features on either face of the blank, these feature be transferred from the face of the punch or die respectively. The coining die and punch flow the metal by squeezing the blank with a confined area, instead of bend the blank. For example: an Olympic medal that was formed by from of coining die may have a flat surface on the back and a raised feature on the front. If the medal was formed (or embossed), surface on the back would be a reverse image of the front.

Compound operations: Compound die are perform multiple operation on the part. The compound operation is act like implementing more than one operation during the press cycle.

Compound die: A type of die that has been die block mounted on a punch plate with perforator in the upper die with the inner punch mounted in the lower die. An inverted type of blank die that punch upwards, leaving the part sitting on the lower punch instead the blanking of part through. A compound die allows the cutting of internal and external part feature on a single press stroke.

Curling: The curling operation is used to roll the material into a curve shape. A door hinge is an example of the part create by a curling die.

Cut off: Cut off dies are used to cut off excess material from a finishing end of a part or to cut off a predetermine length of the material strip for additional operations.

Drawing: The drawing operation is very similar to the forming operation except that the drawing operation undergoing severe plastic deformation and the material of the part extend around the side. A metal cup with a detail of feature at the bottom is an example of the difference between form and drawn. The bottom of cup was formed while sides were drawn.

Extruding: Extruding is the act of severing deform blank of a metal called slugs into finished part such as an aluminum I-beam. Extrusion dies used extremely high pressure from the punch to squeeze the metal into the desire form. The difference between cold forming and extrusion is extruded part do not take the shape of punch.

Forming: Forming dies bend the blank along a curved surface. An example of a part that has been form would be the positive end (+) of a battery.

Cold forming: Cold forming is similar to extruding process that squeezes the blank material but cold forming use the punch and the die to create the desired shape of extruding.

G. Wax Material:

Wax is a diverse class of organic compound that are hydrophobic, malleable solid near ambient temperature. They include higher alkanes and lipid, typically with melting points above about 40 °C (104 °F), melting to give low viscosity liquid. Wax is insoluble in water but soluble in organic, non polar solvents. Natural wax of different types is produced by plants and animals and occurs in petroleum.

H. Abs Material:

Acrylonitrile butadiene styrene (ABS):

ABS is an ideal material whenever superlative surface quality, colorfastness and luster are required. ABS is two phase polymer blend. A continuous phase of styrene-acrylonitrile copolymer (SAN) give the material rigidity, hardness and heat resistance. The toughness of ABS is the result of sub microscope fine polybutadiene rubber particles uniformly distributed in the SAN matrix. Chemical formula $(C_8H_8)_x\cdot(C_4H_6)_y\cdot(C_3H_3N)_z$ is a common thermoplastic polymer. It glass transition temperature is approximately 105 °C (221 °F). ABS is amorphous and no true melting point. ABS is a polymer made by polymerize styrene and acrylonitrile in the presence of polybutadiene.

The proportion can vary from 15 to 35% acrylonitrile, 5 to 30% butadiene and 40 to 60% styrene. The result is a long chain polybutadiene cross with shorter then chain of poly(styrene-co-acrylonitrile). The nitrile group from neighboring chain, being polar, attract each other and binding the chains together, making ABS stronger than pure polystyrene. The styrene gives the plastic a shiny, impervious surface.

The polybutadiene, a rubbery substance, provide toughness even at low temperature. For the majority of application, ABS can be used between −20 and 80 °C (−4 and 176 °F) as its mechanical properties vary by temperature. The properties are create by rubber toughening, where fine particles of elastomer are distributed throughout the rigid matrix.

II. LITERATURE REVIEW

A. Optimization of Plastic Injection Moulding:

Micro injection moulding process parameter tuning, (Michael Packianather et al 2 July 2015) [1]:

Design of experiment using Taguchi L16 Orthogonal Array has been conduct with four process parameters name, mould temperature, barrel temperature, holding pressure, and injection speed in order to understand the effect of these parameter during the micro injection moulding process.

Using the L16 Orthogonal Array experiment have been carry out to produce test parts and obtain the de moulding force would be measured response. Each trial was repeated ten times and average value were used in the analysis of the results. The analysis was carried out using Minitab16. Two criteria for S/N were used The results obtained showed that for nominal-the best an optimum setting level of process parameters were A2B2C2D2 and for bigger the better the optimum setting level of process parameters were A2B2C2D1.

The study also showed some interaction between parameter A and C, A and D, and B and C. This study has shown that the most significant parameter is mould temperature and holding pressure. Different polymers will be used in the future to studies the effects of these process parameters upon different materials.

Injection Mold Design and Optimization of Automotive panel, (Zhenyu ZHAO et al 23 July 2010) [2]:

By analyze the result can be seen, an optimized structure and injection mold parameter can be relatively satisfied with the injection molded parts. According to the results, Automobile Panel mold design would be appropriate to reduce the packing pressure and shorten the injection time, and the cooling water setting should be to avoid the red part of the final filling. According to the result analysis of weld line, the mandrel and air exhaust should be set the part that are easier to produce the weld line or weld lines produce more. Mold Flow software can optimize mold structure, which can simulate the entire production process, and found that mold or forming in advance the quality of existing problems, and thus target to modify the design to regulate the process parameter, to avoid repeated in the actual production of test mold, maintenance mold. This will not only shorten the development cycle of mold, but also saves manpower, material resource, reduce production cost, improve product quality.

The effects of injection moulding temperature on the mechanical properties and morphology of polypropylene man-made cellulose fiber composites, (M.Feldmann et al 23 April 2016) [3]:

This investigation evaluate by influence of the melt temperature during the injection moulding process of PP with 30 wt% manmade cellulose fiber on the morphology and the mechanical property of the material. In addition to the composite properties, thermal stability of the matrix material and reinforcement fiber is also determined using TGA and tensile test. The investigated man-made cellulose fibres showed a stronger increase of mass loss starting at 260 _C between 3 min and 12 min because of chemical structure changes, It was found out that the fiber length distribution in the composite changed significantly at melt temperature above 256 C. It result in a smaller distribution of fiber with a lower mean values. This was due to mechanical stress during processing and lower fiber strength caused by thermal degradation. Both led to a decrease in tensile strength, and, especially, the elongation at break and the notch Charpy impact strength. Moreover, less and shorter fiber pull-outs with fanned fiber end were observed at T-5 (269 _C). The discolouration (DE) and the L-value of the specimen show a linear tendency to darker colours in the range of investigate melting temperature, darken more would the temperature rose. No direct dependency between discoloration and mechanical property is observe by Polypropylene containing 30 wt% of man-made cellulose fibres, which was compound using a single-step pultrusion method, display a high notched Charpy strength (36.5 kJ/m2) and elongation at break (7.4%). Melt temperatures up to 250 _C can be used when injection moulding this composite without strongly effecting the mechanical property. Yet they will still have a slight effect on the colour. An overview of the property change depending on the melting temperature during injection moulding. The results also illustrate the potential usage and effectiveness of these bio-based fiber as reinforcement in matrix materials that possess a higher melting temperature, such as polyamides, or other engineering polymer.

B. Summary and Motivation:

Injection molding is the most commonly use to manufacturing process for the fabrication of plastic part. A wide variety of product are manufactured using injection molding is vary greatly in their size, complexity, and application. The injection molding process require to use of an injection molding machine, raw plastic material, and a mold. This is related to the nature of the injection moulding process which result in higher material compaction in addition to the enhanced of crystalline structure, thus enhancing mechanical strength. However, this study show that the adequate selection of FDM parameter can yield part of acceptable properties. Here, primarily the raster gap is a crucial parameter where a negative gap of−0.05 mm, the material become denser almost reaching density of injected part. That high packs of the materials make the effect of change raster angle insignificant for static test. Thus, a proper selection of the raster angle with respect to the load condition become especially important when printing at positive raster gap. Here, it is to be noted that raster lying parallel to the loading direction will be able to carry all load in contrast to filament intransverse direction that insignificantly contribute to load bearing. In contrast, however, dynamic behavior is significantly affect by the raster angle regardless of the raster gap direction. The plastic is melt in the injection molding machine and then inject into the mold, where it cool and solidifies into the final part. Injection molding is use to produce thin-walled plastic parts for a wide variety of applications, one of the most common plastic housings. Plastic housing is a thin-wall enclosure, often requiring many rib and bosses on the interior. These housing are used in a variety of products including household appliances, consumer electronic, power tool, and as automotive dashboard. Other common thin-walled product include different types of open container, such as buckets. Injection molding is also used to produce several everyday items such as toothbrushes or small plastic toys. Many medical device, including valve and syringe, are manufactured using injection molding as well.

III. DESIGN CALCULATION OF INJECTION MOULDING

1) Select by injection volume:

A guide, generally the injection machine should be selected that molded product volume will become 30 % to 80% of the machine injection volume. When molding, the relation of the machine injection volume Q(g) and one shot weight W(g) should be in the range indicated below.

$$Q = (1.3 \sim 1.5) \times W$$

If the injection volume is too small, plasticization will not be make it, and might lose it original physicality as a molded product because the resin will be sent without enough plasticization. On other hand, if the injection volume is too big, residence time inside the cylinder will be longer then and cause degradation by more chance.

2) Select By Mold Clamping Pressure:

Both toggle type and direct pressure type is suitable when molding. The relation of mold product project area A (cm2) and require to mold clamp pressure P(ton) should be in the range indicate below.

$$P = (0.5 \sim 0.7) \times A$$

3) Nozzle structure:

Open nozzle is common when mold NOVADURAN. The nozzle of commercially-supply to injection machine can be have a temperature control. If drool from the nozzle is concern with use the shut-off nozzle. However, it might cause burn open nozzle or shut-off nozzle but in any types, it is necessary to and sunspot object by resin retention at the slide part, so be careful.

4) Injection mechanism:

Injection mechanism mold by the basic injection machine which , the function of constant injection speed and two-stage injection pressure control, but when the molding product which severe measurement, appearance, and moldability liquidity and demoldability is require, it is effective to use the machine that has a program control of injection speed and injection pressure.

5) Backflow prevention ring:

Backflow prevent the ring is necessary at the screw, because it was relatively low melt viscosity. If this backflow prevention ring is damage by wear or corrode, cushion volume cannot be kept because of the resin backflow from the cylinder to the hopper when injecting pressure keep, and injection pressure hold pressure might not be put proper cavity. In this case, good mold product cannot be made, so cushion volume and its stability must be controlled and maintain when molding. Corrosion and abrasion resistance steel grade is preferable for back flow prevention ring.

6) Drying machine:

Preliminary dry is necessary before molding, and the condition below is general.

120℃ 5～8 hours

130℃ 4～6 hours

Dehumidification dryer is preferred when drying. To prevent the dust and the dirt get inside, a filter should be place to air intake of the drying machine, and its maintenance against clogging is also necessary Shelf-type hot air circulation dryer, hopper dryer.

B. Moulding Condition:

1) Resin temperature:

When mold resin temperature should be generally about 240℃～265℃. Liquidity will be better than the temperature rise, but extremely high temperature will accelerate heat degradation which end up with physicality deterioration of the molded article.

2) Injection and pressure keeping:

Pressure: Injection pressure can be consider were the fill pressure primary pressure and the hold pressure secondary pressure. Generally the fill pressure will be set stronger than the hold pressure. When low-temperature solidification, crystalline resin will be the big shrink, therefore the hold pressure is necessary for filling up and is closely related to the mold shrinkage. Increasing the hold pressure resolve sink and void problem, but if increase too much, it might cause burr, so the attention was required.

Injection speed: In the case of thin mold product or multi-cavity mold product which severe size precision was required, faster injection speed is better. In contrast, slower injection speed is better for thick mold product. Also, the program control of injection speed is effective to resolve to jet and the flow mark.

Injection time: Setting is differ by the mold machine, but basically should be consider below. Injection time filling time + pressure keeping time gate seal time Gate sealing time is the time when resin stop flowing by solidification at the gate part. If pressure keep is put away before the gate is seal, molten resin will backflow from the gate by the tool internal pressure, which will cause measure and physicality variability, and warpage, sink, and void problem, because of decrease in mold products filling density packing property. To estimate the gate sealing time, measure weight of molded product by gradually increasing the injection time, and look for the injection time when the weight of molded became a certain amount and stop changing.

C. Back Pressure:

The measurement might become instable by the gases and the air generate from molten resin when plasticization. To stabilize the measurement and improve the knead effect. However, if the back pressure is too strong, it might degrade the plasticization ability.

Screw Back Pressure （5 ～10kg/cm2)

1) Mold temperature:

Generally, 60℃ to 80℃ is suitable for mold temperature, and this is the most important point in the mold condition. If high cycle mold is intend, molding in temperature of about 20℃ to 30℃ is possible by using chiller temperature control, but require attention because it might cause deformation. The residual strain inside the mold product, and dimension change by after contraction might be big and depend on the usage environment high temperature atmosphere. It is effective to raise the mold temperature to about 120℃, when dimension stability is require to assumed to use under a high temperature atmosphere, or high level of surface gloss is necessary.

2) Preliminary drying:

A resin with that has relative low water absorption rate, but if it receive heat history when it is absorb the water, even a small amount of moisture will cause hydrolysis reaction, and by that molecular weight will decrease which will end up with degrade in physical. Therefore, moisture must be remove well enough before mold. Preliminary drying under condition below is necessary before using. Temperature of hot air 120℃ 5～8hours or 140℃ 4～6hours

3) Retention heat stability:

If high temperature when mold, molecular weight might decrease because of the cut of molecular chain by heat deterioration indicate relation between residence time and physicality. We know that deterioration will be fast the cylinder temperature rise. Furthermore, deterioration will accelerate the moisture rate inside pellet rise, so need special attention. Shortening residence time was especially necessary if adding reprocessed material. Effect to the residence time is up to additive amount and heat history, and in the case of flame resisting grade, residence time should be set even short.

D. Mould Designing:

1) Point of mould designing:

Design of mould product should be done to fulfill demand characteristic of desire product, and need to evaluate material practical physicality, moldability, liquidity, and mold design condition, comprehensively. Basic point of mold design are indicate below. Try not to make the thickness excessively thick, and try to keep it even, so rapid change in thickness will not occur. If the mould product thickness is too thick, it will be the cause of defected sink mark and void. Also, it will take time to cool down and the mold cycle will be longer. When there is need to be thick for function, try to keep it even by place the recess. If there is unevenness or rapid change in thickness, flow mark might occur because it block the resin flow, or warpage might occur by uneven mold shrinkage ratio, or deformation of moulded product might occur by uneven cooling speed.

E. Pressure Loss:

Pressure loss will occur on the sprue, runner, and gate part so be careful. Presum the resin is Newtonian fluid, the amount of pressure loss will be shown the follow, and from that, we know it depend largely on cross-section thickness and diameter.

F. Properties of Plastics:

Following are some of the important properties of plastics.

1) Density:

Density of a material is the measure of the mass per unit volume and was generally express as gram per centimeter cube. Example: Density of General Purpose Polystyrene (GPPS) = 1.06 gms/cc

2) Specific Gravity:

Specific Gravity is ratio of the density in a material. Density of water = 1 gm/cc. The specific gravity of GPPS = (1.06/1) = 1.06 It is a ratio is therefore no units.

Specific gravity is the ratio of the density of a substance to the density of a reference substance equivalent, it is the ratio of the mass of a substance to the mass of a reference substance for the same given volume. Apparent specific gravity ratio of the weight of a volume substance in weight of an equal volume of the reference substance. The reference substance is near away water at its densest (4°C) for liquid; for gases at air in room temperature (21°C). Nonetheless, the temperature and pressure must be specify for both the sample and the reference. Pressure is nearly away Temperature for both sample and reference vary from industry to industry.

3) Water Absorption:

Water Absorption is the percentage increase the weight of a material due to absorption of water. Most plastic are hygroscopic (water absorbing). They are absorb water and form a chemical bond with it. Example - Nylon, Polyester. Figure 3.7.1

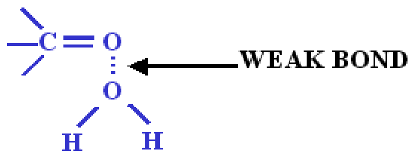


Fig. 3.6.1: Water Absorption

This should not be confuse with Water Adsorption. Adsorption is attraction of water molecule on the surface of the plastic. The phenomenon is similar to water condensing on cold soda can. Example: Polyethylene fed into a hopper from cold silo outside the molding facility

4) Shrinkage:

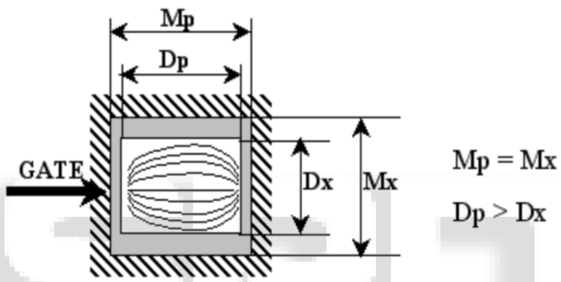


Fig. 3.6.2: Shrinkage

Shrinkage is ratio of the volume of the plastic in the melt to the volume of the plastic in its final use state. When apply to molding, it is expressed were the ratio of the dimension of the mold to the dimension of the part that is mold. Shrinkage is express in inch per inch or in percentage fig 3.6.2. In case of amorphous materials, shrinkage the same in the cross flow and the parallel flow direction. Such material are called isotropic material.

5) Melt Flow Rate or Melt Flow Index:

Melt Flow rate (MFR) or Melt Flow Index (MFI) is the amount of material extrude in 10 minutes under certain testing condition. The plastic is fed into a heated vertical barrel and a known weight is kept on top of the plunge. The amount of plastic that is extrude in 10 minutes is called the MFR or MFI shown in figure 3.6.3.

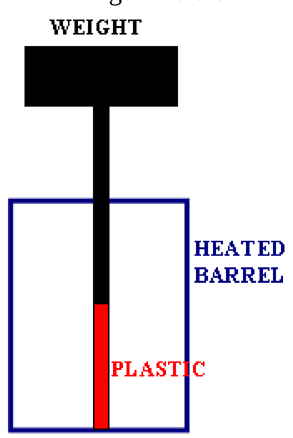


Fig. 3.6.3: Melt Flow Rate or Melt Flow Index

6) Moulding Shrinkage Ratio:

Molding shrinkage will occur in the process of cool solidification of molten resin fill at the cavity, so relatively big molding shrinkage will occur in the case of crystalline resin like PBT resin. Mold shrinkage ratio depend on mutual effect of many factor, and the major factor will be the followings.

1) Resin temperature
2) Mold temperature
3) Injection pressure
4) Injection speed
5) Injection time
6) Molded product thickness
7) Filling material, shape of the reinforcing material

A good estimate of the solid flow rate function of the convey efficiency and the feed depth. The desire output can be found by simulating the effect of these factor on the flow. The solid transport is largely influenced by the frictional force between the solid polymer and barrel and screw surface. A detail analysis of the solid conveying mechanism. This is relate to the nature of the injection moulding process which result in higher material compaction in addition to the enhancing of crystalline structure, thus enhancing mechanical strength. Here, primarily the raster gap is a crucial parameter where at a negative gap of−0.05 mm, the material become denser almost reaching density of injected part. This high packing of the material make the effect of changing raster angle insignificant for static test.

IV. PHOTOGRAPHY OF PROJECT

Fig. 3.7:

Force	107.42×10^3 N
Internal Diameter	127 mm
External Diameter	228.6 mm
Internal cylinder Area	12667.68 mm^2
External cylinder Area	41043.30 mm^2
Pressure	20 bar
Temperature	300˚C

Table 2: SPECIFICATION

Levels	A: mold temperature (˚C)	B: melt temperature (˚C)	C: packing pressure (Mpa)	D: packing time(s)
1	60	274	20	10
2	73.5	287	25	15
3	87	300	20	20

Table 3: Parameter values

S.NO	ABS(N)	ABS+PC(N)	PC(N)	Rein. ABS(N)	Rein. ABS+PC(N)
Point 1	389.5	622.7	743.3	908.0	1058.4
Point 2	436.0	696.7	831.7	1016.0	1185.0
Point 3	495.0	791.0	944.3	1154.0	1345.0
Point 4	543.0	883.2	1062.6	1233.0	1527.0

Table 4: Selection of composite raw material properties

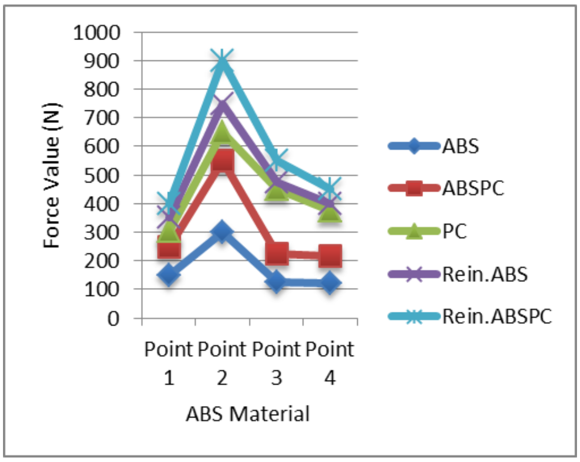
Name	ABS	ABS +PC	PC	Reinforced ABS	Reinforced ABS+PC
Material structure	Amorf	Amorf	Amorf	Amorf	Amorf
Elastic modulus (Mpa)	2240	2780	2280	8489	7944.82
Poisson ratio	0.392	0.41	0.417	0.35	0.4238
shear modulus(Mpa)	805	986	805	1891	4041.05
Specific heat(J/kg ˚C)	2100	1990	1900	1877	1965.8
Conductivity(W/m ˚C)	0.24	0.28	0.24	0.333	0.363
Melt density (g/cm^3)	0.93305	0.96861	1.0433	1.1668	1.1029
Solid density (g/cm^3)	1.0541	1.1161	1.1916	1.2816	1.257

Table 5: Physical properties of materials

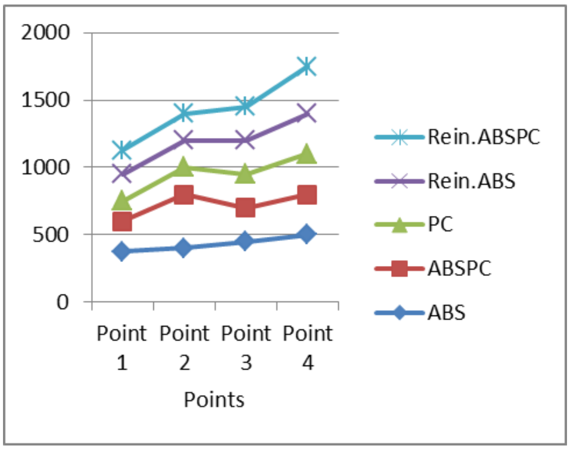
Name	ABS	ABS+PC	PC	Rein. ABS	Rein. ABS+PC
Tensile yield strengths(Mpa)	38.0	60.7	72.4	89.0	103.0

Table 5: Tensile yield strength of the materials

A. Graph for force value to AB:



B. Graph for Force Value to Resin:



V. RESULT

This investigation evaluate by the influence of the melt temperature during the injection moulding process of PP with 30 wt.% manmade cellulose fibres on the morphology and the mechanical property of the materials. In addition to the composite properties, the thermal stability of the matrix material and the reinforcement fibres was also determine to use TGA and tensile test. The investigate man-made cellulose fibres show a stronger increase of mass loss starting at 260 _C between 3 min and 12 min because of chemical structure change (dehydration). These temperature and time did not affect the polypropylene matrix significantly. Also, the tensile strength decrease after the material were subject to exposure in these conditions. It was found out that the fiber length distribution was the composite (injection moulded specimen) changed significantly at melt temperatures above 256˚C. It result is a small distribution of fiber with a lower mean value. This was due to the mechanical stress during processing and the lower fiber strength caused by thermal degradation. Both led to a decrease in the tensile strength, and, especially, the elongation at break and the notch Charpy impact strength. Moreover, less and shorter fiber pull-outs with fanned fiber end were observed at T-5 (269˚C).

Db	30 mm
Screw lead	30 mm
Number of flights ν	1
Flight width *w*FLT	3 mm
Channel width *W*	28.6 mm
Depth of the feed zone *H*	5 mm
Conveying efficiency hF	0.436
Bulk density of the polymer ro	800 kg/m3

Table 6: Barrel diameter

The discoloration (DE) and the L-value of the specimen showed a linear tendency to darker colour in the range of the Investigate melt temperature and darkening more were the temperature rose. No direct dependency between discoloration and mechanical properties was observed.

VI. CONCLUSION

Injection moulding is widely used for industrial sector, which means complicate and highly accuratable jewellery parts manufacturing process. The raw materials of the injection moulding process, which is heat and solidify were a liquid state to flow the melted raw material through zig-zag nozzle motion with spring action operation. The high pressurized wax can be injecting in the rubber die to obtain. In this process we can use the rubber die to obtain a desire design by using mould cavity of Wax, plastic and ABS material. After that mould can be create as the design rubber die formation.

The mould can be attached with the plastic tree to manufacturing the wax duplicate component to change over the original products through copper, silver and gold arranaments.

VII. SCOPE OF FUTURE WORK

In this project mainly contribution to Investment casting is a precision casting method which growing steadily in international significance for reason of economy. The method is to be used to produce larger and larger castings. Easily adapt Investment castings is also employ to an increasing scale for so-called super alloys which require ever more complex melting processes.
The investment casting technique is characterized by,

- Almost unlimited scope for the design shaping of investment castings is have to developed for future enhancement.
- Hardly any restrictions in term of Raw material like ABS, Plastic and Wax formation.
- A high degree of dimensional accuracy due to elimination of the mould parting line and responsible for casting imprecision.
- Facility for complex shape inner contours due to the use of ceramic core.
- Low material allowance on surface to be machined.
- A high standard of surface quality is done by jewelry making process in investment casting operation.

REFERENCES

[1] Pilla S. Engineering applications of bioplastics and biocomposites – an overview. In: Handbook of bioplastics and biocomposites engineering applications. John Wiley & Sons, vol no.2011. PP: 1–15.
[2] Faruk O, Bledzki AK, Fink HP, Sain M. Biocomposites reinforced with natural fibers: 2000–2010. Prog Polym Sci, vol no 2012;37(11):PP:1552–96.
[3] Faruk O, Bledzki AK, Fink HP, Sain M. Progress report on natural fiber reinforced composites. Macromol Mater Eng, vol no 2014;299(1):PP:9–26.
[4] Furtado SC, Araújo AL, Silva A, Alves C, Ribeiro AMR. Natural fiber-reinforced composite parts for automotive applications. Int J Automot Compos, vol no 2014;1(1):PP:18–38.
[5] Kamrani AK, Abouel Nasr E. Engineering Design and Rapid Prototyping. New York, US: Springer; 2010. Sood AK, Ohdar RK, Mahapatra SS. Parametric appraisal of mechanical property of fused deposition modelling processed parts. Mater Des, vol no 2010;31;PP:287–95,

[6] Pham D, Gault R. A comparison of rapid prototyping technologies. Int J Mach Tools Manuf 1998;38:1257–87, http://dx.doi.org/10.1016/S0890- 6955(97)00137
[7] Kumar S, Kruth JP. Composites by rapid prototyping technology. Mater Des vol no 2010;31:PP:850.
[8] Reprap (accessed September 5, 2014). Rodríguez JF, Thomas JP, Renaud JE. Mechanical behavior of acrylonitrile butadiene styrene fused deposition materials modeling. Rapid Prototype J, vol no 2003;9:PP:219–30
[9] Sood AK, Ohdar RK, Mahapatra SS. Experimental investigation and empirical modelling of FDM process for compressive strength improvement. J Adv Res vol no 2012;3:PP:81–90