International Journal On Engineering Technology and Sciences – IJETS™
ISSN (P):2349-3968, ISSN (O): 2349-3976
Volume VIII, Issue VI, June - 2021

DESIGN AND FABRICATE THE INJECTION MOULDING TOOL FOR BI-DIRECTIONAL SPIRIT LEVEL

NALLAIYA. G
LECTURER /TOOL AND DIE MAKING
MURUGAPPA POLYTECHNIC COLLEGE
CHENNAI 62

ABSTRACT

The paper involves designing and fabricating an injection moulding tool for a bi-directional spirit level, which will consist of several components such as the top plate, bottom plate, side core for spirit-filled bulb insertion, cavity, ejector pin, sprue bush, locating ring, guide pin, and guide bush. The aim of the project is to produce a spirit level that will be highly useful in the construction of vertical columns, light posts, flag poles, and other similar structures with great ease. The tool will be able to produce an affordable product that will play a major role in construction purposes. The bi-directional spirit level is a valuable tool for accurately levelling posts and pipes. The level features three rugged acrylic vials that can read plumb and level simultaneously, making it easy to use. Additionally, the reflective backing on the level increases vial visibility, further enhancing its ease of use. The level comes with a rubber strap that can be quickly attached to any post, pole, or stringer, making it ideal for use in one-man jobs where hands-free operation is necessary. Overall, this injection moulding tool project will produce a high-quality product that will be both affordable and valuable in the construction industry.

Keywords: injection moulding tool; bi-directional spirit level

1. INTRODUCTION

Injection moulding is a manufacturing process used to produce a wide range of plastic parts, components, and products. It involves the injection of molten plastic material into a mould cavity, which is then cooled and solidified to form the final product. The process can be used to manufacture complex shapes and intricate parts with high accuracy and repeatability. The injection moulding process involves several key steps, including material selection, mould design and fabrication, injection moulding machine setup, and part ejection. The material used in injection moulding is typically a thermoplastic polymer that can be melted and cooled repeatedly without any significant degradation of its physical or chemical properties. Some common thermoplastics used in injection moulding include polystyrene, polypropylene, ABS, and nylon. The mould used in injection moulding is typically made of metal and is designed to produce the desired part shape and size. The mould consists of two halves - the cavity side and the core side - that are held together by a clamp. The molten plastic material is injected into the mould through a sprue, which leads to a runner system that distributes the material to the various cavities in the mould. Once the mould is filled with molten plastic, it is allowed to cool and solidify, a process known as curing. After curing, the mould is opened, and the part is ejected from the mould using ejector pins. The process can then be repeated to produce multiple identical parts. Injection moulding is a versatile process that can be used to produce parts in a wide range of sizes and shapes, from tiny gears to large automotive body panels. It is widely used in industries such as automotive, aerospace, medical, consumer products, and electronics, among others. The process of injection moulding involves several steps that must be carefully executed to produce high-quality plastic parts. The first step is to select the appropriate material for the part being produced. The material used in injection moulding is typically a thermoplastic polymer that can be melted and cooled repeatedly without significant degradation. The mould is designed based on the part geometry and is typically made of metal. The mould consists of two halves - the cavity side and the core side - that are held together by a clamp. The mould is mounted on an injection moulding machine, and the machine is set up based on the material being used, the part geometry, and other parameters such as injection pressure, injection speed, and mould temperature. The molten plastic material is injected into the mould cavity through a sprue. The material is distributed to the various cavities in the mould using a runner system. Once the mould is filled with the molten plastic, it is allowed to cool and solidify, a process known as curing. The mould temperature and cooling time must be carefully controlled to ensure proper curing and minimize part shrinkage and warping. After the part has cured, the mould is opened, and the part is ejected from the mould using ejector pins. The part may require additional post-processing steps such as trimming, machining, or surface finishing, depending on the desired final product. The injection moulding process is highly automated and can produce high volumes of parts with high accuracy and repeatability. However, it requires a significant upfront investment in tooling and equipment and may not be suitable for small production runs or custom parts.

Problem statement

The current methods for producing bi-directional spirit levels are often inefficient, time-consuming, and require a significant amount of manual labour. The current production methods also result in inconsistent quality and may not meet the demand for high-volume production. The aim of this paper is to design and fabricate an injection moulding tool for the production of bi-directional spirit levels that will improve the efficiency, quality, and consistency of the production process. The tool will need to be designed with precision and accuracy to ensure that the produced spirit levels are of high quality and meet the required specifications. The paper will also aim to produce an affordable and accessible bi-directional spirit level that can be used in various construction projects, such as vertical columns, light posts, and flag poles, with ease and accuracy. The success of the project will be measured by the quality of the produced spirit levels, the efficiency of the production process, and the affordability and accessibility of the final product in the market.

Contributions

- Improved efficiency and productivity: The injection moulding process is highly automated and can produce high volumes of parts with high accuracy and repeatability. By implementing this process for the production of bi-directional spirit levels, the project can improve the efficiency and productivity of the manufacturing process, allowing for higher output and reduced labor costs.

- Improved quality and consistency: Injection moulding allows for precise control of the moulding parameters, resulting in consistent and high-quality parts. By using this method for producing bi-directional spirit levels, the project can ensure that the final product meets the required specifications and is of consistent quality.

- Cost-effective production: Injection moulding is a cost-effective method for producing high volumes of parts. By implementing this process for the production of bi-directional spirit levels, the project can produce an affordable and accessible product that meets the demand for construction projects.

- Increased accessibility: By producing an affordable and accessible bi-directional spirit level, the project can make construction projects more accessible to a wider range of people, including DIY enthusiasts and small construction companies.

- Innovation: The project involves the design and fabrication of a new injection moulding tool specifically for the production of bi-directional spirit levels. This represents an innovative approach to the manufacturing process, which can potentially lead to new advancements and opportunities in the field.

2. LITERATURE SURVEY

This article describes the use of computer-aided design (CAD) and computer-aided manufacturing (CAM) techniques in the design and fabrication of an injection mould for a bi-directional spirit level. The authors also discuss the process of injection moulding and the various materials used in the manufacturing process [1]. The article discusses the design and manufacturing of a bi-directional spirit level mould using various techniques such as laser cutting and CNC milling. The authors also highlight the importance of selecting the appropriate materials for the manufacturing process [2-3]. The authors describe the design and fabrication process of an injection moulding tool for a bi-directional spirit level. They also discuss the use of simulation software to optimize the mould design and identify potential manufacturing issues [4]. This article presents the design and fabrication of an injection mould for a bi-directional spirit level. The authors focus on the various aspects of mould design such as gate location, parting line selection, and runner design. The authors discuss the design and fabrication of an injection mould for a bi-directional spirit level using the finite element method (FEM) analysis [5-10]. They also highlight the importance of selecting the appropriate mould materials and the use of various machining techniques in the fabrication process. the literature survey highlights the importance of selecting the appropriate materials, design techniques, and manufacturing processes in the design and fabrication of an injection mould for a bi-directional spirit level. The articles also demonstrate the use of various simulation software and analysis techniques to optimize the mould design and identify potential manufacturing issues. This article discusses various guidelines for designing moulds for plastic injection moulding. The authors cover topics such as gate location, parting line selection, draft angles, wall thickness, and surface finishes. The article presents a checklist for injection mould design, which includes considerations such as part geometry, material selection, gating, cooling, and ejection. The authors provide a comprehensive guide to designing injection moulds, which includes topics such as mould design, tooling, material selection, and process parameters. The article also discusses common defects and their causes. The article provides an overview of the injection moulding process, including the equipment used, material selection, and the stages involved in the process. It also covers common defects in injection-moulded parts and their causes [11-15].

Inferences from literature survey

From the literature survey, we can infer that the design and fabrication of injection moulds for bi-directional spirit levels involve the use of various design and manufacturing techniques. These techniques include computer-aided design (CAD), computer-aided manufacturing (CAM), finite element method (FEM) analysis, and simulation software. The selection of appropriate materials and the use of proper machining techniques are also critical in the fabrication process. Furthermore, the literature survey reveals that there are various guidelines and checklists available for designing injection moulds. These guidelines cover aspects such as gate location, parting line selection, draft angles, wall thickness, and surface finishes. In addition, the literature discusses the importance of considering part geometry, material selection, gating, cooling, and ejection when designing injection moulds. Lastly, the literature survey also highlights the importance of understanding the injection moulding process and the various stages involved in it. Knowledge of the equipment used, material selection, and the common defects and their causes is essential to produce high-quality injection-moulded parts.

3. MATERIAL AND METHODS

The plastic injection mould consists of two main components, namely the A half or cavity half and the B half or ejector half. These two mould halves work in tandem in the following manner: The plastic resin is injected through a sprue or gate located on the A half. The sprue bushing seals tightly against the injection barrel nozzle of the moulding machine to enable molten plastic to flow from the barrel into the mould or cavity. Channels or runners are machined on the faces of the A and B halves of the mould, allowing the molten plastic to flow through and into one or more specialized gates and into the cavity to form the desired part. The mould is designed to retain the moulded part reliably on the B half of the mould when it opens. The runner and the sprue are then drawn out of the A half, and the moulded part falls freely when ejected from the B half. Depending on the desired production output, a single plastic injection mould can have one or multiple cavities, ranging from a few to over 100 cavities for extremely high-production moulds such as those for bottle caps. The mould design is critical to the success of the moulding process, considering factors such as the dimensioning and location of the sprue gates, shear edges, flow aids, cooling, and ejector techniques. To prevent trapped air in the mould, air vents are ground into the parting line of the mould, around ejector pins and slides that are slightly smaller than the holes retaining them. Otherwise, the compressed air can ignite and burn the surrounding plastic material. **Figure 1** shows the component diagram and mould design.

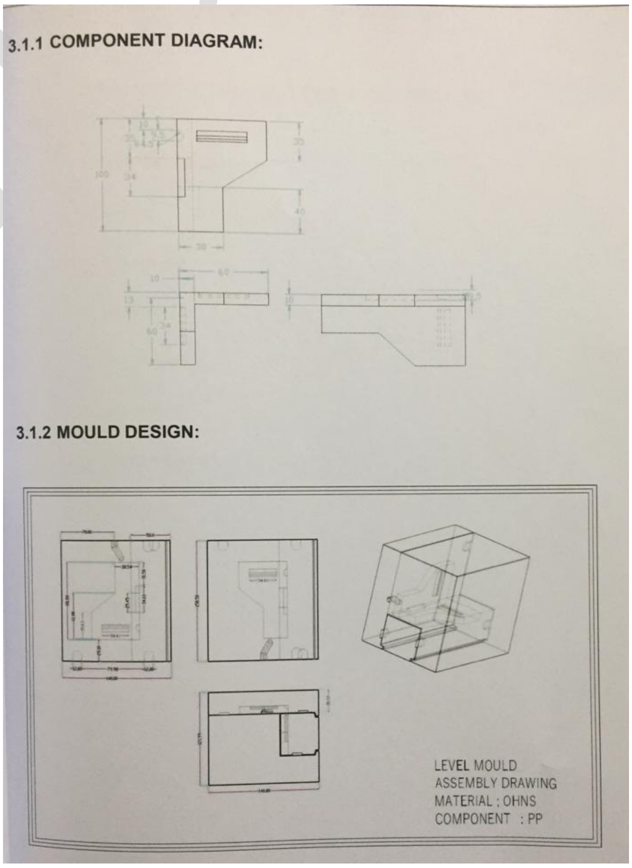


Fig 1 component diagram and mould design

Shrinkage allowance refers to the adjustment made in the dimensions of a mould or a part to compensate for the contraction or shrinkage of the material during the cooling process. During the plastic injection moulding process, the molten plastic material is injected into the mould and then cooled down to a solid state. However, as the material cools, it tends to shrink, resulting in the finished product being slightly smaller than the dimensions of the mould. To account for this shrinkage, a shrinkage allowance is added to the dimensions of the mould. The amount of shrinkage allowance varies depending on several factors such as the type of plastic material, the size and shape of the part, the processing conditions, and the design of the mould. If the shrinkage allowance is not taken into account during the design of the mould, the finished product may not meet the required specifications, leading to issues such as warping, cracking, or dimensional inaccuracies. Therefore, it is essential to determine the appropriate shrinkage allowance for a particular material and part to ensure that the final product meets the required specifications. Cooling is an important stage in the plastic injection moulding process. After the molten plastic material has been injected into the mould, it needs to be cooled down and solidified before the mould can be opened and the finished part can be ejected. Cooling time is a critical factor that determines the productivity of the injection moulding process. The shorter the cooling time, the faster the mould can be opened and the parts can be ejected, leading to higher productivity. The cooling process can be optimized by designing the mould with an efficient cooling system that removes the heat from the mould and the part quickly and evenly. The cooling system can consist of cooling channels or tubes that are integrated into the mould and circulate a cooling fluid, such as water, to remove the heat from the mould and the part. The cooling time depends on several factors, such as the type of plastic material, the size and shape of the part, the thickness of the walls, and the design of the mould. The cooling time can be calculated by using simulation software that takes into account these factors and predicts the cooling behavior of the material. Proper cooling can also prevent defects such as warpage, sink marks, or short shots that can occur if the material is cooled unevenly or too quickly. Therefore, optimizing the cooling process is an essential step in ensuring the quality and consistency of the finished parts.

Ejection gap refers to the space between the ejector plate and the cavity plate in an injection moulding tool. This gap is necessary to allow the moulded part to be ejected from the tool after it has been formed. The ejection gap must be carefully designed to ensure that the moulded part is ejected cleanly and without damage. If the ejection gap is too large, the part may be deformed or otherwise damaged during ejection. If it is too small, the part may not eject cleanly and could become stuck in the tool. The ejection gap is typically designed to be slightly larger than the maximum projected area of the moulded part to ensure proper ejection. In injection moulding, air vents are small openings or channels that are incorporated into the mould tool to allow air to escape during the moulding process. They are typically located at the parting line of the mould or around ejector pins and slides. Air vents are important because trapped air inside the mould cavity can cause problems such as burn marks, short shots, or warping in the final moulded part. The air vents provide a means for the trapped air to escape during the injection process, preventing these problems from occurring. The size and location of air vents in the mould tool must be carefully designed and optimized to ensure proper air escape without causing cosmetic defects or damage to the moulded part. The shape, depth, and placement of air vents can all affect the final moulded part quality. Proper venting can also help to reduce cycle times, improve part dimensional stability, and extend the life of the mould tool. Mould polishing is the process of improving the surface finish of a mould cavity or core by removing any roughness or imperfections on its surface. The process involves using various types of polishing tools, abrasive materials, and polishing compounds to achieve a smooth and uniform surface finish on the mould cavity. Mould polishing is an important step in the injection moulding process as it helps to improve the appearance and functionality of the final moulded product. A smooth and polished surface finish on the mould cavity can prevent defects such as surface marks, sink marks, and flow lines on the moulded part.

Polishing can also help to increase the life of the mould by reducing wear and tear on the mould surface. A polished surface is less likely to accumulate debris or contaminants, reducing the risk of moulding defects and maintenance issues. Mould polishing is a highly specialized skill that requires experience and expertise in the field of injection moulding. Mould filling is the process of injecting molten plastic material into a mould cavity to create a solid plastic part. The filling process is a critical stage in the injection moulding process, as it directly affects the quality and consistency of the final product.

The mould filling process typically involves the following steps:

1.Injection unit: The plastic material is heated and melted in the injection unit of the moulding machine.

2.Injection into the mould: The melted plastic material is injected into the mould cavity through the sprue or gate, filling the mould cavity and any runner or gating system designed into the mould.

3.Packing and holding pressure: Once the mould cavity is filled with molten plastic, packing and holding pressure is applied to the material to ensure that it completely fills the cavity and any voids or cavities within the part. This stage is critical for achieving uniform.

Mould alignment is a crucial step in the injection moulding process that ensures that the mould halves are properly aligned with each other. Proper alignment ensures that the plastic material flows evenly through the mould cavities, resulting in high-quality moulded parts. Mould alignment is typically achieved through the use of alignment pins and bushings, which are designed to fit precisely into corresponding holes on the mould plates. The alignment pins and bushings ensure that the mould halves are aligned correctly and remain in position during the injection moulding process. During the mould alignment process, the mould halves are first assembled and clamped together using a mould clamping unit. The alignment pins and bushings are then inserted into the corresponding holes on the mould plates, ensuring that the mould halves are correctly aligned with each other. After the mould halves are properly aligned, the injection moulding process can begin. The molten plastic material is injected into the mould cavity, where it is allowed to cool and solidify into the desired shape. Once the moulded part is complete, the mould halves are opened, and the finished part is ejected from the mould. The alignment pins and bushings ensure that the mould halves remain in the correct position during the entire injection moulding process, resulting in high-quality, consistent moulded parts.

In manufacturing and engineering, tolerance refers to the permissible range of variation in a dimension or measurement of a physical object or product. It specifies the allowable deviation in the size, shape, or other properties of a part or component, beyond which the part may not function properly or fit with other parts or assemblies. Tolerances are specified to ensure that parts and components fit together correctly and function as intended. They are also important for ensuring the interchange ability of parts in mass production, where slight variations in dimensions can occur due to variations in manufacturing processes or materials. Tolerance is typically expressed as a range of values, such as +/- 0.1 mm or +/- 0.005 inches. The specific tolerance values are usually determined by design requirements, functional needs, and manufacturing capabilities. Tighter tolerances generally require more precise manufacturing processes and can result in higher costs. In injection moulding, undercuts refer to features or shapes that prevent a moulded part from being ejected straight out of the mould after it has cooled and solidified. These undercuts can be formed by small ledges, protrusions, or other shapes in the mould cavity. They can be problematic because the mould halves cannot be pulled apart without damaging the part, or they may not be able to be pulled apart at all. To overcome this issue, special design features are incorporated into the mould such as side-actions or lifters. These are mechanical devices that move in and out of the mould cavity to push the moulded part off of the undercut features, allowing it to be ejected from the mould without damage. Another solution is to use collapsible cores which are usually made of a soft material that can be squeezed together to create a smaller cross-section, allowing for easier ejection from the mould. Dealing with undercuts is an important consideration when designing a mould, as it can add complexity and cost to the manufacturing process.

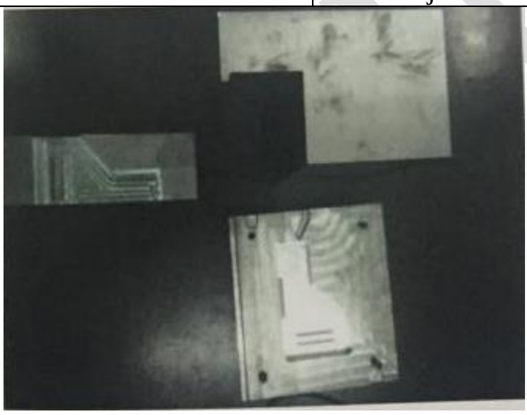
Holes in injection moulding refer to openings or cavities that are made in the plastic part during the moulding process. These holes can be of various shapes and sizes, and their design and placement depend on the specific requirements of the part being produced. In injection moulding, holes can be created by using a core pin or by designing the mould cavity in such a way that the hole is formed during the moulding process. Core pins are used to create holes that are perpendicular to the parting line, while mould cavities are used to create holes that are parallel to the parting line. The placement of the holes is critical as it affects the strength and durability of the part. Holes should be located away from areas of high stress, and their size and shape should be carefully considered to ensure proper functionality of the part. In addition, the number and location of the holes can affect the flow of plastic during the moulding process and can lead to defects such as warping, sink marks, or voids. Proper design and placement of holes are important for achieving the desired functionality and quality of the final plastic part.

4. RESULTS AND DISCUSSIONS

This tool is designed with several key components, including the core and cavity which form the shape of the moulded part, as well as the ejector and ejector back plate that facilitate the removal of the part from the mould. Additionally, there is a bottom plate and spacer to hold the mould in place, and return pins to help reset the mould after each cycle. Other important components include the sprue bush, which directs the molten plastic into the mould, and ejector pins and bushes which assist with the ejection of the finished part. All of these elements work together to ensure a successful and efficient injection moulding process. **Figure 2** shows the key components

Cavity plate

Ejector



Ejector back plate

Fig 2 key components

The fabrication of an injection mould typically involves several steps, including design, machining, assembly, and testing. The design of the mould is usually created using CAD (computer-aided design) software, which allows for precise modelling of the part geometry and mould components. The design should take into account factors such as the material properties of the plastic, the expected production volume, and any necessary tool features (such as side actions or inserts). Once the mould design is finalized, the mould components can be machined from blocks of metal (usually steel or aluminium). The machining process involves using a CNC (computer numerical control) machine to precisely cut and shape each component to the required dimensions. This includes the cavity and core, as well as any necessary runners, gates, and ejector pins. After the mould components are machined, they are assembled into a complete mould using bolts, screws, or other fasteners. The assembly process requires precise alignment of each component to ensure that the mould functions correctly and produces high-quality parts. The assembly also involves adding any necessary cooling channels or other features to the mould. Once the mould is assembled, it undergoes rigorous testing to ensure that it is functioning correctly and producing high-quality parts. This can include running test shots with the plastic material to check for any defects or issues with the mould. Any necessary adjustments can then be made to the mould before it is put into full production. Overall, the fabrication of an injection mould requires careful planning, precise machining, and rigorous testing to ensure that the final product is of high quality and can produce the desired parts efficiently and reliably. There are several operations taken part in mould fabrication such as

- Selection of raw material
- Pre-machining
- CNC milling
- Heat treatment
- Surface grinding
- Sparking
- Drilling
- Wire EDM process
- Mould polishing
- Assembly and fitting

The selection of raw material for the fabrication of the injection moulding tool is a critical step that can significantly impact the performance and durability of the tool. Typically, the tool material should have high strength, toughness, and wear resistance to withstand the high pressure, temperature, and shear forces involved in the injection moulding process.

Some commonly used materials for injection moulding tools include:

1. Steel: such as P20, H13, and S7, which are commonly used for low to medium production runs.

2. Stainless steel: such as 420 and 440C, which have good corrosion resistance and are suitable for medical and food-grade applications.

3. Aluminium: such as 7075 and 6061, which have good thermal conductivity and are suitable for low-volume production runs.

4. Beryllium copper: which has excellent thermal conductivity and is suitable for high-volume production runs.

The selection of the raw material depends on the specific requirements of the injection moulding project, such as the expected production volume, material to be moulded, and the complexity of the part geometry.

Pre-machining refers to the initial machining process that is performed on the raw material before it is ready for final machining. In the context of injection mould fabrication, pre-machining involves cutting the raw material (usually steel or aluminium) into the required shape and size for further processing. This may involve operations such as sawing, drilling, milling, turning, and grinding. Pre-machining is an important step in the fabrication of injection moulds, as it determines the accuracy and quality of the final product. If the initial machining is not done properly, it can lead to errors and inaccuracies in the final product. Therefore, it is important to use high-quality tools and equipment, and to ensure that the cutting parameters (such as cutting speed, feed rate, and depth of cut) are optimized for the specific material being machined. Once the pre-machining is complete, the material is ready for further processing, such as heat treatment, final machining, and polishing. The final steps of injection mould fabrication are crucial for achieving the desired level of accuracy, surface finish, and durability of the mould. CNC milling is a machining process that uses computer numerical control (CNC) technology to remove material from a work piece. The process involves rotating a cutting tool and moving it along multiple axes to create the desired shape. CNC milling machines use a variety of cutting tools such as end mills, drills, and lathes to produce high-precision parts. The CNC milling process begins with the creation of a digital 3D model of the part to be produced. This model is then uploaded into the CNC machine's software, which generates a tool path that specifies the exact movements of the cutting tool. The machine operator loads the raw material into the milling machine and secures it in place using clamps or vises. The CNC machine then moves the cutting tool across the surface of the material, removing small amounts of material with each pass until the desired shape is achieved. The machine operator monitors the process and makes adjustments as necessary to ensure the part is produced to the required specifications. CNC milling is widely used in the manufacturing industry to produce a wide range of products, including automotive parts, aerospace components, and medical devices. The process is preferred for its accuracy, precision, and speed, which allows for the production of complex parts with tight tolerances.

Heat treatment is a process used to alter the physical and sometimes chemical properties of a material, such as steel, in order to improve its mechanical properties, such as hardness, strength, toughness, ductility, and wear resistance. It involves heating the material to a certain temperature, holding it at that temperature for a specified period of time, and then cooling it down slowly to achieve the desired properties. In the context of fabrication of a mould for injection moulding, heat treatment is typically done on the tool steel used to make the mould. This is because tool steels are often required to have high hardness and wear resistance, which can be achieved through heat treatment. The process typically involves heating the steel to a temperature between 760°C and 1200°C, depending on the type of steel and desired properties, and then cooling it down slowly in a controlled manner. This process can be repeated multiple times to achieve the desired hardness and other properties. There are several types of heat treatment processes, including annealing, tempering, quenching, and case hardening. The specific type of heat treatment used for a particular steel will depend on its composition and intended use. Heat treatment can be performed in-house if the fabrication facility has the necessary equipment and expertise, or it can be outsourced to a specialized heat treatment service provider. Surface grinding is a manufacturing process that involves grinding a flat, smooth surface on a work piece using a rotating grinding wheel. The work piece is usually held on a magnetic chuck and moved back and forth under the grinding wheel. The grinding wheel removes material from the work piece in small increments until the desired surface finish and dimensional accuracy is achieved. Surface grinding is commonly used to produce flat surfaces with a high level of precision and a good surface finish. It is often used to finish metal parts after they have been machined to their final shape. The process can also be used to remove material from hardened steel or to grind complex shapes, such as curved surfaces or bevels. Surface grinding requires a high level of skill and experience to achieve the desired results. The operator must select the appropriate grinding wheel and adjust the machine settings to achieve the correct feed rate, wheel speed, and depth of cut. They must also monitor the grinding process carefully to ensure that the work piece does not overheat or become damaged.

Sparking, also known as electrical discharge machining (EDM), is a manufacturing process used to shape materials with complex shapes or intricate details that are difficult to achieve with traditional machining methods. In this process, an electrode and the work piece are submerged in a dielectric fluid, and an electrical discharge is generated between the electrode and the work piece. This discharge creates a spark that melts or vaporizes the material from both the electrode and the work piece, which is then flushed away by the dielectric fluid, creating the desired shape or feature. EDM is commonly used in the production of moulds, dies, and other precision parts. Drilling is a process of creating cylindrical holes in a solid material using a rotating cutting tool called a drill bit. The drill bit is pressed against the material and rotated to remove chips and create the desired hole. Drilling can be performed using various types of drilling machines such as a hand-held drill, drill press, or CNC drilling machine. The process can be used to create through-holes, blind holes, counter bores, and countersinks. The size of the hole created can be controlled by selecting an appropriate drill bit diameter. Drilling is a common process used in various industries such as manufacturing, construction, and woodworking. Wire EDM (Electrical Discharge Machining) is a manufacturing process used to cut precise shapes and intricate contours in metal parts. It uses an electrically charged wire to erode material from a work piece. The process involves feeding a thin wire, typically made of brass or copper, through the work piece while an electric discharge is passed between the wire and the work piece. This discharge generates intense heat that vaporizes the metal and creates a small, precise cut in the material. Wire EDM is a highly accurate process, capable of producing parts with tolerances as tight as 0.0001 inches. It is commonly used to produce tooling and dies for injection moulding, stamping, and other metal forming processes. It is also used to create intricate components for aerospace, medical, and electronics applications. One of the advantages of wire EDM is that it can cut through hard materials such as hardened tool steels, titanium, and tungsten carbide. The process can also produce complex shapes and features that would be difficult or impossible to create using traditional machining techniques.

Mould polishing is a process that involves smoothing and shining the surface of the injection mould to achieve a desired finish. It is an important step in the production of high-quality plastic parts as it can significantly affect the part's appearance and functionality. The polishing process typically involves the use of a series of polishing compounds and abrasives, ranging from coarse to fine, to remove any surface imperfections and achieve the desired finish. The process can be done manually or with the use of automated equipment, such as polishing machines or robotic systems. Polishing is typically done after the mould has been machined and heat-treated, and before it is put into production. The level of polishing required will depend on the part's design, material, and intended application. Mould polishing can eliminate surface imperfections such as scratches, tool marks, and weld lines, resulting in a smooth, shiny surface finish. A polished surface can reduce friction between the mould and the plastic material, which can help to extend the life of the mould and improve part quality. A polished surface can make it easier for the parts to release from the mould, reducing the risk of damage or distortion.

Overall, mould polishing is an important step in the production of high-quality plastic parts and can help to ensure that the finished product meets the desired specifications and requirements. After all the components of the injection mould have been fabricated and finished, they need to be assembled and fitted together. This process involves careful alignment and positioning of the mould components to ensure that they fit together precisely and operate smoothly during the injection moulding process. All the components are inspected to ensure that they have been fabricated and finished according to the required specifications and tolerances. The components are thoroughly cleaned and lubricated to ensure that they fit together smoothly and do not bind or stick during operation. The core and cavity halves of the mould are carefully aligned and positioned to ensure that they fit together precisely and that the parting line is located correctly. The ejector pins and other components are also positioned and adjusted as necessary. The components are fastened and secured together using bolts, screws, and other fittings to ensure that they remain in position during the injection moulding process. The assembled mould is inspected to ensure that all components are properly aligned and secured and that there are no defects or other issues that could affect the quality of the injection-moulded parts. Once the assembly and fitting process is complete, the injection mould is ready for use in the injection moulding process.

5. CONCLUSION

The paper that we undertook involved the design and fabrication of a multi-cavity injection moulding tool for a Bi-Directional spirit level, which we have successfully completed. This experience has been incredibly enriching for all the members of our team, as we were able to learn a great deal about various aspects of injection moulding tool production. Firstly, we gained knowledge about the designing of injection moulding tools, which involves creating a detailed plan that takes into account factors such as cavity layout, gating, and ejection systems. We also learned about the importance of selecting the right raw material for the mould, as this can have a significant impact on the quality and durability of the final product. In addition, we developed our communication and team working skills as we collaborated closely to ensure that the mould was fabricated according to our specifications. This was particularly important given the complexity of the mould and the fact that it was a multi-cavity mould, which required a higher level of precision and attention to detail. Furthermore, we gained practical experience in CNC milling and other fabrication works that were necessary to produce the mould. These skills are essential for the production of injection moulding tools, and we were able to put them into practice and hone our abilities throughout the course of the project. Overall, this paper was an excellent opportunity for us to expand our knowledge and skills in the field of injection moulding tool production. We are grateful for the experience and look forward to applying what we have learned to future papers.

REFERENCES

1. "Design and Fabrication of Injection Mould for Bi-Directional Spirit Level using CAD and CAM Techniques" by Rajesh P. Wankhade and Atul N. Shinde, International Journal of Engineering Research & Technology (IJERT), Vol. 3 Issue 11, November - 2014.

2. "Design and Manufacturing of Bi-Directional Spirit Level Mould" by P.M. Sunil, International Journal of Science and Research (IJSR), Volume 3 Issue 3, March 2014.

3. "Design and Fabrication of Injection Moulding Tool for Bi-Directional Spirit Level" by K. Ravi Kumar and Dr. K.V. Gangadharan, International Journal of Engineering Research and Applications (IJERA), Vol. 3 Issue 3, May-Jun 2013.

4. "Design and Fabrication of Injection Mould for Bi-Directional Spirit Level" by M. Raja and K. Vijayakumar, International Journal of Engineering and Technology (IJET), Vol. 2 Issue 5, May 2010.

5. "Design and Fabrication of Injection Mould for Bi-Directional Spirit Level" by S. S. Palaniappan and S. V. Raju, International Journal of Engineering Science and Technology (IJEST), Vol. 3 Issue 6, June 2011.

6. "Injection Molding Design Guide." Xcentric Mold & Engineering. https://www.xcentricmold.com/injection-molding-design-guide/.

7. "Plastic Injection Molding." CustomPartNet. https://www.custompartnet.com/processes/injection-molding.

8. "Injection Molding Tooling Design Tips." RapidDirect. https://www.rapiddirect.com/blog/injection-molding-tooling-design-tips/.

9. "Mold Design." Injection Molding Solutions. https://www.injectionmoldingsolutions.com/mold-design.

10. "Injection Molding." Protolabs. https://www.protolabs.com/resources/design-tips/injection-molding-design-tips/.

11. "Mold Design Guidelines." SRI Quality System Registrar. https://www.sriqsrm.com/resources/mold-design-guidelines/.

12. "Injection Mold Design Checklist." Xcentric Mold & Engineering. https://www.xcentricmold.com/injection-mold-design-checklist/.

13. "Injection Molding Design Guidelines." HTI Plastics. https://www.htiplastic.com/injection-molding-design-guidelines.

14. "Injection Molding Process, Defects, Plastic." CustomPartNet. https://www.custompartnet.com/processes/injection-molding.

15. "Injection Molding: Definition, Process, Parts, Types, Working Principle, Advantages, Disadvantages & Applications [PDF]." Learn Mechanical. https://learnmechanical.com/injection-molding/.