Experimental Investigation and Mould Design For Injection Moulding

V.H.Shinde[1], R.D.Chavan[2], S.J.Bitake[3], S.I.Pathan[4], L.B.Thorat [5]
[1,2,3,4,5]D.Y.Patil college of engineering,Talegaon,Pune

Abstract*-The injection moulding die design is a key task involving several critical decisions with direct implications to yield quality, productivity and frugality. Injection molding is the most widely used polymeric fabrication process. It evolved from metal die casting, however, unlike molten metal; polymer melts have a high viscosity and cannot simply be poured into a mold. Instead a large force must be used to inject the polymer into the hollow mold cavity. More melt must also be packed into the mold during solidification to avoid shrinkage in the mold. The injection molding process is primarily a sequential operation that results in the transformation of plastic pellets into a molded part. Identical parts are produced through a cyclic process involving the melting of a pellet or powder resin followed by the injection of the polymer melt into the hollow mold cavity under high pressure. The selection of manufacturing conditions is one of the most important aspects to take into consideration in the majority of manufacturing processes, particularly Electrical Discharge Machining (EDM) process. EN31 is high carbon alloy steel and is usually supplied in a hardened and tempered condition. Injection moulding is process of manufacturing plastics products from both thermo and thermosetting plastic materials. This experimental investigation shows the part modeling, design of core-cavity and side core by using SOLID WORKS 2012, the mould flow analysis is carried out by MOLD FLOW ADVISOR 2105 and static structural analysis performed using ANSYS V14.5. The mould tool is of single cavity mould and material planned for producing the component is Acrylonitrile Butadiene Styrene (ABS).*

I. INTRODUCTION

One of the most common methods of converting plastics from the raw material form to an article of use is the process of injection moulding. This process is most typically used for thermoplastic materials which may be successively melted, reshaped and cooled. Injection moulded components are a feature of almost every functional manufactured article in the modern world, from automotive products through to food packaging. This versatile process allows us to produce high quality, simple or complex components on a fully automated basis at high speed with materials that have changed the face of manufacturing technology over the last 50 years or so. The plastic injection molding is one of the most important methods in polymer process industry. In this method, the plastic products having multi-gated molds and obstacles can be produced for industry. Weld lines can occur in the plastic product when two flow fronts meet due to either multi-gated molds or obstacles. The weld line influences the visual and mechanical properties of the products. The weld lines decrease the mechanical properties of injection molded products. This weakness can be minimized through optimization of the injection parameters and design conditions. If it is impossible to change the locations of weld lines by adjusting the locations of gates, the visibility of weld lines can be alleviated by adjusting processing conditions.

Designing plastic parts is a complex task involving many factors that address a list of requirements of the application. "How is the part to be used?" "How does it fit to other parts in the assembly?" "What loads will it experience in use?" In addition to functional and structural issues, processing issues play a large role in the design of an injection molded plastic part. How the molten plastic enters, fills, and cools within the cavity to form the part largely drives what form the features in that part must take. Adhering to some basic rules of injection molded part design will result in a part that, in addition to being easier to manufacture and assemble, will typically be much stronger in service. Dividing a part into basic groups will help you to build your part in a logical manner while minimizing molding problems. As a part is developed, always keep in mind how the part is molded and what you can do to minimize stress.

II. TYPES OF PLASTIC MOULDING

Injection Molding

Injection molding is used for creating high-quality three-dimensional objects, that can be commercially reproduced. The molding process begins by melting plastic in a hopper. Then the plastic is injected into a tightly closed, chilled mold. The plastic quickly takes the shape of the surrounding mold. Once it has completely set, the mold is opened and the plastic object is released. Yogurt pots, butter tubs, toys and bottle caps are made using this process.

Blow Molding

Blow molding is a process used for making piping and milk bottles. Plastic is heated until molten. Then it is injected into a cold mold. The mold has a tube set within it, which has a particular shape when inflated. So, while the plastic is molten, air is blown into the tube and the plastic is formed around the tubing. It is then left to cool and removed from the mold.

Compression Molding

The most labor-intensive type of molding process is compression molding. Therefore, it is only used for large-scale production purposes, and not for mass production. For example, boat hulls and car tires are made using this method. Molten plastic is poured into a mold. Then a second mold is pressed into it. This squeezes the plastic into the desired shape before being left to cool and removed from the mold.

Rotational Molding

Toys, shipping drums, storage tanks and items of consumer furniture are made using rotational molding. Each object is made by coating a mold from the inside. A mold is held in place between two mechanical arms. Then, the arms rotate the mold constantly at the same level, while molten plastic is placed inside. As it turns, the plastic coats the inside of the mold to create a new hollow, plastic object.

The Injection Moulding Procedure

The modern day process has developed and matured significantly to the level where fully automated, closed loop, microprocessor controlled machines are the 'norm', although in principle injection moulding is still a relatively simple process. Thermoplastic injection moulding requires the transfer of the polymeric material in powder or granule form from a feed hopper to a heated barrel. In the barrel, the thermoplastic is melted and then injected into a mould with some form of plunger arrangement. The mould is clamped shut under pressure within a platen arrangement and is held at a temperature well below the thermoplastic melt point. The molten thermoplastic solidifies quickly within the mould, allowing ejection of the component after a pre determined period of cooling time. The basic injection moulding process steps with a reciprocating screw machine are as follows.

Mould Close and Clamping

The mould is closed within the platen arrangement and clamped using necessary force to hold the mould shut during the plastic injection cycle, thus preventing plastic leakage over the face of the mould. Present day moulding machines range from around 15 to 4,000 metric tonnes available clamping force (150 to 4000 kN).Many systems are available for opening/closing and clamping of mould tools, although usually they are of two general types. Direct Hydraulic Lock is a system where the moving machine platen is driven by a hydraulic piston arrangement which also generates the required force to keep the mould shut during the injection operation. Alternatively, smaller auxiliary pistons may be used to carry out the main movement of the platen and a mechanical blocking arrangement is used to transfer locking pressure from a pressure intensifier at the rear of the machine, which moves only by a few millimetres, through to the platen and tool. The second type of general clamping arrangement is referred to as the Toggle Lock. In this case a mechanical toggle device, which is connected to the rear of the moving platen, is actuated by a relatively small hydraulic cylinder, this provides platen movement and also clamping force when the toggle joint is finally locked over rather like a knuckle arrangement.

Injection

At this stage in the machine cycle the helical form injection screw (Figure 1) is in a 'screwed back' position with a charge of molten thermoplastic material in front of the screw tip roughly equivalent to or slightly larger than that amount of molten material required to fill the mould cavity. Injection moulding screws are generally designed with length to diameter ratios in the region of 15:1 to 20:1, and compression ratios from rear to front of around 2 : 1 to 4 : 1 in order to allow for the gradual densification of the thermoplastic material as it melts. A check valve is fitted to the front of the screw such as to let material pass through in front of the screw tip on metering (material dosing), but not allow material to flow back over the screw flights on injection. The screw is contained within a barrel which has a hardened abrasion resistant inner surface.

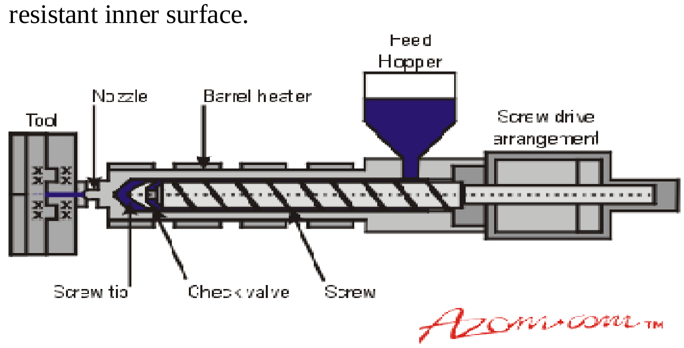


Figure 1. Reciprocating screw injection moulding unit

Normally, ceramic resistance heaters are fitted around the barrel wall, these are used to primarily heat the thermoplastic material in the barrel to the required processing temperature and make up for heat loss through the barrel wall, due to the fact that, during processing most of the heat required for processing is generated through shear imparted by the screw. Thermocouple pockets are machined deep into the barrel wall so as to provide a reasonable indication of melt temperature. Heat input can therefore be closed loop controlled with a Proportional Integral and Derivative (PID) system. The screw (non-rotating) is driven forward under hydraulic pressure to discharge the thermoplastic material out of the injection barrel through the injection nozzle, which forms an interface between barrel and mould, and into the moulding tool itself.

Material Dosing or Metering

During the cooling phase, the barrel is recharged with material for the next moulding cycle. The injection screw rotates and, due to its helical nature, material in granule or powder form is drawn into the rear end of the barrel from a hopper feed. The throat connecting the hopper to the injection barrel is usually water cooled to prevent early melting and subsequent material bridging giving a disruption of feed. The screw rotation speed is usually set in rpm which is measured using a proximity switch at the rear of the screw. Screw rotation may be set as one constant speed throughout metering or as several speed stages.

The material is gradually transferred forward over the screw flights and progressively melted such that when it arrives in front of the screw tip it should be fully molten and homogenised. The molten material transferred in front of the tip progressively pushes the screw back until the required shot size is reached. Increased shear is imparted to the material by restricting the backward movement of the screw, this is done by restricting the flow of hydraulic fluid leaving the injection cylinder. This is referred to as `back pressure' and it helps to homogenise the material and reduce the possibility of unmelted material transferring to the front of the screw.

Mould Open and Part Ejection

When the cooling phase is complete the mould is opened and the moulding is ejected. This is usually carried out with ejector pins in the tool which are coupled via an ejector plate to a hydraulic actuator, or by an air operated ejector valve on the face of the mould tool. The moulding may free fall into a collection box or onto a transfer conveyer, or may be removed by an automatic robot. In this latter case the moulding cycle is fully automatic. In semi-automatic mode, the operator may intervene at this point in the cycle to remove the moulding manually. Once the moulding is clear from the mould tool, the complete moulding cycle can be repeated.

Injection Moulding Machine Selection Criteria

Machine selection, particularly for a range of component types can be quite difficult. It is always wise to talk to machine suppliers in depth regarding overall machine specifications. Rough guidelines do however exist to enable an estimation of machine type and size required.

The Clamping Unit

The clamping unit must be able to supply enough locking force to keep the mould shut during the injection phase, otherwise the mould will part and molten material will flash over the mould split line. As a rough guide of thumb, parts with thin wall sections and deep draw depths require approximately 3-4 tonnes per square inch or 0.5-0.6 tonnes/cm2, and parts with thick wall sections and shallow draw depths require approximately 2 tonnes per square inch or 0.3 tonnes/cm2. To calculate the locking force required for a particular component, this value must be multiplied by the projected area of the component to obtain an overall value in tonnes. The projected area of a component is taken as one side of the moulding only, perpendicular to the injection unit as oriented in the mould. For instance, a simple box housing of 3 mm wall section having a top surface area of 120 cm2 will require at least 120 x 0.3 = 36 tonnes of locking force.

The Injection Unit

The injection unit must be capable of supplying the component shot weight (including the sprue and runner system). The total shot weight should not exceed 90% of the injection capacity of the machine. Injection capacities are usually quoted in grams of polystyrene at a specific gravity of 1.03 g cm-3. If it is intended to process another material, the injection unit shot weight should be recalculated using that particular material's specific gravity. As metering or screw recovery must take place before cooling time has elapsed and the mould opens, the injection unit (size of screw) must be sized to allow this to happen. If recovery does not occur within the cooling period, the overall cycle time will be unnecessarily increased.The maximum temperature possible on the barrel must be great enough to melt the type of plastic being processed. The maximum temperature possible on the barrel must be great enough to melt the type of plastic being processed.The barrel and screw must be specially treated if particularly abrasive materials are to be processed, such as glass fibre filled polyamide (nylon). Also, the screw geometry must be correct for processing specific materials, although general purpose designs are available to cater for a range of commodity thermoplastics.

Polymers Best Suited for Injection Molding

Most polymers may be used, including all thermoplastics and some elastomers. There are tens of thousands of different materials available for injection molding. The available materials mixed with alloys or blends of previously developed materials means that product designers can choose from a vast selection of materials to find the one that has exactly the right properties. Materials are chosen based on the strength and function required for the final part; but also each material has different parameters for molding that must be considered. Common polymers like nylon, polyethylene, and polystyrene are thermoplastic.

III. INJECTION MOLDING EQUIPMENT

Injection Molding Machine:

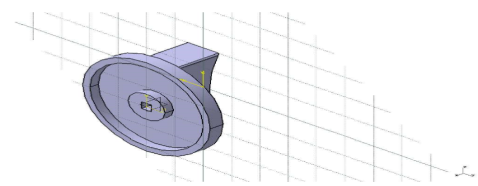
Injection molding machines,* also known as presses, consist of a material hopper, an injection ram or screw-type plunger, and a heating unit. Molds are clamped to the platen of the molding machine, where plastic is injected into the mold through the sprue orifice. Presses are rated by tonnage, which is the calculation of the amount of clamping force that the machine can exert. This force keeps the mold closed during the injection molding process. Tonnage can vary from less than 5 tons to 6,000 tons, although the higher tonnage presses are rarely used. The total clamp force needed is determined by the projected area of the custom part being molded. This projected area is multiplied by a clamp force of from 2 to 8 tons for each square inch of the projected areas. As a rule of thumb, 4 or 5 tons/in can be used for most products. If the plastic material is very stiff, it will require more injection pressure to fill the mold, thus more clamp tonnage is needed to hold the mold closed. The required force can also be determined by the material used and the size of the part with larger plastic parts requiring higher clamping force.

Mold:

The mold or die refers to the tooling used to produce plastic parts in molding. Traditionally injection molds have been expensive to manufacture and were only used in high-volume production applications where thousands of parts were produced. Molds are typically constructed from hardened steel, pre-hardened steel, aluminum, and/or beryllium-copper alloy. The choice of material to build a mold from is primarily one of economics. Steel molds generally cost more to construct but offer a longer lifespan that will offset the higher initial cost over a higher number of parts made before wearing out. Pre-hardened steel molds are less wear resistant and are primarilly used for lower volume requirements or larger components. The hardness of the pre-hardened steel measures typically 38-45 on the Rockwell-C scale. Hardened steel molds are heat treated after machining, making them superior in terms of wear resistance and lifespan. Typical hardness ranges between 50 and 60 Rockwell-C (HRC).

Aluminum molds cost substantially less than steel molds, and when higher grade aluminum such as QC-7 and QC-10 aircraft aluminum is used and machined with modern computerized equipment, they can be economical for molding hundreds of thousands of parts. Aluminum molds also offer quick turnaround and faster cycles because of better heat dissipation. They can also be coated for wear resistance to fiberglass reinforced materials. Beryllium copper is used in areas of the mold which require fast heat removal or areas that see the most shear heat generated.

Introduction Of Gas Stove Switch



IV. METHODOLOGY

a) Manufaturing Steps involved in die manufacturing.

1) Milling:-

Milling is the process of machining flat, curved, or Milling machines are basically classified as vertical or irregular surfaces by feeding the workpiece against rotating horizontal. These machines are also classified as knee-type, cutter containing a number of cutting edges. The milling ram-type, manufacturing or bed type, and planer-type. Most machine consists basically of a motor driven spindle, which milling machines have self-contained electric drive motors, mounts and revolves the milling cutter, and a reciprocating coolant systems, variable spindle speeds, and power-operated adjustable worktable, which mounts and feeds the workpiece. table feeds.

Knee-type milling machines are characterized by a vertically adjustable worktable resting on a saddle which is supported by a knee. The knee is a massive casting that rides vertically on the milling machine column and can be clamped rigidly to the column in a position where the milling head and milling machine spindle are properly adjusted vertically for operation.he plain vertical machines are characterized by a spindle located vertically, parallel to the column face, and mounted in a sliding head that can be fed up and down by hand or power. Modern vertical milling machines are designed so the entire head can also swivel to permit working on angular surfaces, The turret and swivel head assembly is designed for making precision cuts and can be swung 360° on its base. Angular cuts to the horizontal plane may be made with precision by setting the head at any required angle within a 180” arc. The plain horizontal milling machine’s column contains the drive motor and gearing and a fixed position horizontal milling machine spindle. An adjustable overhead arm containing one or more arbor supports projects forward from the top of the column. The arm and arbor supports are used to stabilize long arbors. Supports can be moved along the overhead arm to support the arbor where support is desireddepending on the position of the milling cutter or cutters. The milling machine’s knee rides up or down the column on a rigid track. A heavy, vertical positioning screw beneath past the milling cutter. The milling machine is excellent for forming flat surfaces, cutting dovetails and keyways, forming and fluting milling cutters and reamers, cutting gears, and so forth. Many special operations can be performed with the attachments available for milling machine use.the knee is used for raising and lowering. The saddle rests upon the knee and supports the worktable. The saddle moves in and out on a dovetail to control cross feed of the worktable. The worktable traverses to the right or left upon the saddle for feeding the workpiece past the milling cutter. The table may be manually controlled or power fed.

TURNING

Turning is one of the most common of metal cutting operations. In turning, a workpiece is rotated about its axis as single-point cutting tools are fed into it, shearing away unwanted material and creating the desired part. Turning can occur on both external and internal surfaces to produce an axially-symmetrical contoured part. Parts ranging from pocket watch components to large diameter marine propeller shafts can be turned on a lathe. The capacity of a lathe is expressed in two dimensions. The maximum part diameter, or "swing," and the maximum part length, or "distance between centers." The general-purpose engine lathe is the most basic turning machine tool. As with all lathes, the two basic requirements for turning are a means of holding the work while it rotates and a means of holding cutting tools and moving them to the work. The work may be held on one or by both its ends. Holding the work by one end involves gripping the work in one of several types of chucks or collets. Chucks are mounted on the spindle nose of the lathe, while collets usually seat in the spindle. The spindle is mounted in the lathe's "headstock," which contains the motor and gear train that makes rotation possible. Directly across from the headstock on the lathe is the "tailstock." The tailstock can hold the work by either a live or dead center. Work that is held at both ends is said to be "between centers." Additionally, longer workpieces may have a "steady rest" mounted between the headstock and tailstock to support the work. Typically workpieces are cylindrical, but square and odd shaped stock can also be turned using special chucks or fixtures.

Lathe cutting tools brought to the work may move in one or more directions. Tool movement on the engine lathe is accomplished using a combination of the lathe's "carriage", "cross slide", and "compound rest". The carriage travels along the machine’s bedways, parallel to the workpiece axis. This axis is known as the "Z" axis. Motion perpendicular to the work is called the "X" axis. On an engine lathe this motion is provided by the cross slide mounted on "Straight turning" reduces the work to a specified diameter equally along the work’s axis. "Taper turning" produces a taper along the axis of the workpiece. Tapers are produced by either offsetting the tailstock from centerline or by using a "taper attachment." Some short, steep tapers can be obtained by using the compound restalone. "Contour turning" or "profiling" uses a single-point cutting tool to reproduce a surface contour from a template. This operation has been almost entirely replaced by numerically controlled or "NC" programming. "Forming" uses a cutting tool ground with the form or geometry of the desired shape. This forming tool is advanced perpendicular to the axis of the work to reproduce itsshape on the workpiec

WIRE EDM

The Wire Electric Discharge Machining (WEDM) is a variation of EDM and is commonly known as wire-cut EDM or wire cutting. In this process, a thin metallic wire is fed on-to the workpiece, which is submerged in a tank of dielectric fluid such as de-ionized water. This process can also cut plates as thick as 300mm and is used for making punches, tools and dies from hard metals that are difficult to machine with other methods. The wire, which is constantly fed from aspool, is held between upper and lower diamond guides. The guides are usually CNC-controlled and move in the x–y plane. On most machines, the upper guide can move independently in the z–u–v axis, giving it a flexibility to cut tapered and transitioning shapes (example: square at the bottom andcircle on the top).

The upper guide can control axis movements in x–y–u–v–i–j–k–l–. This helps in programming the wire-cut EDM, for cutting very intricate and delicate shapes. In the wire-cut EDM process, water is commonly used as the dielectric fluid. Filters and de-ionizing units are used for controlling the resistivity and other electrical properties. Wires made of brass are generally preferred. The water helps in flushing away the debris from the cutting zone. The flushing also helps to determine the feed rates to be given for different thickness of the materials. The schematic of wire cut EDM is shown in Figure

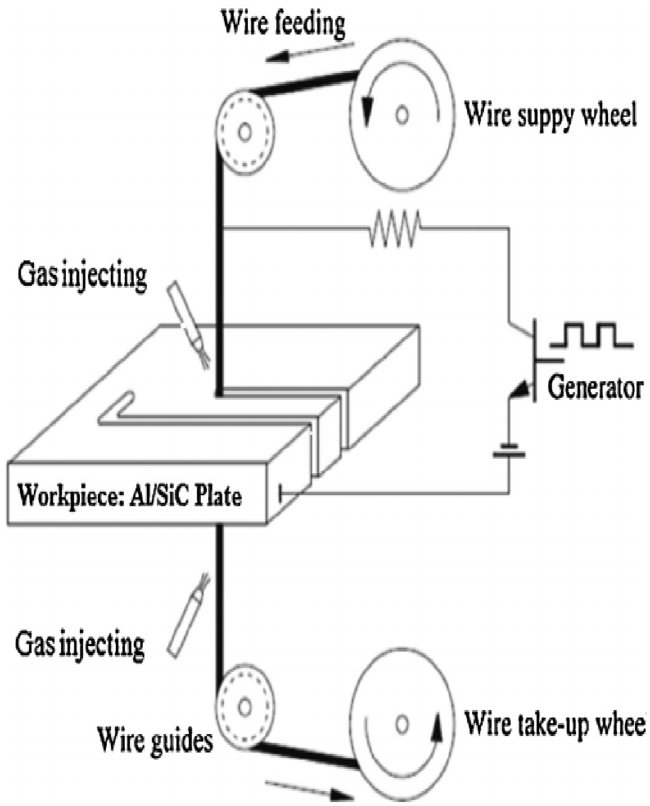


Fig. 1. A schematic diagram of wire EDM process.

Electrical Discharge Machining (EDM).

Is a controlled metal-removal process that is used to remove metal by means of electric spark erosion. In this process an electric spark is used as the cutting tool to cut (erode) the workpiece to produce the finished part to the desired shape. The metal-removal process is performed by applying pulsating (ON/OFF) electrical charge of high-frequency current through the electrode to the workpiece. This removes (erodes) very tiny pieces of metal from the workpiece at a controlled rate. EDM Process,EDM spark erosion is the same as having an electrical short that burns a small hole in a piece of metal it contacts. With the EDM process both the workpiece material and the electrode material must be conductors of electricity. The EDM process can be used in two different ways:

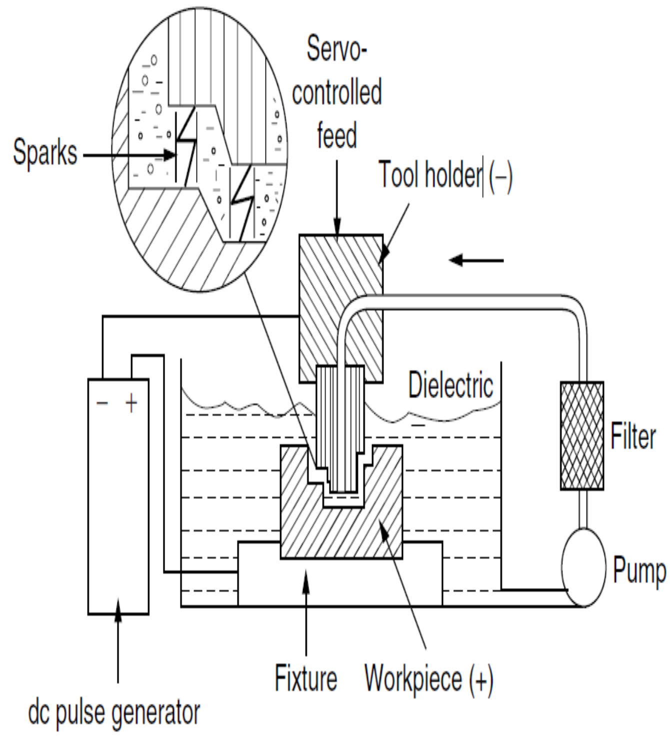


Fig. 1. A schematic diagram of EDM

1. A preshaped or formed electrode (tool),usually made from graphite or copper, is shaped to the form of the cavity it is to reproduce. The formed electrode is fed vertically down and the reverse shape of the electrode is eroded (burned) into the solid workpiece.

2. A continuous-travelling vertical-wire electrode, the diameter of a small needle or less, is controlled by the computer to follow a programmmed path to erode or cut a narrow slot through the workpiece to produce the required shape. Conventional EDM In the EDM process an electric spark is used to cut the workpiece, which takes the shape opposite to that of the cutting tool or electrode. The electrode and the workpiece are both submerged in a dielectric fluid, which is generally light lubricating oil. A servomechanism maintains a space of about the thickness of a human hair between the electrode and the work, preventing them from contacting each other. In EDM ram or sinker machining, a relatively soft graphite or metallic electrode can be used to cut hardened steel, or even carbide. The EDM process produces a cavity slightly larger than the electrode because of the overcut.

DRILLING

Drilling is a process of producing round holes in a solid material or enlarging existing holes with the use of multi-tooth cutting tools called drills or drill bits. Various cutting tools are available for drilling, but the most common is the twist drill.

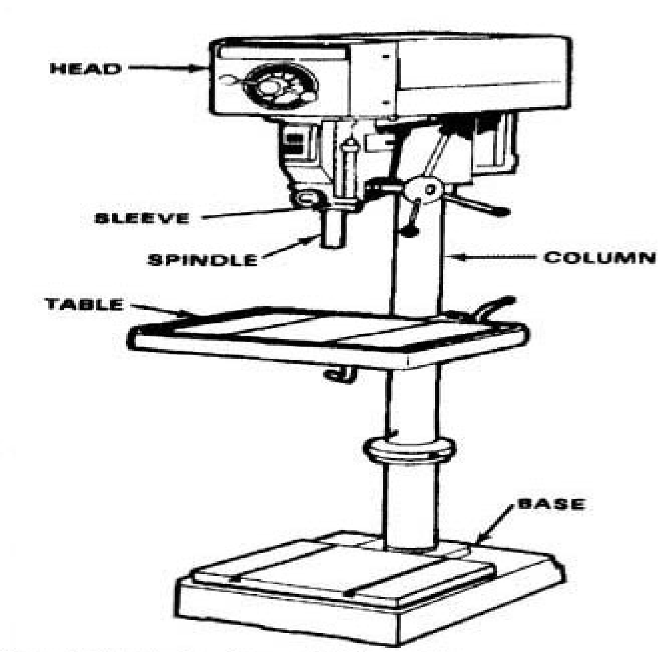


Figure 4-4. Construction of an upright drilling machine.

V. LITERATURE REVIEW

1) Demirer a, Y. Soydan b, A.O. Kapti b,Concluded that," In this experimental study, optimal process conditions, variations in length-wise and width-wise shrinkage rates, warpage rates and densities of samples produced of ABS and PP polymers were determined with respect to the changes in process temperature and injection pressure when using HRS, in comparison with CRS. It was observed that the required injection pressure in HRS was considerably lower. When using HRS, injection moulding process can be performed at lower process temperature and injection pressure than the case of using CRS. It was noted that the pressure gain can reach up to 33.33% for ABS and 42.85% for PP. Such a saving in required power results in accordingly less energy consumption by the injection moulding machine and hence smaller machines with less power can be utilized for producing relatively large components. This gain reduces the requirement for mould clamping force, increases the lifetime of the mould and injection machine, and allows the significant cuts in production costs. It was observed that the shrinkage rate decreased with increasing injection pressure for both runner systems. Results showed that the usage of HRS decreases the shrinkage rates for both of the polymers in comparison with CRS. This shrinkage-decreasing effect of HRS results from more influential packing stage due to late solidification of the gates, lower heat and pressure losses and better fluidity of molten plastic.[1]

2) N. Crisan a, S.Descartes a,n, Y.Berthier a, J.Cavoret a, D.Baud b, F.Montalbano c said that," This study has allowed the identification and evaluation of defaults that occur during plastic injection process, at microscopic scale. The results obtained highlight the different damage mechanisms sustained by the mold surface, as a function of polishing, geometry and injected material. It can be also observed that for each material injected there is a difference of level of wear and damage mechanism between the stamp and the matrix.

Surlyns injection exhibited consider able amount of deposits on the mold stamp. It seems that the physico-chemical conditions, created during the injection by this type polymer, favored the adhesion. Also in this case, the coupling effects of polishing qual- ity ,the injected material ,adhesion and the lack of the mold feature that evacuates air , tend to form corrosion pits on a mirror polished surface.[2]

3) Michael Dawoud∗, Iman Taha, Samy J. Ebeid explained that," The mechanical properties of ABS prepared by injection molding generally came superior in all the conducted tests if compared to those of 3D printing. This is related to the nature of the injection molding process which results in higher material compaction in addition to the enhancement of crystalline structure, thus enhancing mechanical strength. However, this study shows that the adequate selection of FDM parameters can yield parts of acceptable properties. Here, primarily the raster gap is a crucial parameter where at a negative gap of−0.05 mm, the material becomes denser almost reaching density of injected parts. This high packing of the material makes the effect of changing raster angle insignificant for static testing. Thus, a proper selection of the raster angle with respect to the loading direction becomes especially important when printing at positive raster gaps. Here, it is to be noted that raster's lying parallel to the loading direction will be able to carry all load in contrast to filaments in transverse direction that insignificantly contribute to load bearing. In contrast, however, dynamic behavior is significantly affected by the raster angle regardless of the raster gap direction. This behavior was related to internal stresses caused by thermal gradients during the printing stage.

parts fabricated according to the −P45 printing conditions are able to reach 91% of the tensile strength and86% the flexural strength of injection moulded products. This slight reduction in strength often lies within the acceptable range for simple DIY parts, proving the applicability of self-made 3D printers for personal use. Summarizing, a negative raster gap in combination with a 45◦raster angle is recommended if a high density part with enhanced mechanical properties is desired. Next to mechanical performance, the FDM technique applied in this study proved acceptable dimensional accuracy, lying within the nominal range of the specified size. Further, no significant dimensional variation was observed with altering raster angle and gap.[3]

4) M. Reiterb S. Stemmler Ch. Hopmann A. Ressmann D. Abel said that,” a Model Predictive Controller for cavity pressure control in an injection moulding process was presented along with an underlying, physically motivated control model. The model is simple enough to be intuitively checked for plausibility. Also, the proposed experiments that were used to parameterize the model can be easily incorporated into a real-life workout, as they are mostly based on well-established control strategies. Although a robust PI-controller for cavity pressure is used to conduct one of the experiments, this PI-controller does not necessarily need to yield high performance. Further research and experiments are planned to cross-verify how well the proposed concept can be transferred to other combinations of moulds and sensors.[4]

5) Harshal P. Kale1, Dr. Umesh V. Hambire told that,” A review of literature on optimization techniques has revealed that there are, in particular, successful industrial applications of design of experiment-based approaches for optimal settings of process variables. Taguchi methods are robust design techniques widely used in industries for making the product/process insensitive to any uncontrollable factors such as environmental variables. Taguchi approach has potential for savings in experimental time and cost on product or process development and quality improvement. There is general agreement that off-line experiments during product or process design stage are of great value. Reducing quality loss by designing the products and processes to be insensitive to variation in noise variables is a novel concept to statisticians and quality engineers. Taguchi and ANOVA methods were used to investigate the effects of melt temperature, injection pressure, packing pressure, packing time and cooling time on the shrinkage of the HDPE material.[5]

6) Xiang AnYang a,b, FengRuan b,concluded that,” The new method has also been compared with the existing three methods. The characteristic and application scope for the four methods is also discussed. The works done reveal that RN method should be applied in those fields where the normal direction of nodes reflects the tendency of spring back, for example in the case of simple shallow shape forming. When less rotation and larger displacement occur during spring back and thus the normal direc- tion of nodes would not reflect the tendency of spring back, such as the flange part of U-shape product, RD method would be suitable for that. While taking the advantage of RN method and RD method, CC method is general and gives good performance. As to RS method, as reported early [17], it would not converge for those features as the side-wall and have poor precision. In one-step compensation for springback, CC method performs very well when the right compensation magnitude is applied. But since the compensation magnitude is different at each node for complex shape, more works should be done to determine the reasonable compensation factor in the future.[6]

7) S.C. Yoon, P. Quang, S.I. Hong, H.S. Kim said that,” The effects of the die corner angle and the strain hardenability of the workpiece material on the plastic flow behavior and deformation inhomogeneity during ECAP were studied using FEM analyses. The less sheared zones are formed in non-strain hardening materials of the round corner die conditions and in strain hardening materials. In the strain hardening materials, the deformed geometry was predicted to be almost independent of the die corner angle. The optimum die corner angle among the conditions in this study is $\Psi = 3^{\circ}$, which may enhance the homogeneity of strain of severely plastic deformed workpieces during ECAP.[7]

8) B.Koteswararaoa, K.Siva kishore Babua , D.Ravib ,K.Kishore Kumara , P.Chandra shekarc told that,” Experiments were conducted according to Taguchi method by using the machining set up and the designed U-shaped tubular electrodes with internal flushing. Discharge current is most influencing factor on MRR and then pulse duration time and the last is diameter of the tool. In the case of Tool wear rate the most important factor is discharge current then pulse on time and after that diameter of tool. In the case of over cut the most important factor of discharge current then diameter of the tool and no effect on pulse on time. In the present study on the effect of machining parameters on MRR, TWR and OC of the EN31 alloy steel component using the cu tool with internal flushing system tool have been investigated for EDM process. The experiments were conducted under various parameters setting of Discharge Current (Ip), Pulse On-Time (Ton), and diameter of the tool. L-18 OA based on Taguchi design was performed for Minitab software was used for analysis the result and theses responses were partially validated experimentally. Discharge current is most influencing factor on MRR and then pulse duration time and the last is diameter of the tool. MRR increased with the discharge current (Ip). As the pulse duration extended, the MRR decreases monotonically. In the case of Tool wear rate the most important factor is discharge current then pulse on time and after that diameter of tool. In the case of over cut the most important factor is discharge current and then diameter of the tool and no effect on pulse on time.[8]