A Guide
to
INJECTION MOULDING TECHNIQUE

Dinbandhu Singh

A Guide

to

INJECTION MOULDING TECHNIQUE

By : Dinbandhu Singh

First Impression: 2017

A Guide to Injection Moulding Technique

ISBN : 978-81-930928-3-5

DISCLAIMER

Information contained in this book has been published by Empyreal Publication House and has been obtained by the author from sources believed to be reliable and are correct to the best of his knowledge. The authors are solely responsible for the contents of the articles compiled in this book. Responsibility of authenticity of the work or the concepts / views presented by the author through this book shall lie with the author and the publisher has no role or claim or any responsibility in this regards. Errors, if any, are purely unintentional and readers are requested to communicate such error to the author to avoid discrepancies in future.

Published by:
Mr. Dinbandhu Singh
Assistant Professor, Department of Mechanical Engineering
Vidya Vihar Institute of Technology, Maranga
Purnea-854303, Bihar
Mobile No: 9525874626 / 7017509776
Email: dinosingh@hotmail.co.uk / dinomathur@gmail.com

Dedicated to my uncle

Shree Jitendra Singh

ACKNOWLEDGEMENT

I am very thankful to the Management of R.V. College of Engineering, Bangalore and Sai Consultancy Services & Tools, Bangalore for providing me an opportunity to work on this project.

It gives me immense pleasure to express my deep sense of gratitude to our beloved Principal, Dr. B.S. Satyanarayana, whose words of advice have always been a constant source of inspiration for me.

With a profound sense of gratitude and regards I convey my sincere thanks to my project guide Dr. H.N. Narasimha Murthy, Professor, Dean-PG Studies, whose timely precious advice, valuable suggestions and support throughout the project. His clear thinking penetrating insight into matters relating to the study has enabled me to transfer the path with considerable smooth.

I would like to express my heartfelt thanks to the Dr. B. Anand, Head of the Department of Mechanical Engineering, for his valuable advice and encouragement to me in completing this project work.

I express my deep gratitude to Mr. Anup Singh, Sai Consultancy Services &Tools, Bangalore for his guidance and untiring support throughout the development of this project.

My special thanks go to my parents Sri. Anil Singh and Smt. Lila for their blessings has been a constant source of inspiration for me.

Finally, I would like to thank all the faculty and supporting staff of the department of Mechanical Engineering and Sai Consultancy Services & Tools, Bangalore and to all those who have directly or indirectly helped me towards the successful completion of this project.

DINBANDHU SINGH

ABSTRACT

Injection moulding is one of the most popular commercial manufacturing techniques in plastic industry. It has the ability to produce complex design products and is an automated, highly cost-effective, precise and competent manufacturing technique. The design of a plastic injection mould is an integral part of plastic injection moulding, as the quality of the final plastic part is greatly reliant on the injection mould.

The book entails design, manufacturing and validation of an injection mould for 'Rotor and Cover' of a plastic component used in a particular model of a two wheeler. The component weighing five gram is made of Nylon-6/6 (13% glass filled). The customer requirement was to design a flash free component for a life of 300,000 components per tool, catering to the recent design changes. Important features of the project included design of injection mould, feeding system, ejector system, selection of suitable machine based on clamping force calculation and manufacture of mould.

The project begins with the study of component for shape, size and critical parameters to aid in the proper selection of parting surface, ejection system and feed system. Design protocol was on conventional lines, based on tests and trail norms developed through experience, reported in the literature. The design exercise involved the choice of the parting surface, allowance for shrinkage, draft, gate location, venting, thermal balancing of mould and choice of ejection system. The mould has one side core, one lifter, and one blade ejector for the rotor part and pin ejector for the cover part. CAD of core and cavity and assembly of the injection mould were performed using CATIA V5 for better visualization and, detailed drawings by Auto Cad to serve as an input to manufacturing.

The final phase was manufacturing and assembling of the parts of the mould. Manufacturing of the mould was strictly done as per the drawing specifications and the mould trial was carried out on the injection moulding machine. The final product produced was found to meet all the specifications as per the customer requirements. The study of design variations in the mould may be explored for adopting the same for the other two-wheelers.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF DRAWINGS

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

Injection moulding requires that the material be heated to a plastic state in a barrel and then forced into the mould cavity with a plunger under high pressure. The plasticization of the material in the barrel is facilitated by shear and friction from a rotating screw that also feeds the raw material. In thermoplastics, the material is cooled in the mould until it can be ejected. With thermo-set, the material is heated in the mould to "cure" the material before ejection. The construction of mould depends upon the shape of component, quantity required, its size and complexities of the parting surface/plane.

Before designing the mould, the designer should carefully analyze the shape, size, undercuts, holes, bosses, and sharp corners of the component. One should also try to check from preliminary calculations, the clamping force required, size of insert required, which intern decides the size of the mould, and the injection moulding machine required for production. The number of cavities in mould also plays a main role in deciding the size of the mould. The number of cavities in the mould is sometimes, as per the specification given by the customer. In this book, a family mould is considered for the design of two different types of components.

The book entitles the design, fabrication and validation of the mould for the manufacture of plastic components 'Rotor and Cover' weighing five grams, aimed at high productivity. Since this component is an automobile part of a two-wheeler, its life should be for 300000 per mould as per customer specification, which is due to new design change requirement these days. A new mould with higher productivity with flash free component is the primary requirement.

To accomplish this, a careful examination of the component drawing is carried out along with the 3D-solid for its geometric profiles, surface quality, and material specification, in-service condition and so on. Design exercise involves the choice of parting surface, allowance for shrinkage, draft, gate location, venting, thermal balancing of mould and choice of ejection system. The software is designed based on the knowledge of scientific and engineering principles and advanced mathematics. It is not based on intuition or guess. Therefore mould making, using the software is not just an art and but it has become an applied science.

Technology demands a good understanding of the fundamentals of physics, injection moulding process and computer skill in handling CAE and CAD. It may be difficult to combine these skills in one person to begin with. It is knowledge centered technology and demands teamwork and new work culture. For any design to be successful, the designer should have knowledge of different types of plastic materials used in the injection moulding process, their properties and the machine suitable for each of them. The forthcoming section of the review of literature deals with widely used plastic materials, machinery used for processing of components and mould design considerations.

1.2 PLASTIC MATERIAL

The selection of plastic material depends on the application of the product. The most commonly used plastic raw materials in injection moulding processes are Polypropylene (PP), Polycarbonate (PC), Polyvinyl chloride (PVC), Acrylonitrile-Butadiene-Styrene (ABS), Polystyrene (PS), PA-66 (Polyamide), etc. Among these, only a few, namely PC, PVC, PP, PA, ABS and Nylon use in automotive components due to their specific characteristics and strength.

In this work, the plastic material used is Nylon-6/6 (13% glass filled) manufactured by Dupont. Zytel is the material specified material by the customer.

1.2.1 ZYTEL 70G13HS1L NC010, BLACK

Generic Class:	Nylon 6/6, 13% Glass filled
Trade name:	ZYTEL 70G13HS1L
Applications:	Automotive (Gears, Bearings and equipments requiring high strength and durability, air intake manifolds, various types of covers, throttle bodies, fasteners, ski bindings, switch gears circuit breakers, etc.)

Injection Moulding Processing Conditions:

Drying: Nylon resins are hygroscopic and drying is required prior to processing. Suggested drying conditions are 80-90°C for a minimum of 4 to 6 hours. Resin moisture content should be less than 0.1%.

Suggested melt temperature: 280-305° C (Aim: 290°C).

Mould temperature: 0-95°C can be used recommended is 70°C (Mould temperatures control the gloss Properties; lower mould temperatures produce lower gloss levels).

Injection pressure: 35-140 MPa. Parts should be moulded at the highest practical injection pressure.

Injection speed: Moderate - high.

1.2.2 CHEMICAL AND PHYSICAL PROPERTIES OF NYLON

The polyamides are a group of polymers characterized by a carbon chain with -CO-NH- groups interspersed at regular intervals along it. They are commonly referred to by the generic name *nylon* and may be produced by the direct polymerization of amino acids or by the reaction of a diamine with a dibasic acid. Different nylons are usually identified by a numbering system, which refers to the number of carbon atoms between successive nitrogen atoms in the main chain. Polymers derived from amino acids are referred to by a single number; for example, nylon-6 is polycaprolactam. Polymers derived from a diamine and a dibasic acid are given two numbers with the first referring to the number of carbon atoms contributed by the diamine and the second referring to the number of carbon atoms supplied by the dibasic acid. Thus Nylon 6/6 is derived from hexamethylenediamine and adipic acid to give the structure shown in Figure 1.1.

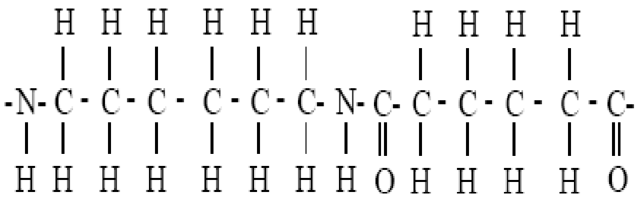


Figure 1.1 Nylon 6/6

The general properties of Nylon are presented in Table 1.1.

Mechanical Properties		**Conditions** State 1	**ASTM**
Flexural Modulus (MPa)	4830 - 5175	23 ºC	D790
Tensile Strength (MPa)	104 - 118	at break	D638
Flexural Strength (MPa) at yield or break	190	dry (0.2% water content) D790	
Flexural Strength (MPa) at yield or break	104	50% relative humidity	D790
Elongation at break (%)	3 - 5		D638
Hardness	95	Rockwell M	D638
Izod Impact (J/cm of notch) 1/8" thick specimen unless noted	0.5 - 0.6		D256A
Thermal Properties		**Conditions** **Pressure**	**ASTM**
Deflection Temperature (ºC)	257	0.46 MPa	D648
	233-244	1.82 MPa	D648
Physical and Electrical Properties		**Conditions** **State**	**ASTM**
Specific Gravity	1.21 - 1.23		D792

Water Absorption (% weight increase)	7.1	Saturated	D570
	1.1	after 24 hrs	D570
		Conditions	
Processing Properties		**State**	**ASTM**
Melting Temperature (ºC)	257	T_m, crystalline	
Processing Temperature (ºC)	272 - 299	Injection moulding	
Moulding Pressure (MPa)	49 - 138		
Compression Ratio	3 - 4		
Linear Mould Shrinkage (cm/cm)	0.005 - 0.009		D955

Table 1.1 General Properties of Nylon

1.3 LITERATURE SURVEY

1.3.1 Core and Cavity extraction

J.Y.H. Fuhetal [1] recommended 6 steps for parting design module in the 3D modeling.

1) Determination of the parting direction – Direction of the core and cavity open are the parting direction.
2) Recognition and patching the ‘through’ hole – if there is hole in the product, must indicate the parting location for the hole and determine the parting surface for these hole.
3) Determination of parting lines and the extruding direction – Parting line is the intersection between the two groups surfaces there are core and cavity.
4) Generation of the parting surfaces – a surface are mating surface of the core and cavity. Usually the parting surface as splitting surfaces to separate the mould into two block, cavity and core.
5) Creation of containing box – Containing box is used to create core and cavity. The size of the containing box is based on the dimension of the object, strength mould, moulding parameter.
6) Generation of mores and cavities – the box will be split into two mould halves, the core block and cavity block. The empty space inside the containing box is the place where the molten plastic will ne inject and solidify. The flow chart of parting design module is as shown in Figure 1.2.

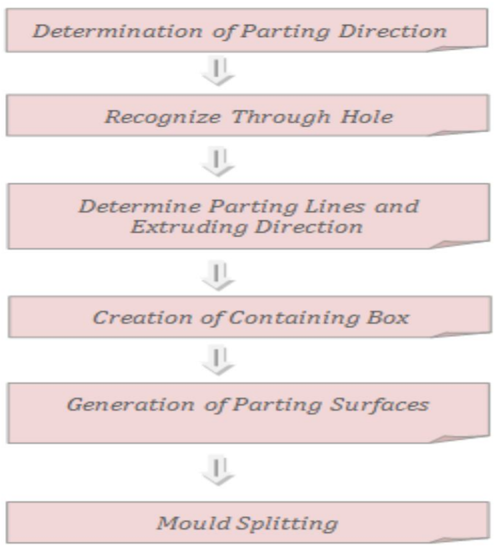


Figure 1.2 Parting Design Modules [1]

1.3.2 Runner Balancing for Multi cavity Moulds

Kevin Alam et al [2] suggested on optimization of runner balancing in multi cavity moulds. For runner systems with equivalent secondary runner diameters and lengths, cavities farthest from the injection point will experience greater shrinkage because the longer path length causes an increase of pressure drop. Accordingly, the traditional approach to runner balancing reduces the runner diameters of the closest cavities and increases the runner diameters of the furthest cavities to balance the pressure drops among them. A larger primary runner diameter also reduces pressure drops to the furthest cavities. Such designs minimize differences in shrinkage among the cavities and decrease the effects of variation on the runner balancing process.

When runner length was manipulated to achieve runner balancing, it became desirable to increase the length of the runners closest to the injection point to increase the pressure drops to these cavities, while decreasing the lengths of the runners furthest from the injection point to decrease the pressure drops to the furthest cavities. Both strategies help balance the pressures among the cavities, which reduce the need to increase the diameters of the primary runner and of the furthest secondary runners to achieve that balance. This ultimately reduces the runner system volume and associated material costs. Thus, the manipulation of the runner lengths and diameters is more effective in balancing the runner system than the manipulation of runner diameters alone.

1.3.3 Cooling Channel Design and Gate Design

D.E. Dimla M. et al [3] reported that conformal cooling channels in injection moulding design are advantageous then straight cooling passages. His Results from an investigation of the effectiveness of conformal channels through the construction of three different moulds with and without conformal cooling showed that the latter technique led to significant improvements and a general reduction of the cycle time while ameliorating heat transfer. According to this study, as with most manufacturing fields, production time and costs are strongly correlated. The longer it takes to produce parts the more are the costs, and with injection moulding production industries cooling time is often taken as the indicator of cycle time. Improving cooling systems will reduce production costs. A simple way to control temperature and heat interchange is to create several channels inside the mould where a cooler liquid is forced to circulate. Conventional machining like CNC drilling can be used to make straight channels. Herein, the main problem is the impossibility of producing complicated channels in three-dimension, especially close to the wall of the mould. This produces an inefficient cooling system because the heat cannot be taken away uniformly from the mould and the different shrinkage causes warpage and cooling time increase. On the other hand, if the cooling channels can be made to conform to the shape of the part as much as possible, then the cooling system the cycle time can be significantly reduced with cooling taking place uniformly in all zones.

Furthermore, if the part is ejected with the same temperature in every point the subsequent shrinkage outside the mould is also uniform and this avoids post-injection warpage of parts. Another advantage is that a mould equipped with conformal channels reaches the operation temperature quicker than a normal one equipped with standard (or drilled) cooling channels. In this way one can reduce the time required when the moulding machine is started. When the polymer is injected, it solidifies immediately touching the wall of the mould. If the volume of the part is sufficiently big and its thickness is too small, polymer solidified can obstruct the flow and hinder a complete filling of the cavity. In this case the mould must be heated to a particular temperature in order to permit the polymer to flow. Despite all these advantages it may be noticed that new technologies involved in the production of moulds with conformal channels can increase initial costs for the additional complexity of the construction process.

Charles Shaw et al [4] reported that the melts front moves through the mould, molten plastic contacts with wall of the mould and immediately begins to cool and solidify. Thus the first resin to contact the wall is further along in the cooling process at the end of filling than the last resin to be injected. In additional, thicker portions of the mould cool slower than thinner portions. This difference in cooling time aerates uneven volumetric contraction in the part, which lead to warpage of the plastic product. Thus gate point should always be provided at the higher thickness of the part. If the part does not design properly, the warpage to the product will exceed the required tolerances, the produced part is fail.

1.3.4 Manufacturing and Process Planning

Shaw C. Feng et al [5] reported that the process planning is a key activity for designers to evaluate manufacturability and manufacturing cost and time in the early product development stage plastic injection mould industry. So it is important to integrate the design and process planning message exchange to save time and cost of development.

Z.lou et al [6] suggested the new procedure for the development of the mould. The procedure should implement through an integrated, knowledge-based system to improve mould design efficiency and shorten design time. Normally, the conventional designer will separate the mould design are divide into three stages: detailed design of the mould, design of the mould-base and ordering the mould base from the manufacturer. In the new mould development process was purpose by X.Ruan, firstly the design mould-base according to dimension of the product. Secondly, begin to design the detail of the mould while ordering the mould–base from the manufacturer. The step of the design the detail of mould and ordering the mould-base are carry out at the same time, enable to shortening the product development cycle.

Wong Choon Tat et al [7] Injection moulding is where the molten plastic fill into the mould (desired shape) so the Injection of plastic into the mould has been categories into 3 phase: Filling Phase, Pressurization Phase, and Compensating Phase. *Filling Phase* – this phase where the molten material are filling the cavity of the product inside the mould. The molten material contact to the wall of the mould will solidify very fast than other area. *Pressurization Phase* – Even all the cavity is filled with the molten material but at the corner or edges, or the angle may not fulfill with the molten plastic because of the path is too narrow, so extra pressure and material need to complete the fullness of the cavity. *Compensating Phase* – Plastic is a high shrinkage material. From the molten phase to the solid phase, it will shrinkage around 25% after the pressurization phase.

S.H. Tang et al [8] suggested that, the sequence of manufacturing of mould by following steps can reduce time and helps in easy of manufacturing. According to him, the sequences of machining processes like drilling, milling and tapping are important to get a good quality of work and also save the machining time. In machining process, there are some steps that should be highlight and pay attention. For drilling process, especially for cooling channel, the drilling machine that has been used is radial drilling machine instead of vertical drill press. This is because the spindle speed for radial drilling machine is controlled by gear while the vertical drill press is used belt. Thus, the radial drilling machine can produce higher drilling torque as compared to the vertical drill press.

Suqin Yao [9] suggested that process sequence is divided into two principles-

Principle 1: The setup sequence must be arranged according to the sequence of design features. Principle 2: The setup sequence must be arranged according to the feature's pre-defined process sequence.

The process sequence should always follow the basic manufacturing principle, there is doing rough cuts first, and semi and finish cuts in a prescribed order.

By literature review[1-9], it can be understood that, runner balancing in multi cavity mould design depends on length and diameter rather than diameter alone and how these two should be manipulated for effective balancing. Many authors have reported the steps to be followed for core and cavity extraction. There is general consensus that conformal cooling is more effective than straight cooling system and gate point provision. Mould flow analysis which helps to validate mould design also suggests the possible defects in the mould and manufacturing and process planning aspects suggest that the sequence should be followed for manufacture of mould in order to reduce machining time and cost.

1.4 ORGANIZATION OF THE REPORT

CHAPTER 1.INTRODUCTION: deals with introduction to injection moulding process and literature survey for various techniques involved in mould design.

CHAPTER 2.OBJECTIVES AND METHODOLOGY: presents the statement of the problem, objectives of the project work and methodology followed in carrying out the project work.

CHAPTER3.MOULD DESIGN: describes mould design procedure, design considerations and design calculations.

CHAPTER 4.MOULD FABRICATION, ASSEMBLY AND TRY OUT: explains mould manufacturing steps, process sequence assembly of mould and mould tryout.

CHAPTER 5.COST ESTIMATION: in this chapter, approximate mould cost is calculated by considering raw material cost, processing cost etc.

CHAPTER 6.CONCLUSION AND SCOPE OF FUTURE WORK: presents conclusions arrived from this work and future scope for this project work.

CHAPTER 2

OBJECTIVES AND METHODOLOGY

2.1 STATEMENT OF THE PROBLEM

The work deals with the design and manufacturing of one plus one cavity injection mould for **"Rotor and Cover"** of a Two-wheeler part**.** Tail cover is employed as an aesthetic component capable of breaking the flow of air during vehicle movement. Nylon 6/6 (13% glass filled) material is used for the production of this component. The mould has to be designed to produce a good quality component considering the ease of manufacturability, assembly and positive ejection of the component.

2.2 OBJECTIVES OF THE STUDY

The main objectives of project are outlined as:

- To design a (one plus one) cavity injection mould for Rotor assembly. The mold to be designed to produce a quality component considering the ease of manufacturability, assembly and positive ejection of the component within the minimum possible time and cost.
- To generate detail drawings of the mould for manufacturing.
- To carry out manufacturing of mould as per drawing specifications.
- To perform mould try out and troubleshooting of defects.

The Scope of the work includes:

- Study of Component
- Concept Design
- Design Calculations
- Core and Cavity Extraction
- Preparation of Assembly Drawing
- Preparation of Mould Base Drawing
- Mould Manufacturing
- Mould Assembling and Trials

2.3 PROJECT METHODOLOGY

The basic concept involved in this method is to attain the objective of the systematic and correct mould design through a well planned approach. The methodology consists of the following as shown in the Figure 2.1.

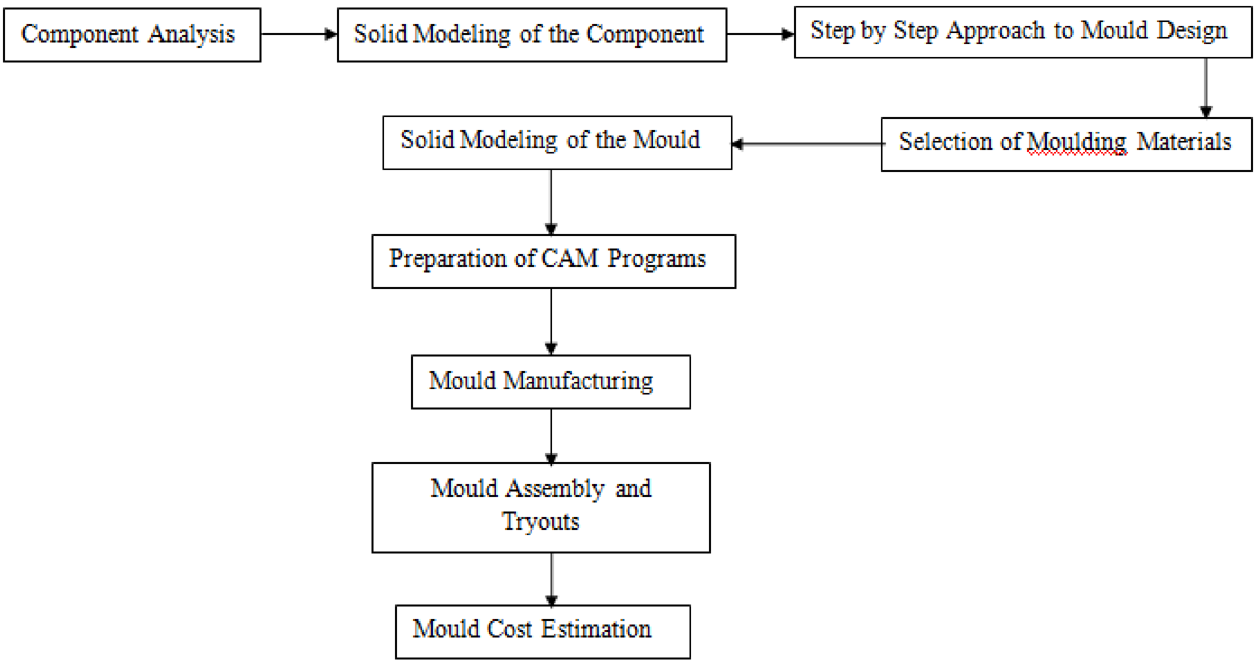


Figure 2.1 Project Methodology

2.4 COMPONENT ANALYSIS

2.4.1 Component description

The components Rotor and Cover are used in automotive two wheelers. Tail cover is employed as an aesthetic component capable of breaking the flow of air during vehicle movement. The cavity surface has an aesthetic value and hence it should have glossy finish and no visible marks are allowed on the surface. Isometric view of the component is shown in Figure 2.1. No Weld lines, sink marks or other visible defects are permitted in the component moulding. Surfaces must be smooth and clean.

2.4.2 Important Features of Rotor and Cover

The average length, width and height of the components when placed or oriented in the draw or moulding direction is 50 x35 x19 mm. Maximum Allowable draft is 1deg. Components weigh 5.9gms without runners and gates. As per the customer requirement, the mould will be of two cavities (type of family mould). The components are provided with ribs to strengthen. The rotor component has undercut and hence a side core is provided for proper ejection of cover.

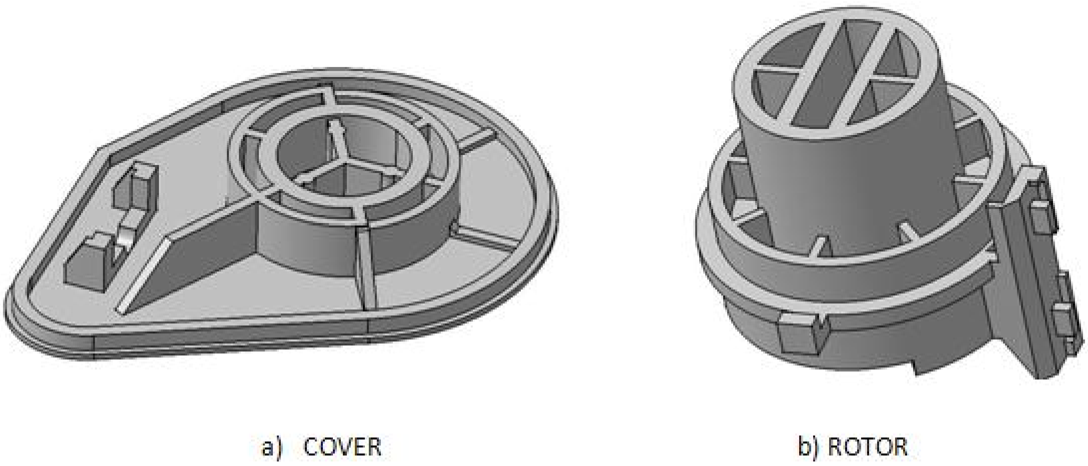


Figure 2.2 Cover and Rotor

2.4.3 Materials Specifications of Rotor and Cover

The component is made of Nylon 6/6 – 13% Glass filled conforming to the specifications given by Dupont Inc, USA. Chemical composition and mechanical properties are detailed in Table 1.1

2.4.4 Physical and Geometrical Data of Component

Physical and Geometrical data of component is detailed in Table A2.1 and A2.2 (Appendix).

2.5 SOLID MODELING OF ROTOR AND COVER

Components were modeled by employing 3D Geometry Software (CATIA). 3D Solid Model of the component created for the purpose is shown in Fig. 2.1 is used to determine the location of centroid, the mass, volume, surface area of the Component, parting surface data etc, projected area, length of strokes of side cores and lifters.

2.6 STEP BY STEP APPROACH TO MOULD DESIGN

- Study the component for its geometry, material and its properties and number off and rate to be manufactured etc.
- Selection of parting surface.
- Determination of number of cavities.
- Determination oflocking force required.
- Selection of Injection Moulding Machine.
- Design of feeding system.
- Design of core and cavity.

- Design of cooling system.
- Design of ejection system.
- Design of other parts of the mould.
- Selection of mould materials.

These are discussed in detail considering the components Rotor and Cover to be produced.

2.7 SELECTION OF MOULD MATERIALS

The criterion for the selection of mould materials for the mould depends on the following consideration. Mould steels are distinguished by the method and care exercised during manufacturing. Hardenable alloys of iron made with high level quality control may also be suitable for making the moulds.

There are four general classes of steel used by the mould makers: They are

- Low carbon steel (less than 0.20% carbon). This steel does not contain enough carbon to enable it to harden to any appreciable extent when heated and quenched.
- Medium carbon steel (0.20% to 0.60% Carbon).
- High Carbon steel (0.70 to 1.3% Carbon).
- Alloy steels containing various elements besides carbon, each element serving to contribute a definite property to the material.

The choice of mould steel depends on the type of plastic raw material used, life of the mould required and the ease of machining and maintenance. Usually pre-hardened steel is employed for general purpose moulds. Most often them not, the choice of the mould steel depends on customer specifications. Based on these criteria, P20 steel for core cavity and C45 for Core and cavity holders have been considered.

2.8 SOLID MODELING OF THE MOULD

This was carried out using 3D Geometry Software (CATIA). This software is having moulds to perform the operations such as extrude, cut, sweep etc. The Software also has moulds to assemble parts modeled with suitable constraint. Creating the assembly helps to check a prior possibility of interference between mating parts, create 2D drawings which are enclosed at the end.

2.9 MOULD FABRICATION

Methods of moulds fabrication are given in Chapter 4.

2.10 MOULD ASSEMBLIES AND TRYOUT

This is discussed in Chapter 4.

2.11 MOULD COST ESTIMATION

Mould cost estimation is also one of the integral parts of this methodology which discussed in Chapter 5. The estimated mould cost and material cost can be calculated from the Table 5.3. Mould cost has to correspond with the life cycle, production, productivity and quality aspects in the manufacture of components and hence assumes significance in the mould design exercise.

CHAPTER 3

INJECTION MOULD DESIGN

In this section, step by step approach to the design of Plastic Injection Mould based on experience, empiricism and expertise as applied to “Tail cover” and the various design calculations are emphasized.

3.1 COMPONENT DETAILS

Material	:	Nylon 6/6 – 13% Glass filled
Trade name	:	Zytel
Density	:	1.23×10^{-3} gm/mm^3
Volume	:	(1877+2465) mm^3 = 4342 mm^3
Mass	:	0.00534 kg (5.34gms)
Number to be manufactured	:	500 / month

3.2 SELECTION OF PARTING PLANE

The selection of parting plane entirely depends on the shape and geometry of the component. After careful study of the component drawing, solid 3D model and component sample, a profile parting plane is chosen. This permits easier ejection of the moulding. Figure 3.1 shows the parting plane and feed system.

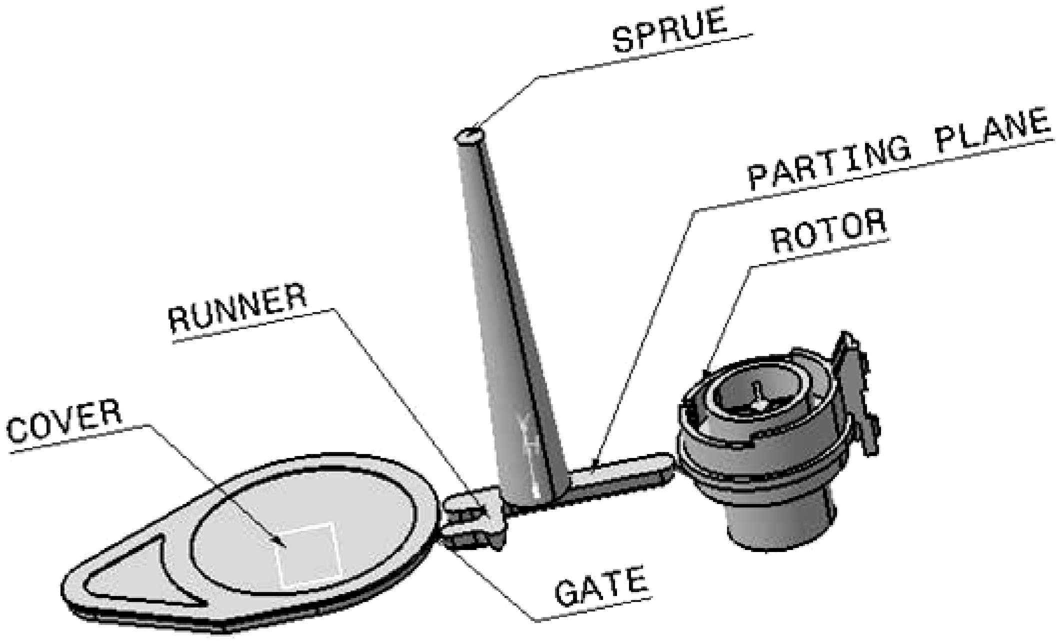


Figure 3.1 Parting Plane and Feed system

3.3 DETERMINATION OF NUMBER OF CAVITIES

While deciding the number of cavities, geometry of the component is taken into account. The approximate dimension of Rotor is 22 x 25 x 25mm and Cover is 6 x 35 x 51 mm respectively. The shot weight, locking force, injection pressure and number of components to be produced from the mould, about 500/month, need to be considered. The mould is to be loaded on DGP Windsor 80 tonnes machine at the customer’s end and the trial is to be taken on the same machine. The customer’s specification for the mould was 1+1 cavity (2 cavities) mould. By considering all these aspects 1+1 cavity mould and is a family mould has been considered in this work.

3.4 DETERMINATION OF LOCKING FORCE [11, 12]

Locking force is the force required to keep the mould halves together when injection takes place. Locking force (LF) is given by the formula

L.F required = (Projected Area of Component x n + runner + gate) x P x FOS…..…. (3.1)

Where projected area of the component from Table A2.1, A2.2 of Appendix is 1668.015 mm^2

P = 100.0 MPa or 1000 kg/cm^2

FOS = 1.25, n = 2

To calculate total projected area (Shot area) assuming 10% of projected area of component for runner and gate.

Total Projected Area = Projected Area of Component + 10% of Projected Area of Component

= 1668.015+ 1668.015x 0.10 mm2

Total Projected Area = 1834.81 mm^2

On substituting the values the Equation 3.1, we get

L.F required = 2.3 Tons≅ 2.5 Tonnes.

3.4.1 Shot Weight Required

Shot weight (Ws) required in kg is given by

Ws = Vs x ρ x C…………………………………………………….…. (3.2)

Where

s = Shot volume in mm^3

$\rho = 1.23 \times 10^{-3}$ gm/mm^3

C = 0.83 is a constant for amorphous material, which depends on the component weight

= Shot Volume of the Component + Volume of the Feed System (Vf)

∴ Shot Volume (Vs) = Vc + Vf, --- Vc from section 3.1 above.

= 4342 + 4342x 0.10 = 4985.2mm

∴ On substituting these values in the Equation 3.2, we get

Ws =5.9 gm

3.4.2 Shot capacity of the machine

Shot capacity of the machine is calculated using the formulae

Shot Capacity (S_M) = Swept Volume X Density of Material X C…………………. (3.3)

Swept volume = 120 cm^3 (Table 3.1)

Density = 1.23 gm/ cm^3

C = 0.83 for crystalline materials

Substituting in the above,

Shot capacity of the machine (M) = 122.508 gm

3.4.3 Plasticizing capacity of the machine

Plasticizing capacity of the machine is calculated using the formulae

Plasticizing Rate of Material = Plasticizing Rate of Polystyrene X Total Heat of PS

Nylon (g/hr) Total Heat of Nylon….. (3.4)

Plasticizing capacity of polystyrene (PS) = 11.1 gm/sec= 39960 gm/hr (Appendix)

Total heat of polystyrene = 57 cal/gm

Total heat of Nylon = 56 cal/gm

Plasticizing rate of material Nylon (g/hr) = 40000 g/hr = 40 kg/hr.

3.5 SELECTION OF INJECTION MOULDING MACHINE

As per the requirement of customer, the mould is designed for DGP Windsor 80 Tonnes Injection moulding machine. The clamping tonnage available on the machine is 80 tonnes, shot weight is 122.508 gm, and Plasticizing capacity 40 kg/hr, which are more than the actual requirements as presented in Section 3.4.

Table 3.1 shows the machine specifications for DGP WINDSOR.

MACHINE MODEL		**SPH 80**	**SPH 130**	**SPH 180**	**SPH 250**	**SPH 350**
INJECTION UNIT		230	664	664	1310	1930
Screw diameter	mm	34-42	45-50	50-60	60-70	70-80
Injection pressure	bar	1900- 1250	2200- 1800	1800- 1250	2100 1570	1763- 1350
Stroke volume	cc	120-185	300-375	375-535	678-924	1078-1407
Screw stroke	mm	135	190	190	240	280
Shot Weight(PS)	gm	108-167	270-338	338-482	610-831	969-1266
Injection rate	cc/sec	163-249	175-217	287-413	312-425	374-488
Plasticizing capacity (PS) @ max. screw speed	g/sec	14.5- 21	23- 26	36- 47	43- 62	68- 85
L/D ratio	-	22-17.8	22-19.8	22-18.3	20.4-17.5	21-18.4
Screw speed (max.)	rpm	245	165	230	180	190
Heating capacity	kW	7.6	12.8	14.1	22.4	26.7
No. of heating zones	-	4	5	5	5	6
Nozzle contact force	kN	58	60	60	84	106

LOCKING UNIT

Closing force	kN	800	1300	1800	2500	3500
Mould opening stroke	mm	450	550	550	750	1000
Min. mould height	mm	200	200	200(350)	250	300(500)
Max. daylight	mm	650	750	750(900)	1000	1300 (1500)
Distance between tie bars	mm	385 x 385	440 x 440	520 x 520	640 x 640	700 x 700
Size of mould plates (hxv)	mm	590 x 590	650 x 660	770 x 770	970 x 970	1020 x 1035
Ejector force	kN	42	42	75	75	95
Ejector stroke	mm	80	95	120	240	200

GENERAL DATA

Pump drive	kW	18.5	22.5	30	37	45
Total connected power	kW	26.1	35.3	44.1	59.4	71.7
Overall Dimension (L x B x H)	mm	4650 x 1350 x 1950	5160 x 1380 x 1960	5800 x 1500 x 2030	7240 x 1530x 2330	8850 x 1630 x 2470
Weight	tons	4.6	5.9	8.2	10.7	15.8

Table 3.1 Machine specification

3.6 DESIGN OF FEED SYSTEM [11]

Runner Type: In this design modified Half Round runner is employed to feed the component.

3.6.1 Runner Design

The runner diameter is calculated by the following formulae.

$$D = \frac{\sqrt{W} \times \sqrt[4]{L}}{3.7} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots (3.5)$$

Where W= weight of the moulding = 5.9 gm (From 3.2)

L= Length of the runner =20 mm

Substituting the values in equation

$$D = \frac{\sqrt{5} \times \sqrt[4]{20}}{3.7}$$

Dia of the runner (D) = 1.22 mm

The half round diameter of the runner d= √2 X D

D= 1.22 mm. Therefore for half round gate, diameter, d = 1.72 mm

But in the present design trapezoidal runner is used. Figure 3.2 shows the runner cross section.

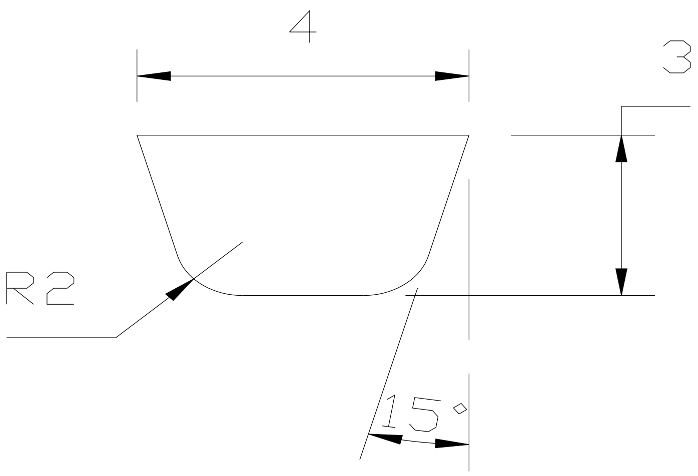


Figure 3.2: Runner Cross Section

3.6.2 Gate Design

In this design single point feeding with submarine gate is employed to feed the components. The length is taken as 2mm; the diameter of gate is taken as 0.5mm with an included angle of 8^o and at an angle of 45^o with the horizontal to get high shear rates.

3.7 DESIGN OF CAVITY AND CORE INSERTS [12]

In mould design, dies (core and cavity inserts) are called as heart of the mould. Therefore design should be economical to manufacture and should give good service.

3.7.1 Strength of Cavity and Core Insert

Considering the component shape a Rectangular cavity insert is considered to achieve ease of manufacturing. For design purpose, the maximum allowable deflection of 0.025-0.05 mm is considered. The approximate thickness of side wall required is calculated from the following formulae.

$$t = \sqrt[3]{\frac{12 * P * L^4 * A}{384 * E * b * \delta}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots (3.6)$$

Where

Y = Max. Deflection of side walls (Cm.) = 0.005 cm

L=max. Length of the component (49mm)

A=Pocket depth. Max dept of the component(20mm)

P = Max. Cavity pressure (85% of the injection pr) =850 Kg/Cm^2

b=Cavity plate thickness (20+25) From GTTC std. 46mm

E = Modulus of elasticity for steel (2.1 X 10^6 Kg/Cm^2)

Substituting these values in the above equation

t = Minimum thickness of Cavity insert wall = 0.86cm =9mm is maintained as show in the drawing of the cavity and the core insert.

3.7.2 Core Insert

The minimum wall thickness for core insert is same as the cavity insert i.e. 9 mm is maintained as shown in the drawing.

3.8 SOLIDIFYING TIME [12]

The solidifying time is proportional to square of the wall thickness.

$T = \rho . a. \ \delta^2/8 \lambda (Tmatl – Tmould)$ ………………………………………………… (3.7)

Where

ρ = Density of the plastic material = 1.23 gm/cc

a = Heat content of Nylon = 56 cal/gm

δ = Average wall thickness of the moulding = 0.13cm

$\boldsymbol{\lambda}$ = Thermal conductivity of the Nylon = $6x10^{-4}$ cal/cm. Sec C

Tmatl = Injection temperature of the melt = 290° C

Tmould = Temperature of the mould = 70 ° C

Substituting the values in the equation 3.7, we get

$T = 1.23 X 56 X 0.13^2/ (8 X 6X10^{-4} (290 – 70))$

Solidifying time (T) = 1.3 sec

3.9 CYCLE TIME ESTIMATION [11]

3.9.1. Injection time

Injection time of material into the feed System (Cavity Runner and gate) is equal to fill time. From practical considerations it is approximated to 2.4 sec

3.9.2. Solidifying time is 1.3 sec (from 3.8)

3.9.3. Mould opening and closing time = 2.0 td

Where td = Machine cycle time = 5 sec (Experimentally verified).

= 2 X 5 =10 Seconds

3.9.4. Part and feed system ejection time:

Normally in two plate mould system the part is ejected with in 5sec, since the feed system is being operated manually.

Therefore the approximate Ejection time =16 sec

3.9.5. Mould lubrication time (After ejection of component, mould has to be lubricated)

Lubrication Time (TL) = 3 + (ns + nc) + (nc – 1)

Where nc - number of cavities = 2

Ns - Number of side cores = 1

$\therefore$ Lubrication time = 3 + (1 + 2) + (2 – 1)

= 8 sec

Cycle time = Fill time + Solidifying time + Mould opening. And closing time + Ejection time + Lubrication time……………………………………………………..…………………………………………… (3.8)

Cycle time (Ct) = 2.4 + 1.3 + 10 + 16 +8 = 40 sec

Therefore, No .of shots / hr = Total time / total cycle time = 3600 / 40 = 90 shots / hr.

3.10 DESIGN OF COOLING SYSTEM [12]

As the geometry of the component is very small the amount of heat dissipated is also very small, and the surface area of the mould is sufficient to handle the dissipation of the heat generated.
Hence no cooling circuit has been provide in the inserts , but plate cooling provided is sufficient for the dissipation of the generated heat.

3.11 DESIGN OF EJECTION SYSTEM [12]

For the components pin and blade ejection is chosen along with the lifters according to the features in the component.

Ejection force required to eject the component is determined by using the formula.

$P = S \times t \times E \times A \times \mu / d\ (d/2t - d\gamma/4t)$...…………………….………………………..….. ……………… (3.9)

Where

P = Force required, N

t = Average thickness of component = 0.13 cm

E = Elastic modulus of Plastic=1000kg/cm^2

A = Total contact area between mould and mould faces in line of draw =16.68 cm^2

γ = Poisson's ratio of the Mould steel = 0.3

μ = Coefficient of friction between plastic and steel = 0.5

d = Circumference of moulding surface in ejecting the male core = π X 0.6 = 3cm

S = Thermal contraction of alloy across the circumference, 'd'

= Co-efficient of thermal expansion x temperature difference between

Softening point and ejection temperature (^{0}c) x d (cm)

= 13 X 10^{-5} X 160 X12.97 =0.269

On substituting the above values in the equation 3.9, we get

Therefore, Ejection force required = 5 kgf

Ejection force required in tones = 5/1000 =0.005 Tons

The ejection force required to eject the component is within in the capacity of 80 tonnes M/c. Therefore the design is safe.

DME standard mould set has been considered since it reduces the cost of production and can be easily procured from the manufacturer.

Ejector Plate

Ejector plate for mould depends on the size of the component, number of ejector pins, size and position and also depends on return pin position and ejector guide pillar and bush.
Length of the ejector plate = 246 mm (width of core housing)

Width of the ejector plate = 122 mm

Thickness of the ejector plate = 12mm

Ejector back plate

Length and width of the ejector back plate is similar to ejector plate and thickness is taken as 16 mm 4 mm more than the ejector plate according to standards to prevent bending. Ejector back plate facilitates accommodation of ejector guide pillars and guide bushes, retractable rods.

Design of Spacer

Length of the spacer = length of core housing = 246

Width of the spacer = (Width of core housing-width of ejector plate)/2 – 1mm for clearance

= [(196-122)/2]-1 =36 mm

Ejection stroke = This Stoke length depends on the stroke of lifter which is 110 mm+10mm.

= 110 + 10mm = 120 mm

Thickness of spacer = Ejection stroke + Ejection plate thickness + Ejector back plate thickness + Thickness of rest button = 33+12+16+5 = 66 mm

Therefore, Dimensions of spacer are 246x36x66 mm (2 Nos. are used here).

3.12 DEFLECTION OF CORE HOUSING [11]

Due to the high injection pressure acting on the mould cavity, stresses are developed in the mould blocks leading to mould deflection. The recommended maximum deflection for core housing is 50 microns. According to the hydrostatic principles, the pressure acts equally in all directions. This force develops a bending stress in the moving half of the mould.

For a simply supported beam, we have deflection ‘δ’.

$\delta = WL^3/384EI$………………………………………………………………..……………………… (3.10)

Where

L= Distance between spacer = 124 mm

E = Young’s modules for steel = 2.1×10^5 N/mm^2

I= Moment of Inertia = $bd^3/12$

b= length of core housing = 246 mm

d= depth of core housing = 82 mm

For 80 tonnes M/c only 75% of its total capacity is available. (25% of its capacity is taken as loss due to wear tear, leakage etc.)

$\therefore$ W = 60 X10^3 Kg = 600X10^3 N

I = $bd^3/12 = 246 \times (82)^3/12 = 11.3 \times 10^6$ mm^4

On substituting these valves in the equation 3.10, we get

$\therefore \delta$ = 0.13 microns < 50 microns. Hence design is safe.

3.13 DESIGN OF GUIDE PILLARS AND GUIDE BUSH [11]

Guide Pillar

The pillar diameter is given as follows.

Side thrust (θ) in N= ahP_f

Where

a = Maximum side of core = 4.7cm

h = Height of core = 8.2 cm

P_f = Cavity pressure = 400 kg/cm²

On substituting these values in Equation above

θ = 15416

Working diameter (d) of guide pillar $\geq\sqrt{4\theta/\pi Nfs}$
Where N = Number of pillars = 4
fs = Shear stress = 1600 kg/cm²and
d> 17.5 mm
∴ Working diameter of pillar is taken 20 mm for safe design.

Stem diameter of guide pillar = working diameter (d) + wall thickness (2 mm)….. (3.11)

= 20 + 4 X 2 =28 mm

Collar diameter = stem diameter of guide pillar + minimum thickness (2 mm)…… (3.12)

= 28 + 2 X 2 =32 mm

All the above calculations are based on the theoretical assumptions. However, in the practice design of injection mould, we shall have a significant deviation from the above said values. However, these values can be used as a basis in to which the other deviations in the practical design can be added.

Design of return pin and stopper Pin, rest buttons, support block

Length of return pins = core housing thickness + ejection stroke + ejector plate + 0.2……………………………………………………………………………..….. (3.13)

= 82 + 25 + 12 =119

Four return pins of Φ 38 are used here refer drawing.

Rest Button
Four rest buttons of DME standard STD-DME-2025-EB-36-46/1/66 are used, and fitted on to the ejector back plate.

DME standard mould set has been considered as per the customer requirement.

Figure 3.3 and 3.4 shows the assembled view of core and cavity half.

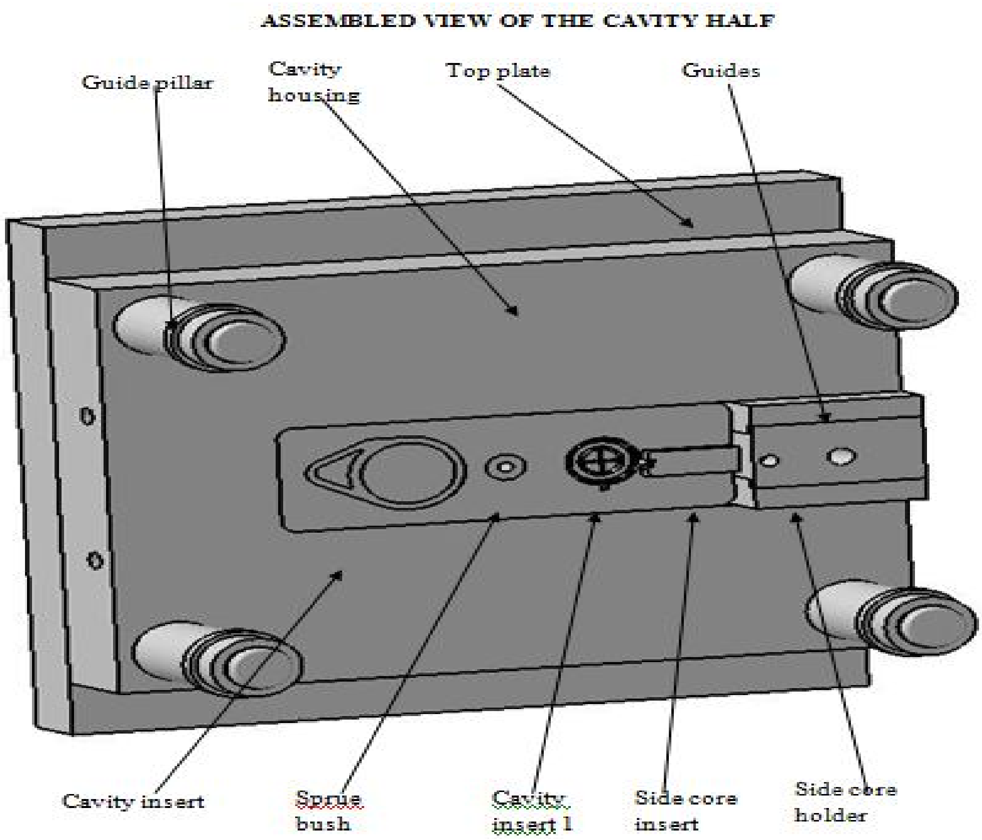


Figure 3.3 Assembled View of Cavity half

ASSEMBLED VIEW OF THE CORE HALF

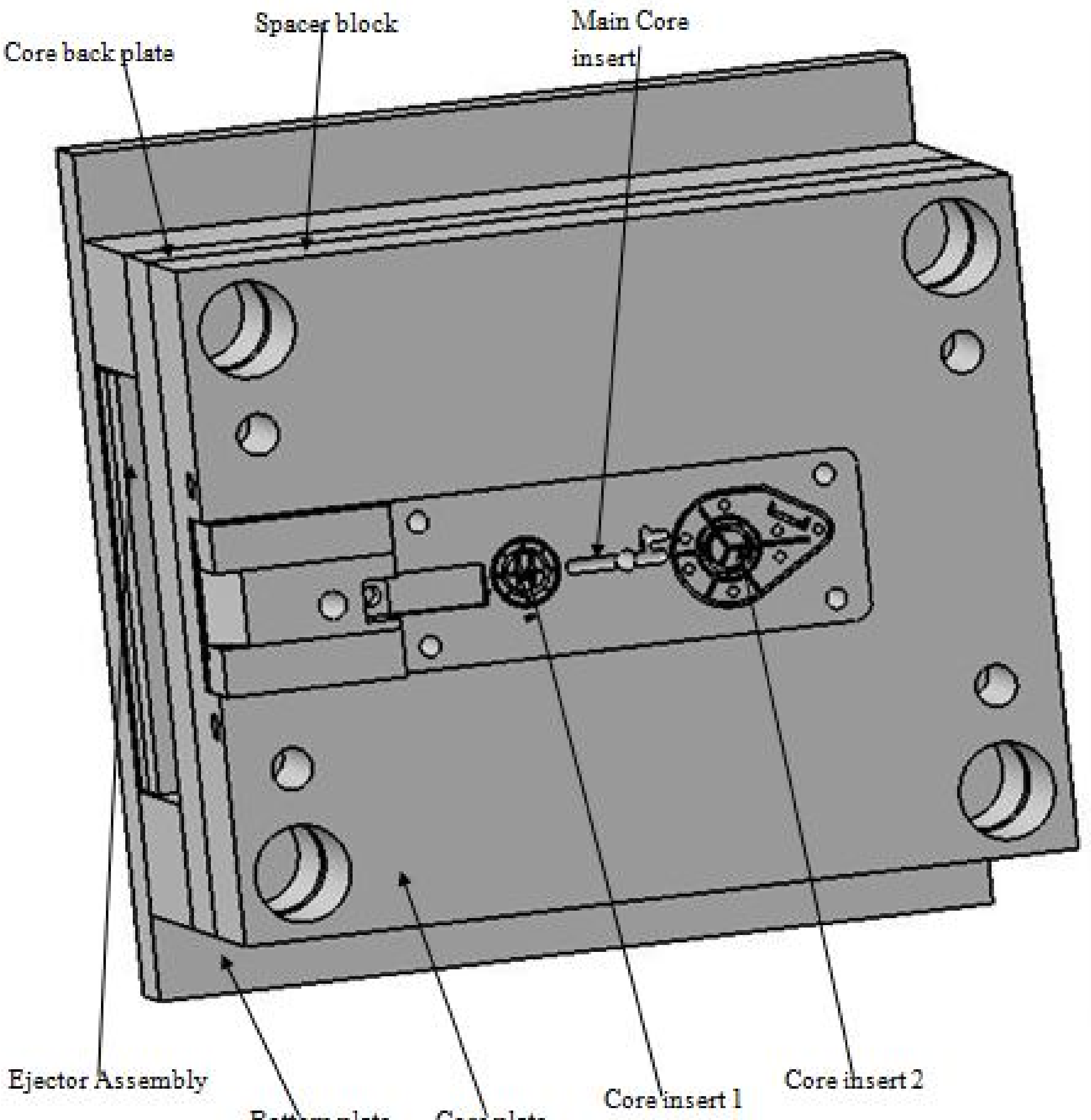


Figure 3.4 Assembled View of Core half

CHAPTER 4

MOULD FABRICATION, ASSEMBLY AND TRY OUT

4.1 MOULD FABRICATION

Raw material is transformed to the finished mould during fabrication. Each part of the mould is manufactured by referring to their respective design drawings. Before taking up fabrication, the mould drawings are studied and a process plan for each part is prepared. Attention is given on machinablity during design stage itself. Tolerance and hardness required and related aspects are selected for implementation.

During mould fabrication, the following procedure was followed:

4.1.1 Process planning

4.1.2 Electrode planning, and

4.1.3 Manufacturing of the mould

4.1.1 PROCESS PLANNING

Systematic determinations of the detailed methods by which parts can be manufactured economically from initial to finished stages are taken up. The process sequence is determined for a particular part based on the availability of specific machines. Plans are presented in the form of process sheets, giving sequence of operations, machine mould employed, and an estimate of time for each operation and related details.

4.1.2 ELECTRODE PLANNING

EDM is a chip less manufacturing operation, the metal removal by electric discharge between a shaped electrode and the electro-conductive work piece in the presence of di-electric liquid. Electric spark is used to erode the work piece, which takes the shape opposite to that of electrode.

EDM simplifies the machining of complicated shapes especially when there is a need for accuracy in the manufacture of die, which is in a hardened state.The electrode is of a smaller size than the profile required in the die; about 0.2 to 0.5mm per side to account for spark gap. Spark gap of 0.2mm per side for smooth finishing and 0.5mm per side for rough finishing is recommended. Copper is used as an electrode in this work.

4.1.3 MANUFACTURING OF MOULD

According to the process plan presented in the form of process sheets, manufacturing of each components are carried out. Here manufacturing of cavity plate is detailed. Following machine operations are carried out.

- **Pre-moulding**

Raw material is cut to overall profile considering machining allowances. This operation is carried out generally by sawing and flame cutting.

- **Surface Grinding**

After pre-moulding, cleaning of all the sides and maintaining right angles, the block is further smooth machined on a grinding machine.

- **Conventional machining**

According to drawing and process sheet, the raw material is rough machined to the geometric features and size of the part by providing sufficient allowance for finishing. Conventional machining is generally carried out on a lathe, milling and shaping machines.

- **CNC Machining**

Features such as pockets, profiles, which require close tolerances, are machined in this stage.

- **Stage Inspection**

After each operation stage inspection is carried out to examine geometric accuracy of the profiles.

- **Bench Work**

Drilling and tapping are carried out using drilling machines. Finishing, corner radius, cavity profiles are completed at this stage.

- **Heat treatment**

No Heat treatment for P20 it is required.

- **Sparking and Wire cutting**

Features that cannot be machined by conventional machine are realized during this process. This is done after required hardness is achieved.

- **Finishing**

This is the last stage in manufacturing carried out on a Grinding, Polishing or Lapping machines.

- **Final Inspection (CMM)**

Co-ordinate measuring machine is used to check complex profiles which require close tolerance.

4.2 MOULD ASSEMBLY

Following procedure is followed in mould assembly.

- **Finishing the impression.**

To check for all the important parts after receipt from manufacturing stage, ensuring freedom from cutter marks and burrs are not present on the mould.

- **Fixing core and cavity inserts.**

After finishing the surfaces of cavity and core inserts these are assembled in core and cavity housing respectively.

- **Bedding down.**

This is the process of checking the moulding material escape between the two halves of die when the moulding material is injected under high pressure. This involves coating one plate using mould-maker blue. The two halves are brought together and where there are high spots on the plate cavity, plate blue will be picked up. Scraping and filing are employed to remove these spots. The two halves are brought together and joined at mould interface; 0.15 mm feeler gauge is used to check gaps.

- **Fixing the ejector system in the moving half.**

Ejector guide pillars are press fitted to the core housing. Ejector plate is fixed with guide bush for easy movement of plate. Ejector pins, return pins and fasteners are fixed accordingly.

- **Assembly of the fixed half.**

Cavity insert is fixed and fastened by using fasteners sprue bush and cooling sleeves are fixed accordingly.

- **Checking the water cooling circuit.**

Cooling holes drilled for water circulation are checked to ensure flow is unidirectional and leak proof

- **Aligning fixed and moving halves.**

Finally the fixed and moving halves are aligned by guide pillars and guide bushes.

4.3 MOULD TRYOUT

After the mould is manufactured and assembled, the mould is 'tried' to see that component made is true to the geometry and dimensions specified by the customer. Following procedure is employed.

- Check the Locating ring diameter with that of the hole diameter of the machine platen.
- Check the mould for free movement of ejector system.
- Suit the nozzle tip to sprue bush.
- Clean the mounting surfaces with lubricant.
- Check shut height and adjusts the machine by loosening the lock nut provided on tie bar.
- Align the mould both horizontally and vertically.
- Clamp the fixed half of the die.
- Assemble ejector rods and clamp the moving half.
- Start the machine and check for free movement of all functions of the machine and the mould.
- Slowly and uniformly heat the mould to the required preheating temperature.
- Start trial moulding using only low injection pressure and speed. Adequately lubricate the mould and check the moulding for drag marks, ejection marks and sink marks.
- Gradually increase the injection pressure and rate of inflow to the chosen process parameters and the mould is now ready for production.

4.4 SAMPLE PROCESS SHEET FOR CORE PLATE

Table 4.1 shows the operations carried out for the realization of the core plate.

SIZE OF RAW MATERIAL: 250X200X50		
Sl. No.	**OPERATION**	**MACHINE**
1	Pre moulding (Milling)	Milling
2	Surface grinding	Surface grinder
3	Spotting of holes	CNC
4	Drilling of holes	Drilling machine
5	Jig boring	Jig boring machine
6	Tapping of threads	Drilling machine
7	Counter boring	Drilling machine
8	Pocket/slot milling	Milling

Table 4.1 Sample Process Sheet for Core Plate

Process sheets to facilitate shop for manufacturing is shown in the figure as follows:

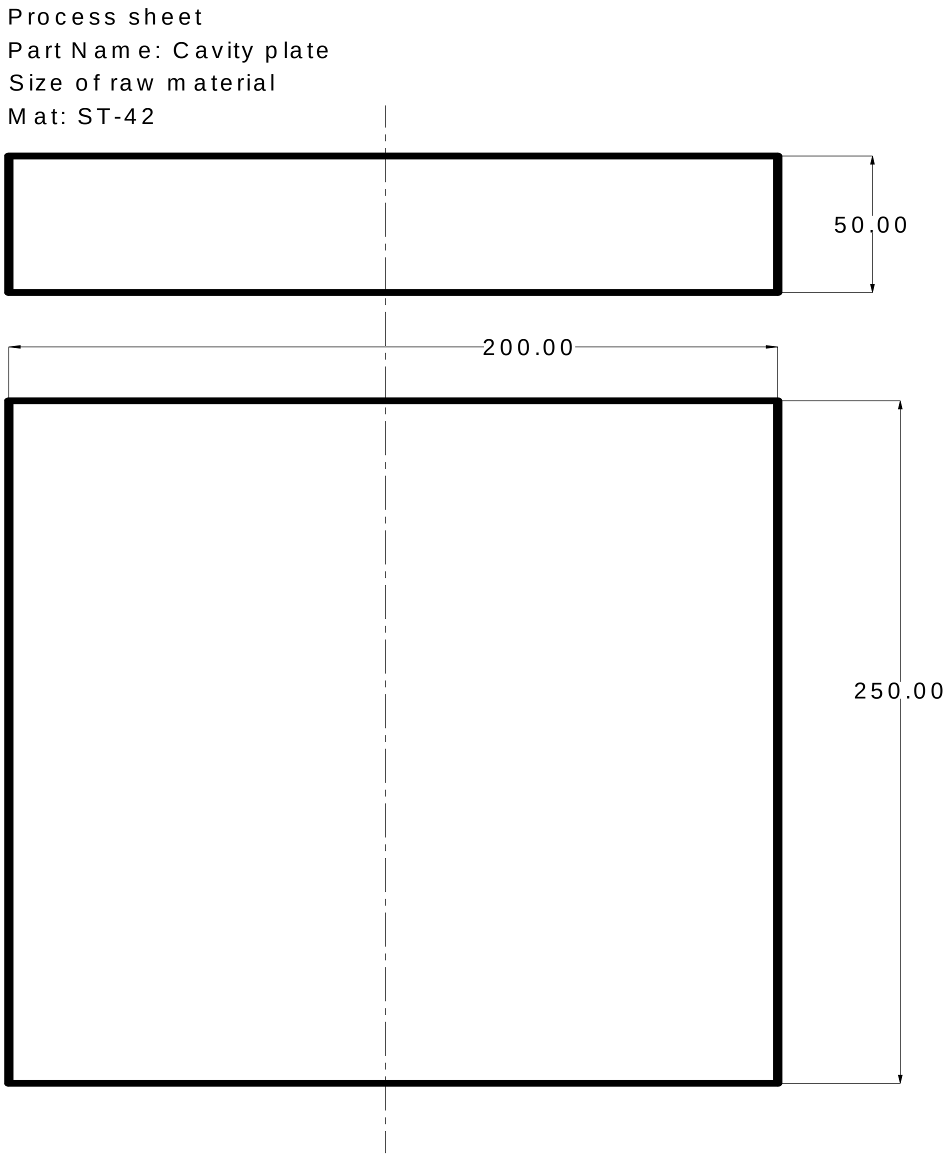


Figure 4.1 Size of Raw Material

Process sheet
Operation No.:01 (Pre tooling)

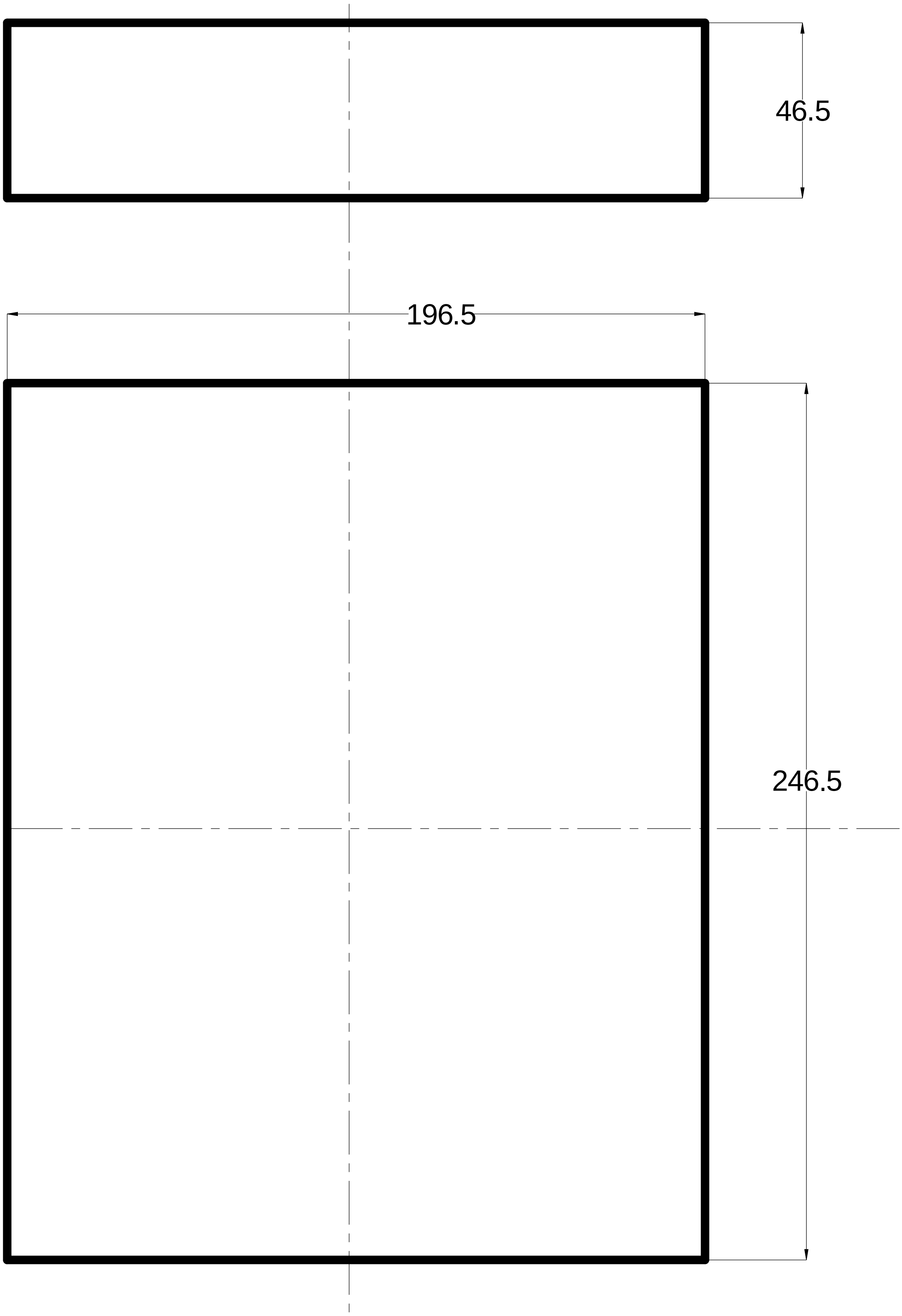


Figure 4.2 Pre-moulding Sample Process Sheet

Process sheet

Operation No.:02 (Surface grinding)

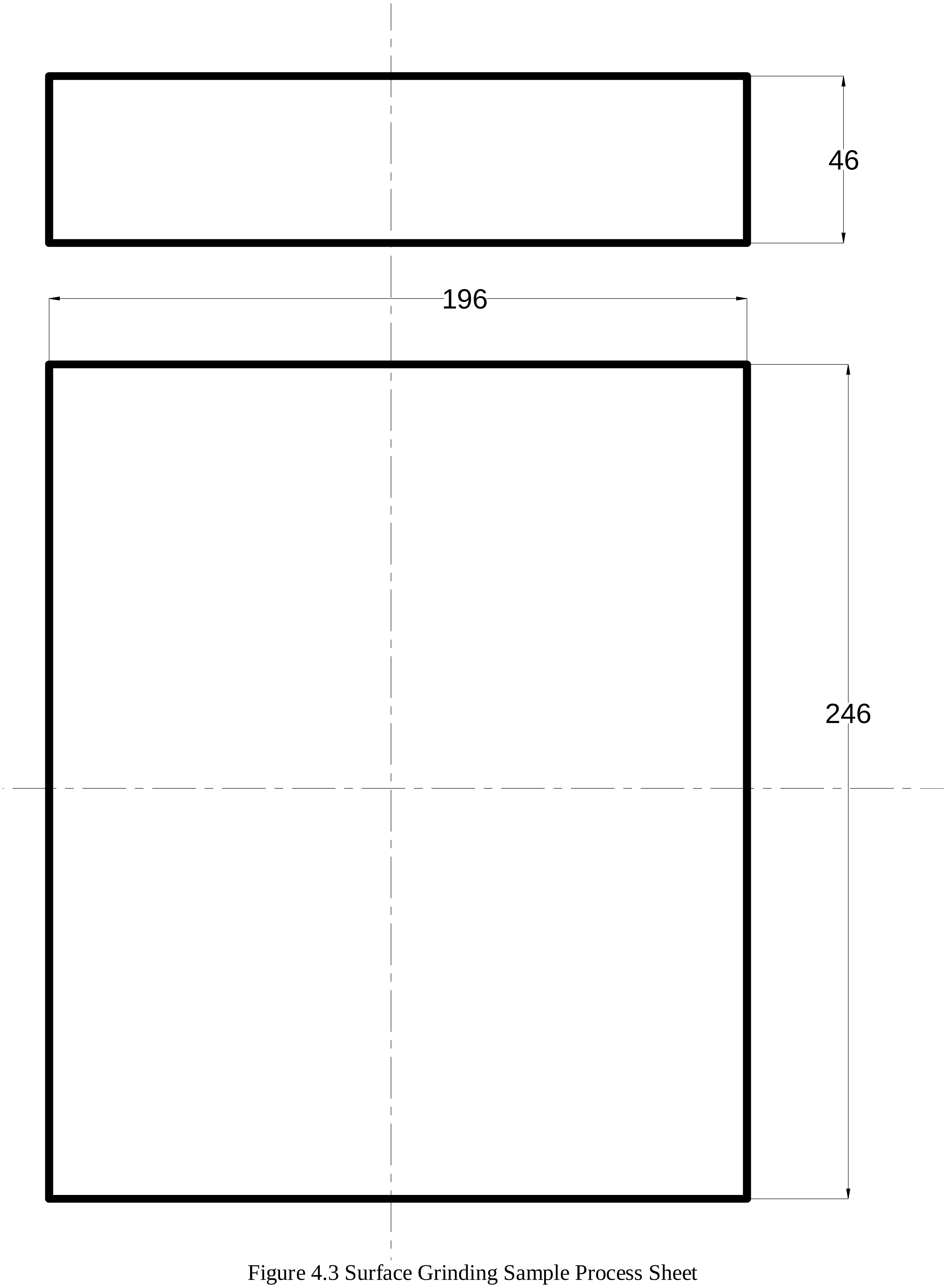


Figure 4.3 Surface Grinding Sample Process Sheet

Process sheet
Operation No.:03 (Spotting of holes)

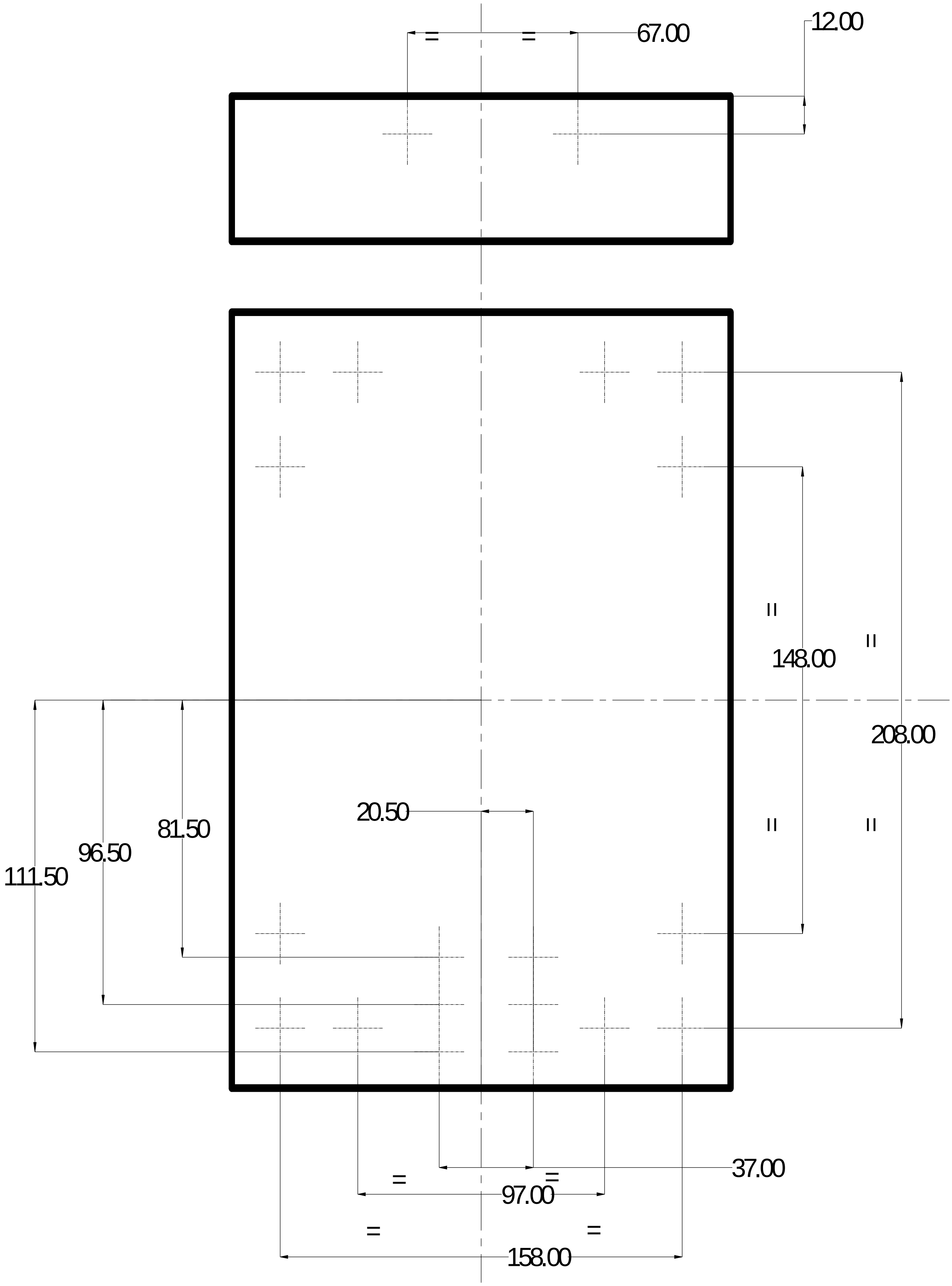


Figure 4.4 Sample Process Sheet for Spotting of Holes

Process sheet
Operation No.:04 (Drilling of holes)

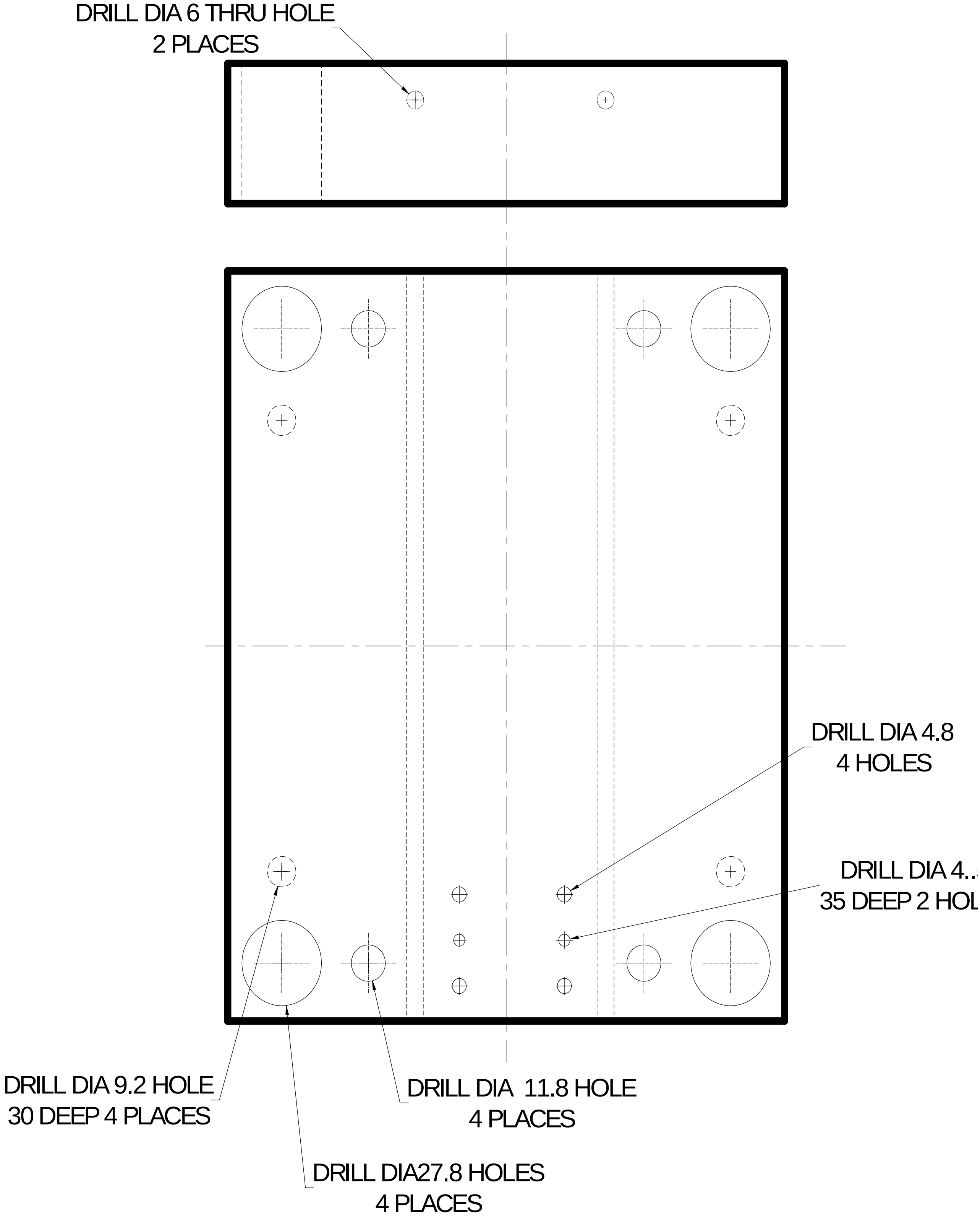


Figure 4.5 Sample Process Sheet for Drilling of Holes

Process sheet

Operation No.:05 (Jig grinding)

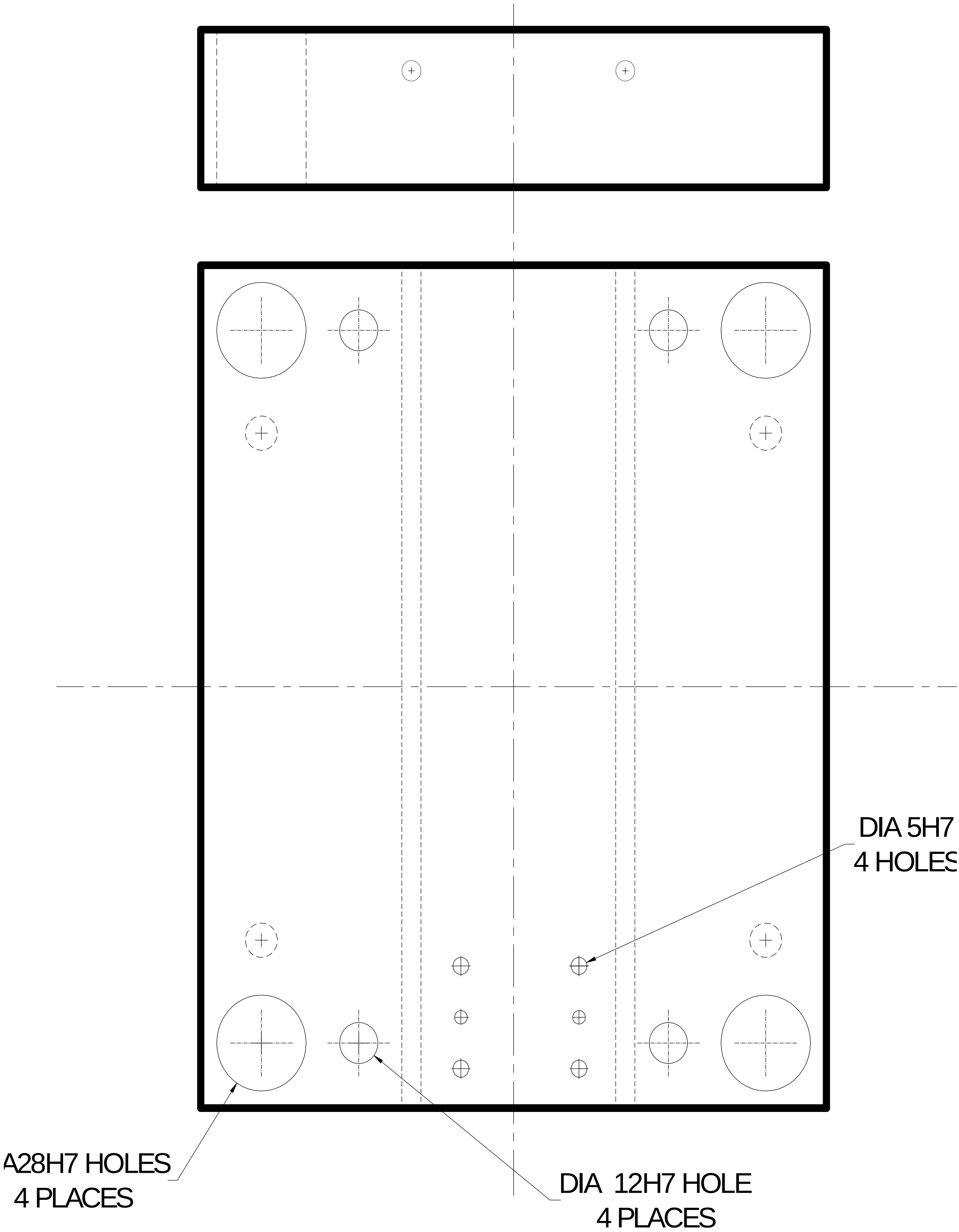


Figure 4.6 Sample Process Sheet for Jig Grinding

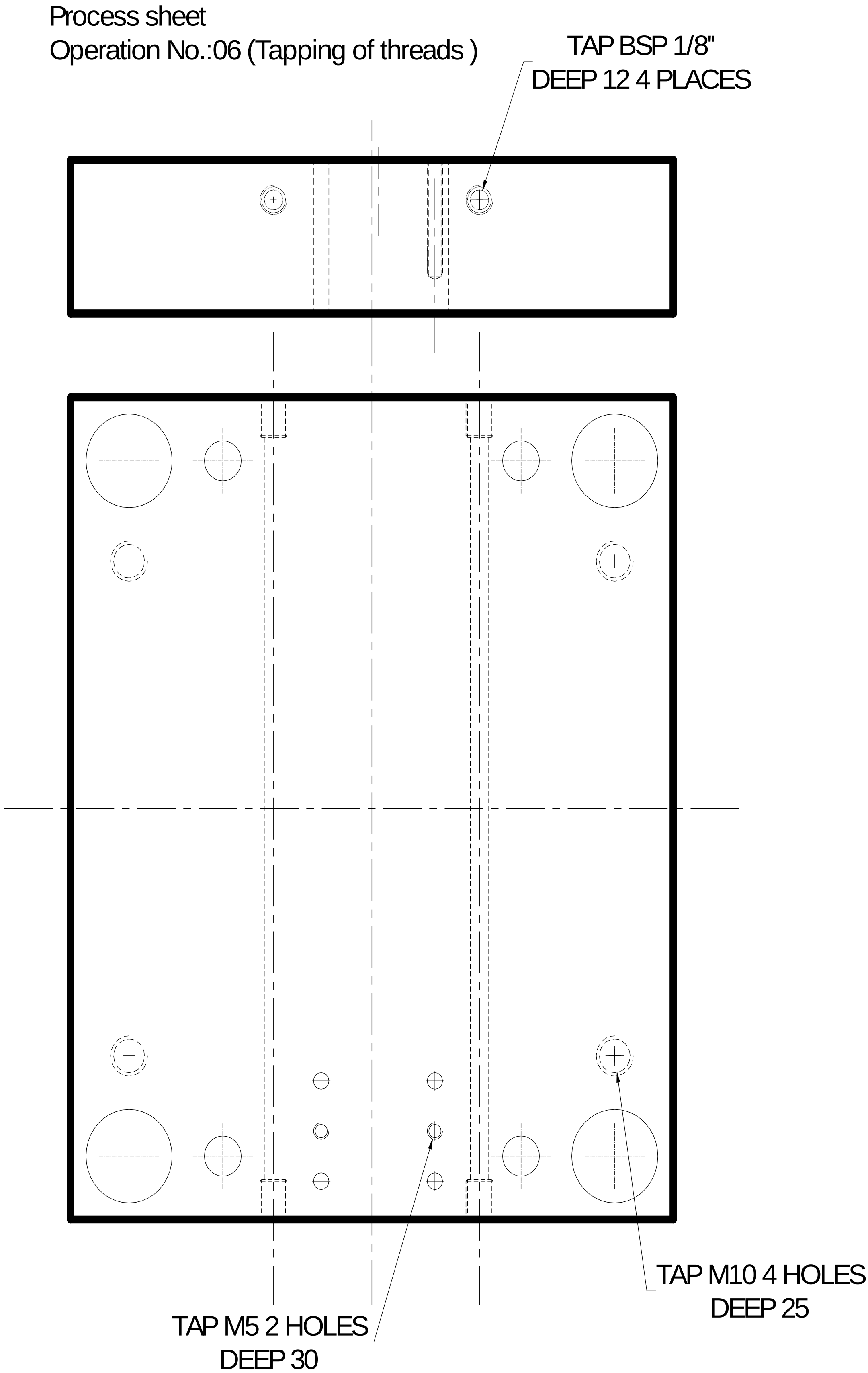


Figure 4.7 Sample Process Sheet for Tapping of Threads

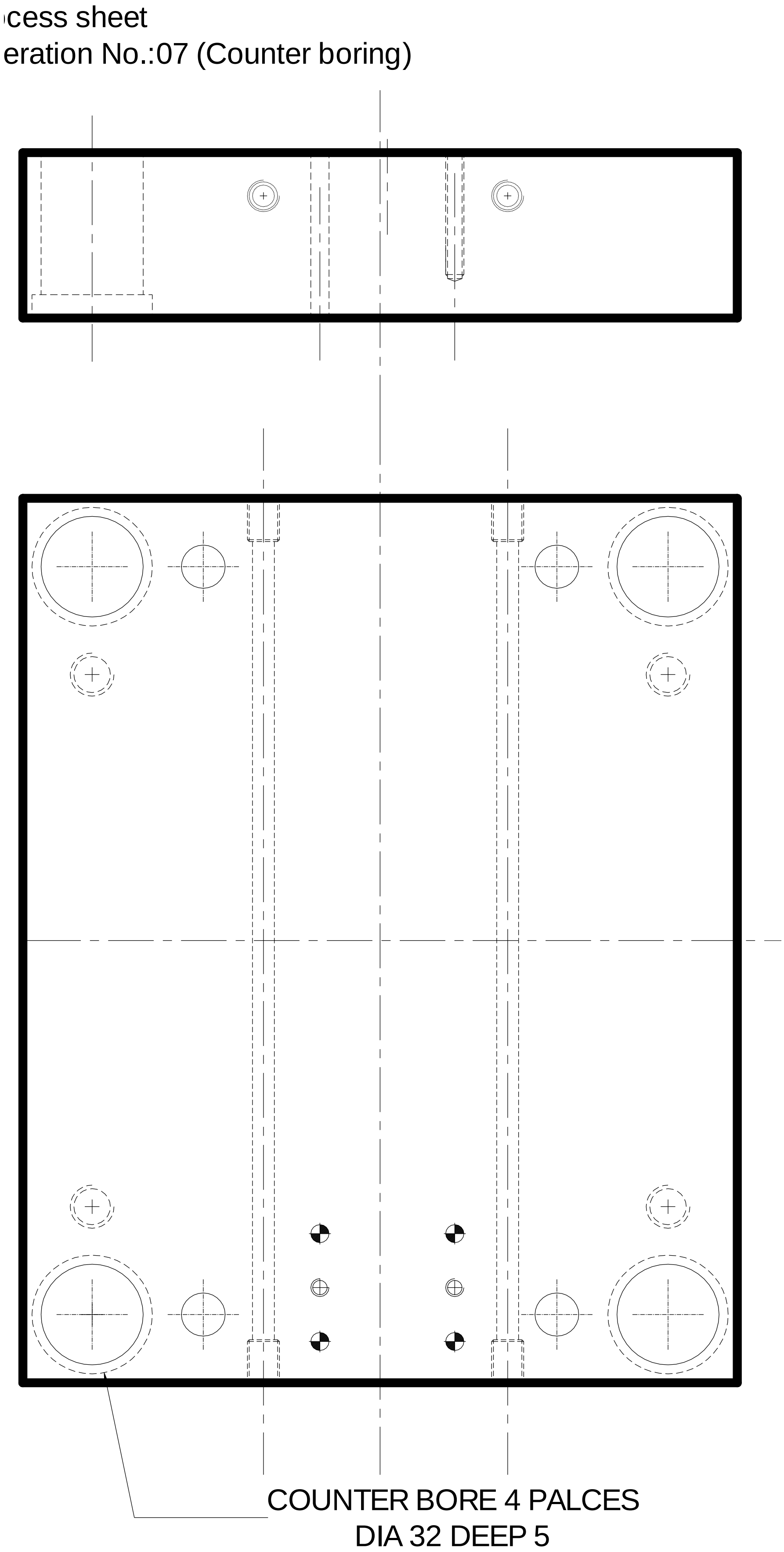


Figure 4.8 Sample Process Sheet for Counter Boring

Process sheet
Operation No.:08 (Pocket/Slot milling)

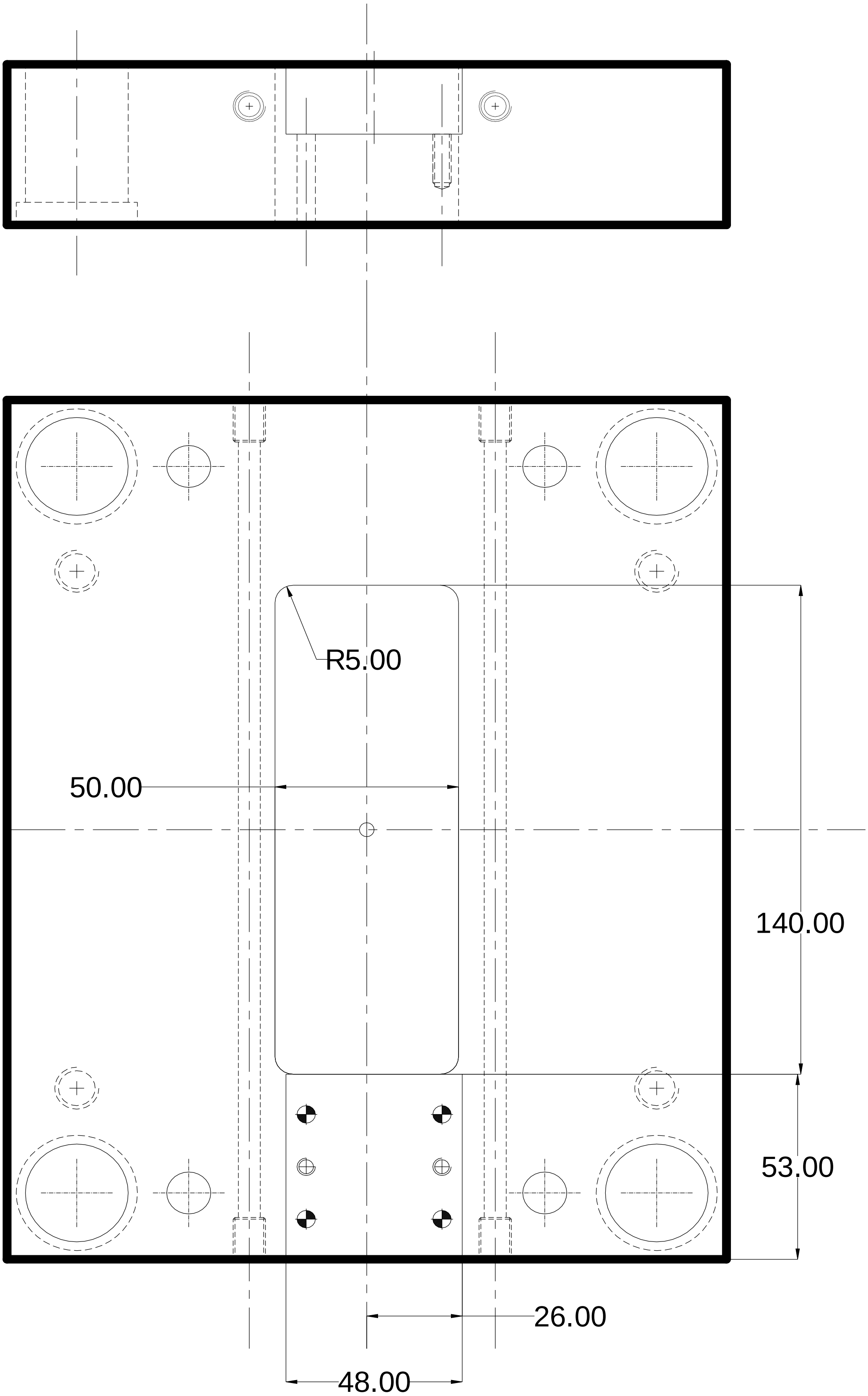


Figure 4.9 Sample Process Sheet for Pocket / Slot Milling

CHAPTER 5

COST ESTIMATON

Cost estimation is defined as an art of finding the cost, which is likely to be incurred during the manufacture of a die set before it is manufactured. Thus it is the probable cost of an article before the manufacturing starts. By compiling statement of the quantities of the material and production times the probable cost can be computed. The cost of the mould can be found by considering three major cost constituents namely, material cost, process cost and overhead costs.

Material Cost
Cost incurred on raw material used in making mould.

Process Cost
Cost incurred in converting raw material into finished product.

Overhead Cost
Cost incurred indirectly, such as stationery, rent, transportation etc.

Following machine hour rates are considered for the purpose of estimating the mould price. The price of various materials and type of materials used for moulds are shown in Table 5.1 and 5.2.

SL. NO.	MACHINING PROCESS	MACHINE HOUR RATE (Rs.)
1	Pre moulding	40
2	Turning	75
3	Surface Grinding	60
4	Cylindrical Grinding	60
5	CNC Milling (without program)	250
6	CNC Milling (with program)	400
7	Drilling	80
8	Jig Grinding	500
9	Heat Treatment	25/Kg
10	Vacuum Heat Treatment	150/Kg
11	CNC Turning	300
12	Assembly EDNC	150 450

Table 5.1 Machine Hour Rate

SL.NO.	MATERIAL	PRICE / Kg (Rs.)
1	Mild steel (St-42)	28
2	Hot Die Steel (T35Cr5MoV1)	120
3	Mould steel (T110W2Cr1)	75
4	Case hardening steel (20MnCr5)	50

Table 5.2 Price of various materials used for moulds

SPECIMEN CALCULATION FOR CORE PLATE

Raw material : St-42
Density of material : 7.85 g/cc = 7.85×10^{-3} gm/mm^3
Raw material size : 250 x 200 x 40.
Volume of material : 2000000 mm^3
Mass of raw material : volume X density (ρ)
$2.0 \times 10^6 \times 7.85 \times 10^{-3}$ = 15700 gms
$\approx$ 15.7 kg

Material cost = weight of cavity X Rs. Per kg.

= 15.7 x 28

= Rs. 440.

Machining cost = Number of hours X cost per hour.

Pre moulding = 6 x 40 = Rs. 240

Surface grinding = 3 x 60 = Rs.180.

Locating of holes = 2 x 80 = Rs.160

Drilling = 3 x 80 = Rs.240.

Jig Boring = 2 x 500 = Rs.1000

CNC milling = 3 x 250 = Rs.750.

Total machining cost of cavity Plate = 240 + 180 + 160 + 240 +1000 + 750

= Rs.2570.

No heat treatment charge for the above plate. And also no assembly charge and mould try out for the above.
Total cost of cavity Plate =Material cost + Machining cost

= 440 + 2570 = Rs.3010

Total mould price is calculated as detailed here.

Overhead cost = 10% of total material cost.

= 0.1 x 3010 = 301.0

Total material cost = Material cost + over head cost

= 3010 + 301.0 = 3311.0

Similarly cost of other detailed components of the mould can be estimated.
20% of the total material cost is considered as the cost apportioned for standard components which are not planned for fabrication.

About 5 to 6 times the total cost of the material is considered as incurred for the process/ methodizing/ Quality control and overhead costs. Thus the cost the mould is estimated.
Cost of the Standard components = 20% of Rs. 3772 (From table 5.3)

=754

Total material cost = 3772 + 754= 4526

Total mould cost = 6 times of total material cost = 4526 x 6 ≈ Rs. 27000

Table 5.3 shows the cost of Standard Components.

Sl. No.	Description	Qty	Material	Raw Material Size	Cost/Kg	Wt in Kg	Total Material Cost
1	Top Plate	1	St-42	250x200x30	28	11.8	330.4
2	Cavity plate	1	St-42	250x200x40	28	15.7	439.6
3	Core Plate	1	St-42	250x200x50	28	19.7	551.6
4	Core Back Plate	1	St-42	250x200x40	28	15.7	439.6
5	Spacer Block	2	St-42	250x70x40	28	5.5	308
6	Ejector Plate	1	St-42	250x125x15	28	3.7	103.6
7	Ejector Back Plate	1	St-42	250x125x20	28	4.9	137.2
8	Bottom Plate	1	St-42	250x250x30	28	14.8	414.4
9	Register Ring	1	St-42	□ 110x20	28	1.5	42
10	Guide Pillar	4	20MnCr5	DME-2025-E	50		Cost added to the total mat cost. **Bought out items**
11	Guide Bush	4	20MnCr5	DME-2025-E	50		
12	Ejector Guide Pillar	4	20MnCr5	DME-2025-E	50		
13	Ejector Guide Bush	4	20MnCr5	DME-2025-E	50		
14	Return Pin	4	T110W2Cr1	DME-2025-E	75		
15	Rest Button	4	T110W2Cr1	DME-2025-E	75		

16	Side Core Holder	1	T110W2Cr1	30x30x45	75	0.3	22.5
17	Wedge	1	T110W2Cr1	15x20x45	75	0.15	11.25
18	Guide Rail2	1	T110W2Cr1	20x30x55	75	0.25	18.75
19	Guide Rail1	1	T110W2Cr1	20x30x60	75	0.3	22.5
20	Sprue Bush	1	T110W2Cr1	□□30 x 60	75	0.34	25.5
21	Finger Cam	1	T110W2Cr1	□ 15 x 60	75	0.09	6.75
22	Wear Plate	1	T110W2Cr1	10x50x60	75	0.25	18.75
23	Side Core Insert	2	T35Cr5MoV1	15x20x40	120	0.16	38.4
24	Core Block	1	T35Cr5MoV1	50x55x150	120	3.25	390
25	Cavity Block	1	T35Cr5MoV1	40x55x150	120	2.6	312
26	Core Insert1	1	T35Cr5MoV1	□ 20 x 50	120	0.13	15.6
27	Core Insert2	1	T35Cr5MoV1	□ 30 x 50	120	0.28	33.6
28	Core Insert3	1	T35Cr5MoV1	□ 30 x 50	120	0.28	33.6
29	Core Insert4	1	T35Cr5MoV1	□ 20 x 50	120	0.12	14.4
30	Cavity Insert1	1	T35Cr5MoV1	□ 30 x 40	120	0.22	26.4
31	Cavity Insert2	1	T35Cr5MoV1	□ 20 x 40	120	0.1	12
32	Cavity Pin	1	T35Cr5MoV1	□ 10 x 40	120	0.03	3.6
33	SHCS- CavityInsert	4	STD	M6x40			500
34	SHCS-Core Insert	4	STD	M6x55			
35	SHCS-Wedge	1	STD	M6x35			
36	SHCS-Side Insert	1	STD	M5x20			
37	SHCS-Guide Rail	4	STD	M5x40			
38	Dowel-Guide Rail	4	STD	□□5 x 35			
39	□ 1.5 Stepped Ejector Pins	3	STD	L=122.10			
40	□ 3 Stepped Ejector Pins	7	STD	L=126.70			
41	1.2x5.0 Blade Ejector Pins	2	STD	L=115.60			

Table 5.3 Cost of Raw Material

Total cost excluding that of standard components = Rs. 3772

CHAPTER 6

CONCLUSIONS AND SCOPE FOR FURTHER STUDIES

6.1 CONCLUSIONS

- Design, manufacturing and user validation through in plant trail of the injection mould for rotor and cover was carried as part of the project. The design of the injection mould was completed with the required standards and quality set by the organization.
- The mould was manufactured using the in-house mould room facilities according to standards specified by the organization.
- Pilot production of the injection mould revealed that the manufactured mouldings were found satisfactory. Hence, the design and manufacturing of the injection mould for the components was successful. The parts produced were free from any defects.
- Cost estimation of the injection mould suggested that the mould fabrication was financially feasible and acceptable by the organization.

6.2 FURTHER SCOPE OF THE PROJECT

There is a huge scope to carry out the research work in the area of design.

- Fatigue analysis of the mould can be carried out for the mould to predict the life of the mould.
- Mold flow analysis can be carried out to control the material temperature, injection speed, injection pressure, cycle time, etc. The results can be used for, further optimization of the mould design.
- Study of design variations in the mould may be explored for adopting the same for other two-wheelers.

APPENDIX

A1. GLOSSARY OF TERMS

Air Vent : A small grove inscribed on the core surface in order to reduce shot fill and to remove any air or gas trapped in the metal during Solidification.

Bottom Plate : The plate fixed to the moving half of mould and facilitates Clamping.

Moulding : Plastic object moulded to the required shape by pouring or injecting melt into a mould, as distinct from one shaped by cutting or a mechanical working process.

Cavity : Control size, shape and surface texture of the moulded.

Core : Male part of the mould gives the moulding its inner profile.

Inserts : It is the heart of the mould, which has got male and female portions.

Draft : An angle or taper on the surface of a pattern or inserts that facilitates easy removal of parts from a mould or cavity.

Ejector : A mechanism that pushes the solidified moulding out of the mould.

Feeding : In a Moulding, providing molten Plastic to a region undergoing Solidification, usually at a rate sufficient to fill the mould cavity ahead of the solidification front and to compensate for any shrinkage accompanying solidification.

Flash : A thin section of Resin formed at the mould, core, and cavity joint or Parting in a moulding due to mismatch or high locking force.

Gate : The portion of the runner in a mould through which molten Plastic enters the cavity.

Guide Pillars and Bush : Cylindrical members meant to align the die halves and are made of hardened steel. Bushes are meant for wear resistance between housing and pillars.

Housing : A metal block which houses inserts (core and cavity), pillars, return pins etc.

Injection : The process of forcing molten metal or alloy under pressure into moulds.

Insert : A part formed from a second material, usually a metal that is placed in the moulds and appears as an integral structural part of the final Moulding.

Layout: The technique of arranging the impression, feeding system, Cooling system, return pins etc., to achieve good service, economy and productivity.

Parting Plane : In Moulding, the dividing plane between moulds halves.

Runner : A Channel through which molten Plastic flows from gate to impression to another.

Sleeve : Hollow ejector pins used to eject the components of circular bosses.

Sprue : The mould channel that connects injection unit of a machine and the Mould inserts via runner and gate.

Spreader : It deflects the molten plastic to the runner.

Tie Bar : A bar-shaped connection added to a casting to prevent distortion Caused by uneven contraction between two separate members of a mould.

A2. GEOMETRIC DATA OF COMPONENT

Geometric data of Cover is shown in the Table A2.1.

Mass of Comp **M** gm	2.31
Volume of Comp**Vc** (mm^3)	1877
Surface area of comp **Ac** (mm^2)	1251.05
Length of comp **Lc** mm	49
Width of cop **Wc** mm	35
Depth of comp **dc** mm	2
Max thickness $\mathbf{t_{max}}$ mm	1.2
Min thickness $\mathbf{t_{min}}$ mm	0.5
Prominent thickness **tp** mm	1.2

Table A2.1 Geometric Data of Cover

Geometric data of Rotor is shown in the Table A2.2.

Mass of Comp **M** gm	3.03 gm
Volume of Comp**Vc** (mm^3)	2465
Surface area of comp **Ac** (mm^2)	416.96
Length of comp **Lc** mm	26
Width of cop **Wc** mm	25
Depth of comp **dc** mm	15
Max thickness $\mathbf{t_{max}}$ mm	1.3
Min thickness $\mathbf{t_{min}}$ mm	0.7
Prominent thickness **tp** mm	1.3

Table A2.2 Geometric Data of Rotor

Cavity Fill Time is shown in the Table A2.3.

Thickness (mm)	Fill time(s)
0.51	0.15
1.02	0.50
1.25	1.02
2.03	1.90
2.25	2.10
2.50	2.45

Table A2.3 Cavity Fill Time

Gate thickness is shown in the Table A2.4.

Plastics	Thickness (mm)
ABS	1 to 2.0
PP	0.6 to 2
PC	1.5 to2.5
Nylon	0.5 to 1.5

Table A2.4 Gate Thickness

Table A2.5 shows Chemical Composition of Mould Steels.

Material Steel	IS No.	Designation of steel	AISI	Composition %							
				C	**Si**	**Mn**	**Cr**	**Mo**	**V**	**W**	**other**
P20 Pre-Hardened Steel	3748-1966	T55Ni2Cr65Mo30	P20	0.8	0.3	0.4	1	0.3	-	-	Ni.3.0
Cold Work steel	3749-1966	T90Mn2W50cr45	A4	0.9	-	2.0	0.45	-	-	0.5	
Case hardening steel	4432-1967	17mn1cr95 (T35Cr5MoV1)	1030	0.35	1	0.37	5	1.4	1	-	

Table A2.5 Chemical Composition of Mould Steels

A3. Production Data / Trail Data

Table A3.1 shows Temperature settings.

Zone	Temperature.^{0}C
Nozzle	265
Nozzle/shutoff	260
Zone1	255
Zone2	255
Zone3	190
Zone4	180

Table A3.1Temperature Setting

Table A3.2 shows Clamping and Opening Pressure, Speed and Position.

Step	Position mm.	Speed %	Pressure %
1	50	45	35
2	11p	42	04
3	11p	38	75
4	1000p	20	25
5	500	45	30
6	950	25	20

Table A3.2Clamping and Opening Pressure, Speed, Position

Velocity to pressure control transition

Injection time = 6 sec.

Screw position = 4 mm.

Table A3.3 shows Injection Pressure profile

Step.	Speed %	Pressure %
1	35	70
2	45	80
3	45	65
4	30	35

Table A3.3Injection Pressure Profile

Table A3.4 shows Plasticizing profile.

Stage	Speed %	Pressure %	Position (mm).
1	75	75	30
2	75	75	92
3	20	20	4

Table A3.4Plasticizing Profile

Table A3.5 shows Ejection profile.

Ejector position.	Speed %	Pressure %	Position %
Forward.	08	15	95
Backward.	35	40	03

Table A3.5Ejection Profile

Cushion Position = 2.22

Injection period = 5.99 sec.

Cycle time = 70 sec.

Table A3.6 shows Tolerance used for the fabrication of Moulds. (All dimensions are in mm unless otherwise stated)

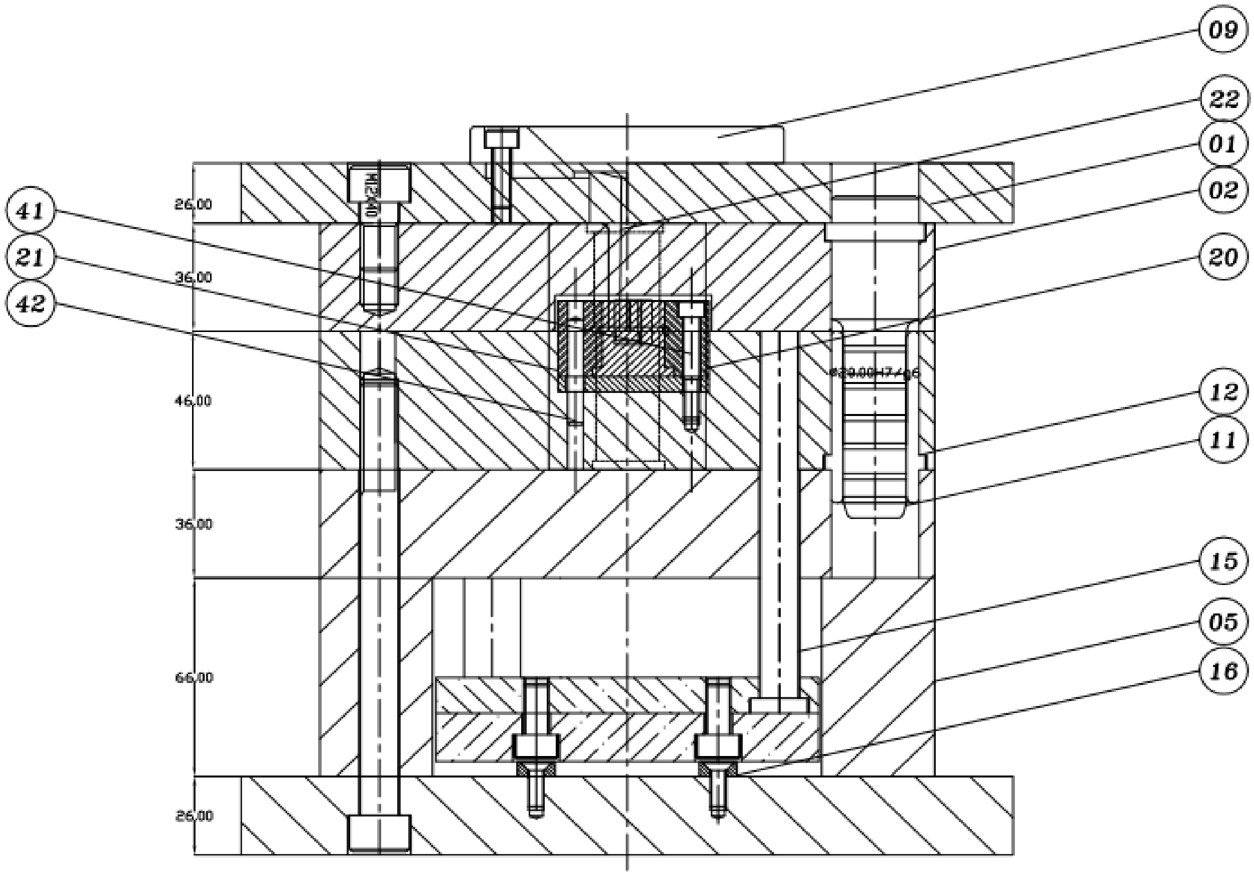
General Tolerance for Metric Dimensions where not specified							
Decimal Dimensions	**One Place ± 0.1**		**Two place ± 0.02**		**Three Place ± 0.005**		**Angles ± 0.5^0**
Dimensions other than decimals	0.5 – 3	3 – 6	6 - 30	30 - 120	120 - 400	400 – 1000	Tolerance Class Medium
	± 0.1	± 0.1	± 0.2	± 0.3	± 0.5	± 0.8	

Table A3.6 Tolerance used for the fabrication of Moulds

REFERENCES

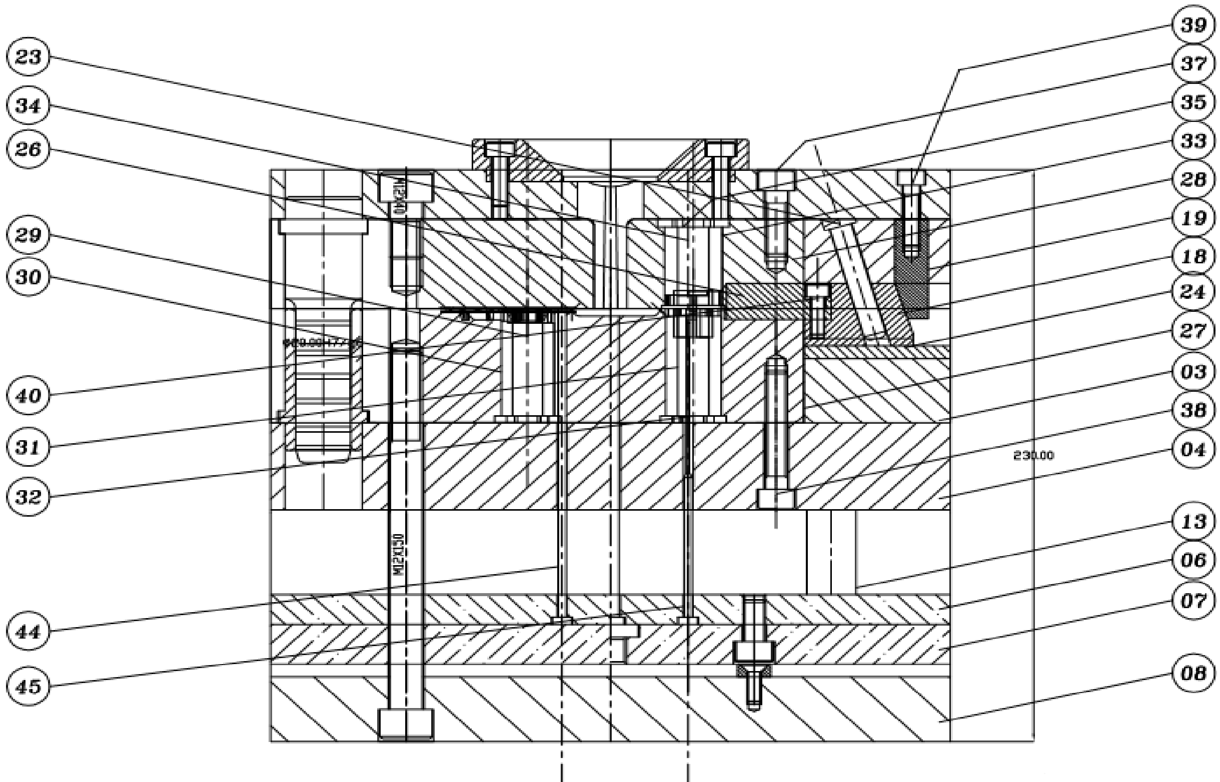
1. L. Kong, J.Y.H. Fuh, K.S. Lee, X.L. Liu, L.S. Ling, Y.F. Zhang, and A.Y.C. Nee, Determination of optimal parting directions in plastic injection mould design, CIRP Ann. Manuf. Technol. 2003, pp-429–432.
2. Kevin Alam, Musa R. Kamal, A robust optimization of injection moulding runner balancing, McGill University,Montreal,2005, pp 1934-1944.
3. D. E. Dimla, M. Camilotto, F. Miani, Design and optimization of conformal cooling channels in injection moulding moulds School of Design, Engineering and Computing, Bournemouth University, UK ,2004, pp 1294-1300.
4. Charles Shaw, Department of Manufacturing Engineering, Gate location design in injection moulding of an automobile junction Hong Kong ,1999, pp110-115.
5. Shaw, Sun y, The application of U shaped grooves for cooling of injection moulds,Vol216, 2002, pp 1561-1573.
6. Z.lou ,H,Jiang, X. Ruan ,Mould manufacturing sequence in injection moulding, 2004, pp1673-1685.
7. Wong, C. T., Shamsuddin, Sulaiman Design and Simulation of Plastic Injection Moulding Process, Malaysia,2004, pp 85-99.
8. M. K. A. Ariffin, S. Sulaiman, M. M. Hamdan, Z. Leman, N. I smail,Design and Simulation of Plastic Injection Moulding Process, University Putra Malaysia,2004, pp 85 – 99.
9. S.H. Tang**,** Chew W, Wiecek, M. M., and Zhang, J, Journal of Mechanical Design, 121, 1999,pp 179 [9]
10. Jansen, K. M. B., and Titomanlio, G. Polymer Engineering and Science, 36, 1996, pp-2029.
11. Dominick V Rosato and Donald V Rosato “Injection Moulding Handbook”, 2nd Edition, 1998, Hanser Publisher.
12. R. G. W. Pye, “Injection Mould Design”, 4th Edition, 2000, Pearson Scientific and Technical Publications.

DRAWING



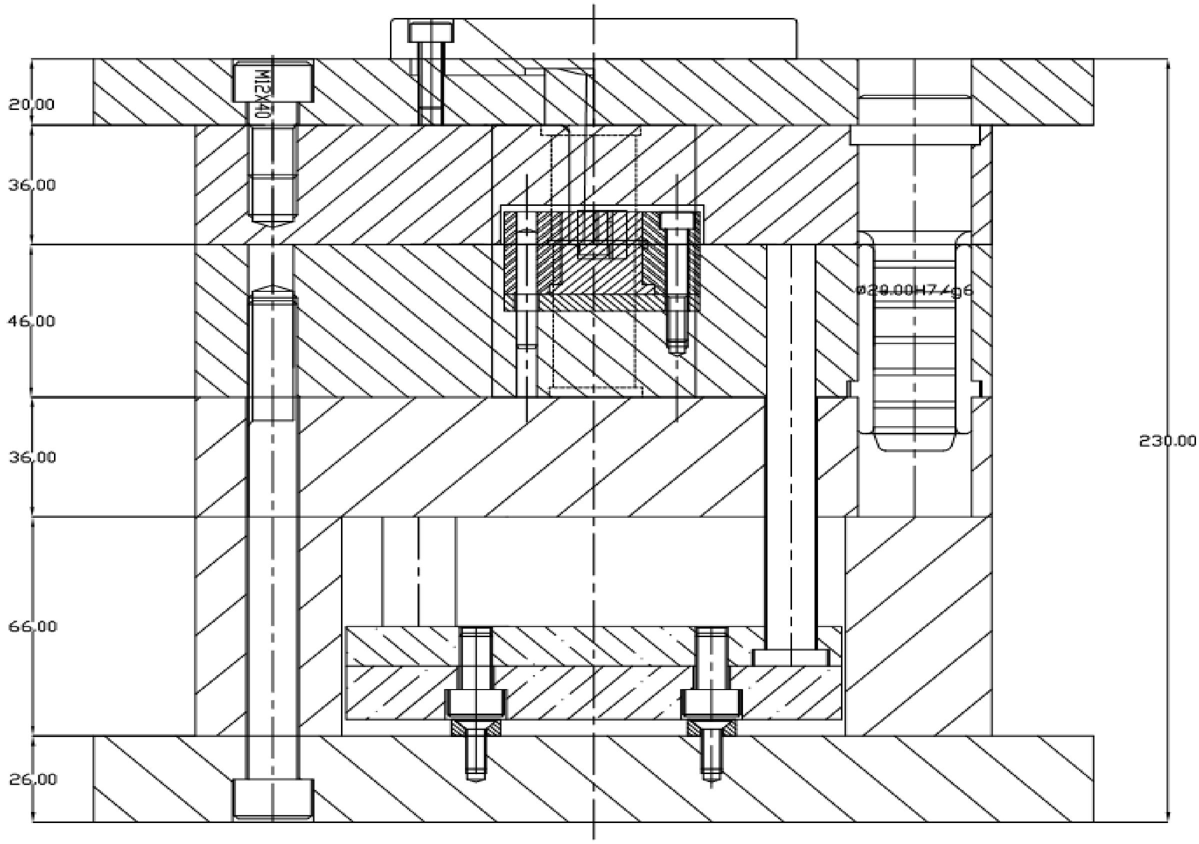
MOULD ASSEMBLY– SEC"A–A"

Drawing-1



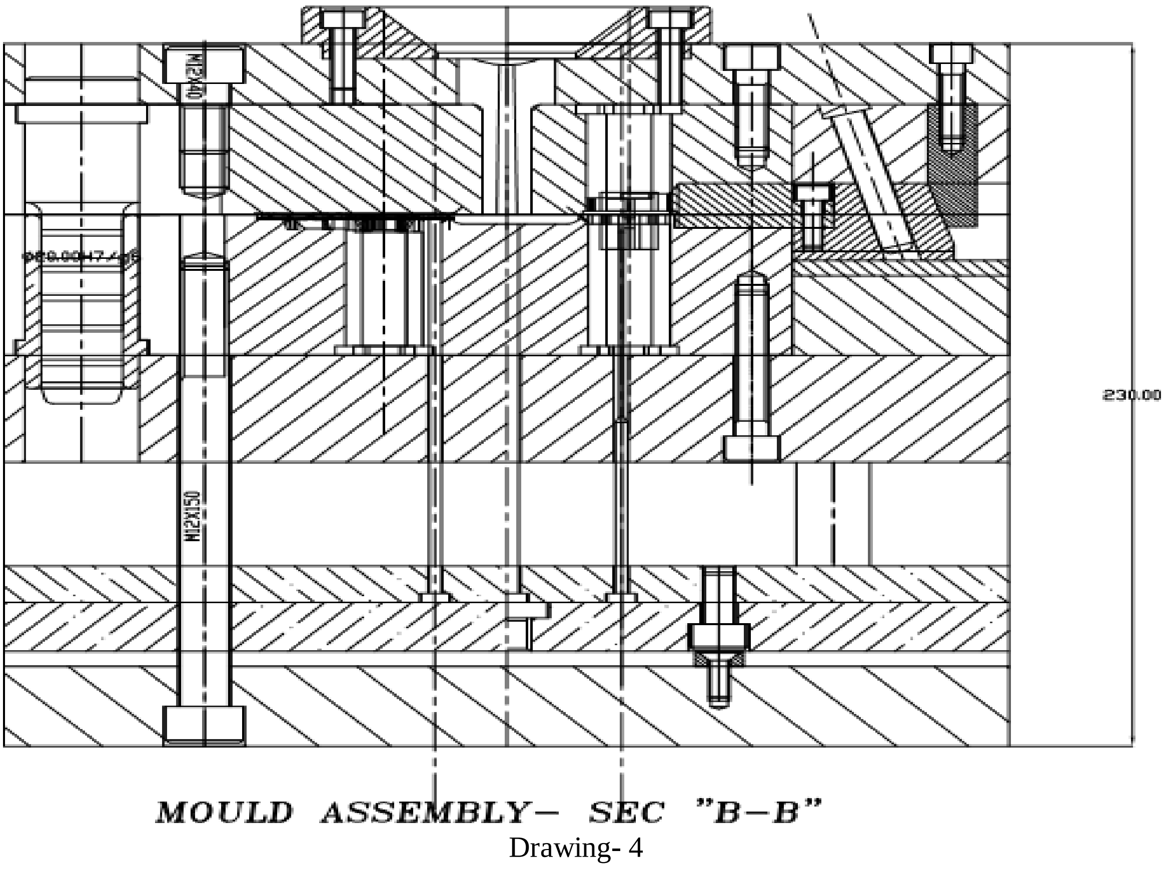
MOULD ASSEMBLY– SEC "B–B"

Drawing-2



MOULD ASSEMBLY– SEC"A–A"

Drawing-3



MOULD ASSEMBLY– SEC "B–B"

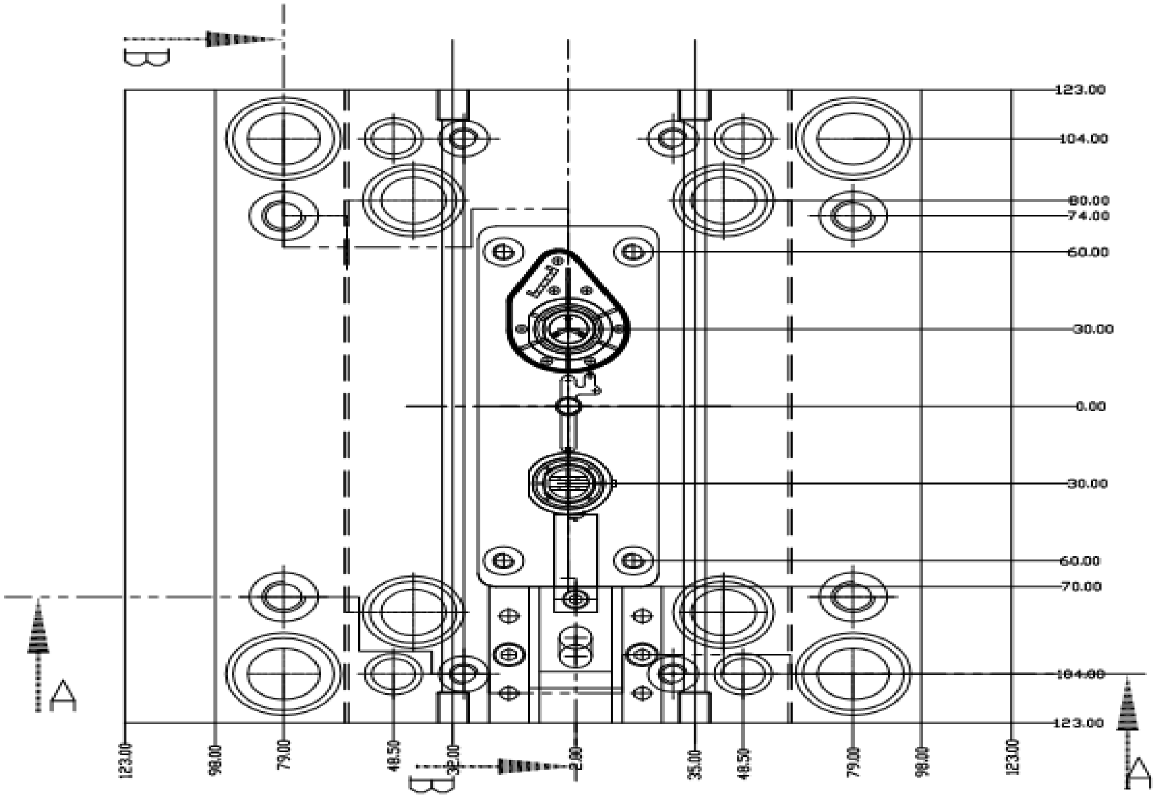
Drawing- 4

45	1.2X5.0 -BLADE EJ PINS	STD	02	L=115.60	
44	Ø3-STRAIGHT EJ PINS	STD	07	L=126.70	
43	Ø1.5-STEPPED EJ PINS	STD	03	L=122.10	
42	DOWEL-GUIDE RAIL	STD	04	Ø5X35	
41	SHCS-GUIDE RAIL	STD	04	M5X40	
40	SHCS-SIDE CORE INS	STD	01	M5X20	
39	SHCS-WEDGE	STD	01	M6X35	
38	SHCS-CORE INS	STD	04	M6X55	
37	SHCS-CAV INS	STD	04	M6X40	
36					
35	CAVITY PIN	HDS	01	Ø5X35	45-48HRC
34	CAVITY INSERT 2	HDS	01	Ø15X35	45-48HRC
33	CAVITY INSERT 1	HDS	01	Ø26X35	45-48HRC
32	CORE INSERT 4	HDS	01	Ø15X46	45-48HRC
31	CORE INSERT 3	HDS	01	Ø24X46	45-48HRC
30	CORE INSERT 2	HDS	01	Ø24X46	45-48HRC
29	CORE INSERT 1	HDS	01	Ø15X46	45-48HRC
28	CAVITY BLOCK	HDS	01	36X50X140	45-48HRC
27	CORE BLOCK	HDS	01	46X50X140	45-48HRC
26	SIDE CORE INSERT	HDS	02	12x16x38	45-48HRC
25					
24	WEAR PLATE	OHNS	01	5x48x53	
23	FINGER CAM	OHNS	01	Ø12X50	45-48HRC
22	SPRUE BUSH	OHNS	01	Ø 24x53	45-48HRC
21	GUIDE RAIL 1	OHNS	01	14X25X53	45-48HRC
20	GUIDE RAIL 2	OHNS	01	14X25X53	45-48HRC
19	WEDGE	OHNS	01	12X18X40	45-48HRC
18	SIDE CORE HOLDER	OHNS	01	25X26X40	45-48HRC
17					
16	REST BUTTON	OHNS	04	STD-DME-2025-EB-36-46/1/66	
15	RETURN PIN	OHNS	04	STD-DME-2025-EB-36-46/1/66	
14	EJE GUIDE BUSH	20MnCr5	04	STD-DME-2025-EB-36-46/1/66	
13	EJE GUIDE PILLAR	20MnCr5	3+1	STD-DME-2025-EB-36-46/1/66	
12	GUIDE BUSH	20MnCr5	3+1	STD-DME-2025-EB-36-46/1/66	
11	GUIDE PILLAR	20MnCr5	3+1	STD-DME-2025-EB-36-46/1/66	
10					
09	REGISTER RING	MS	01	Ø100x17	
08	BOTTOM PLATE	MS	01	246X246X26	
07	EJECTOR BACK PLATE	MS	01	246X122X16	
06	EJECTOR PLATE	MS	01	246X122X12	
05	SPACER BLOCK	MS	02.	246X66X36	
04	CORE BACK PLATE	MS	01	246X196X36	
03	CORE PLATE	MS	01	246X196X46	
02	CAVITY PLATE	MS	01	246X196X36	
01	TOP PLATE	MS	01	246X196X26	
Sl No.	DESCRIPTION	Matl	QTY	SIZE	REMARKS

DESCRIPTION : MOULD ASSEMBLY

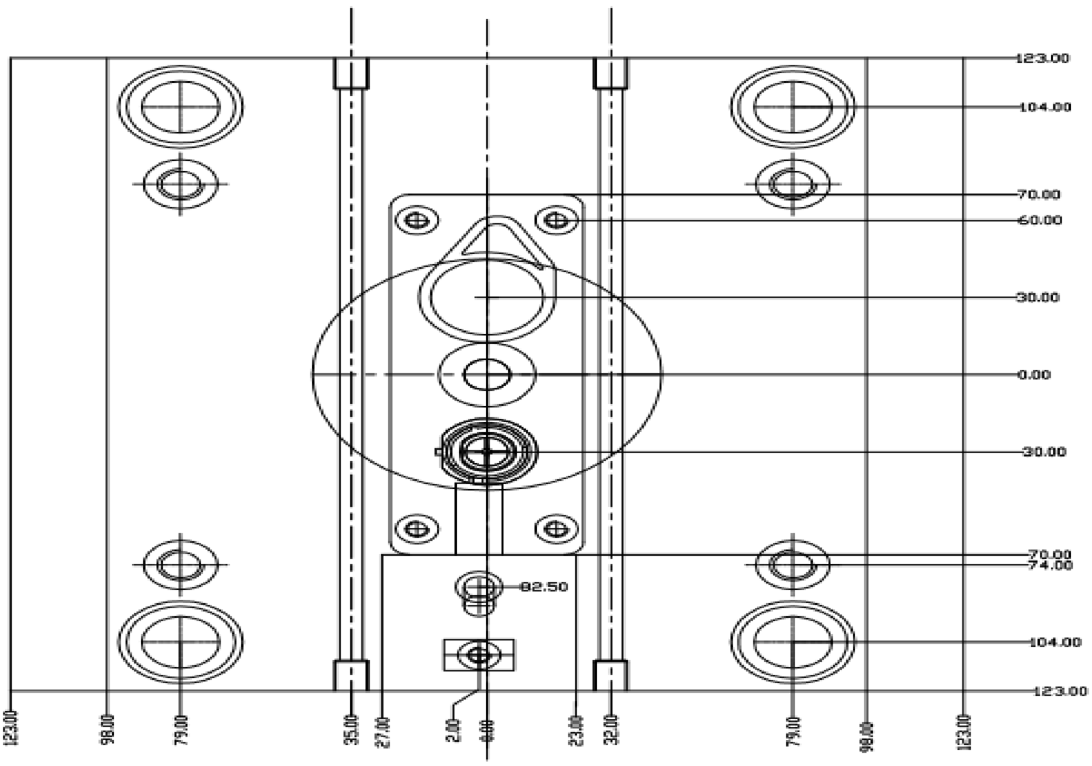
FAMILY MOULD FOR ROTOR & COVER

Drawing-5



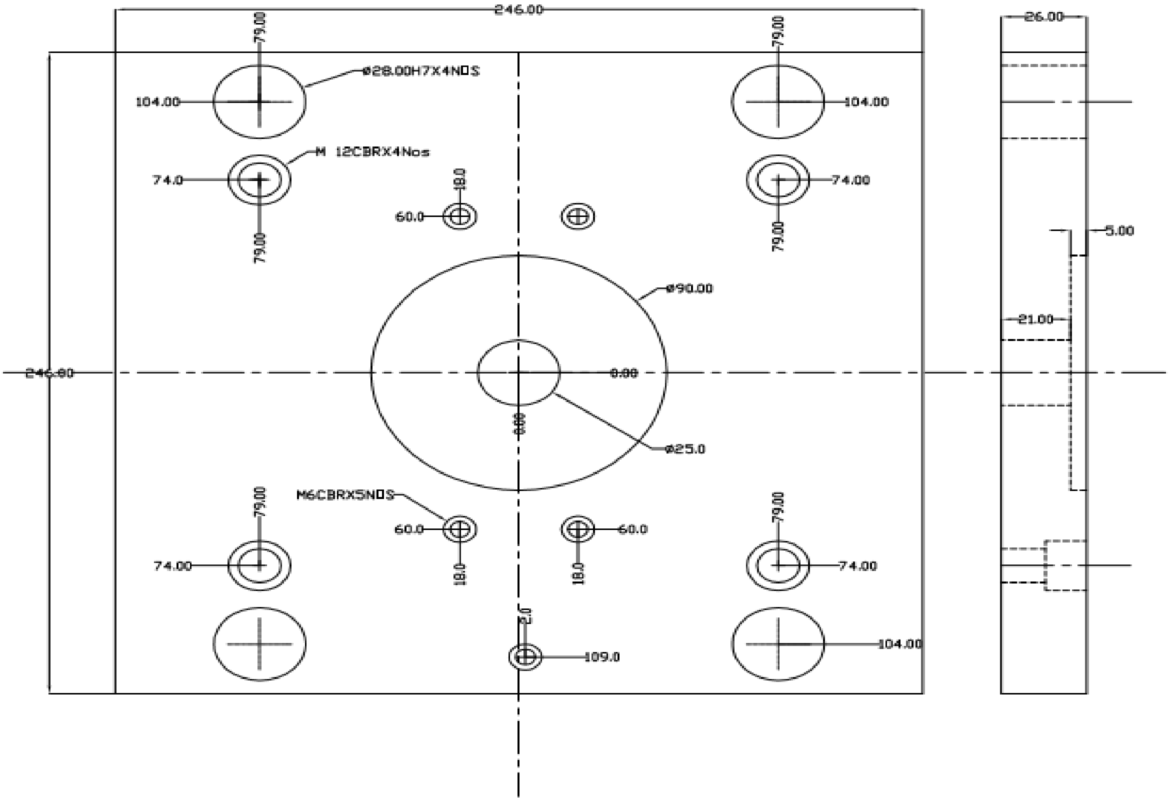
MOULD ASSEMBLY – TOP VIEW OF CORE HALF

Drawing-6

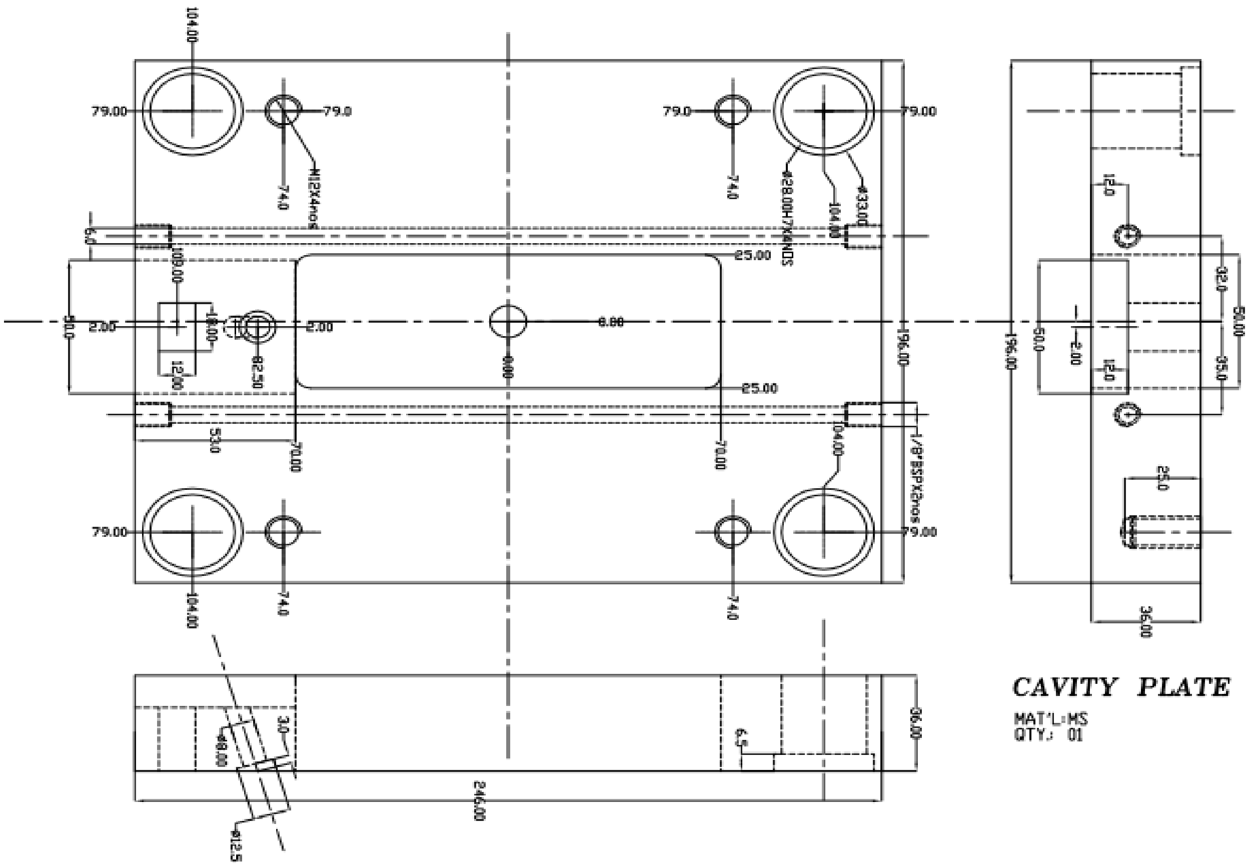


MOULD ASSEMBLY– TOP VIEW OF CAVITY HALF

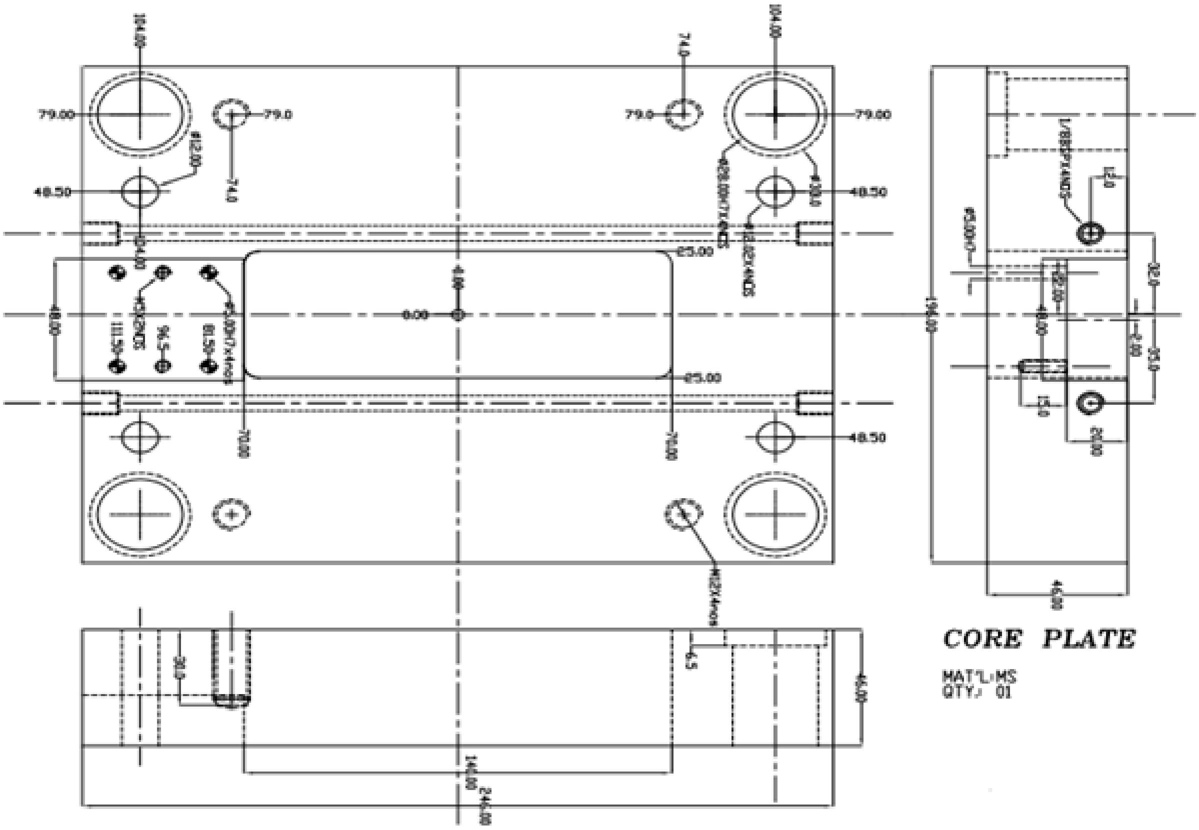
Drawing-7



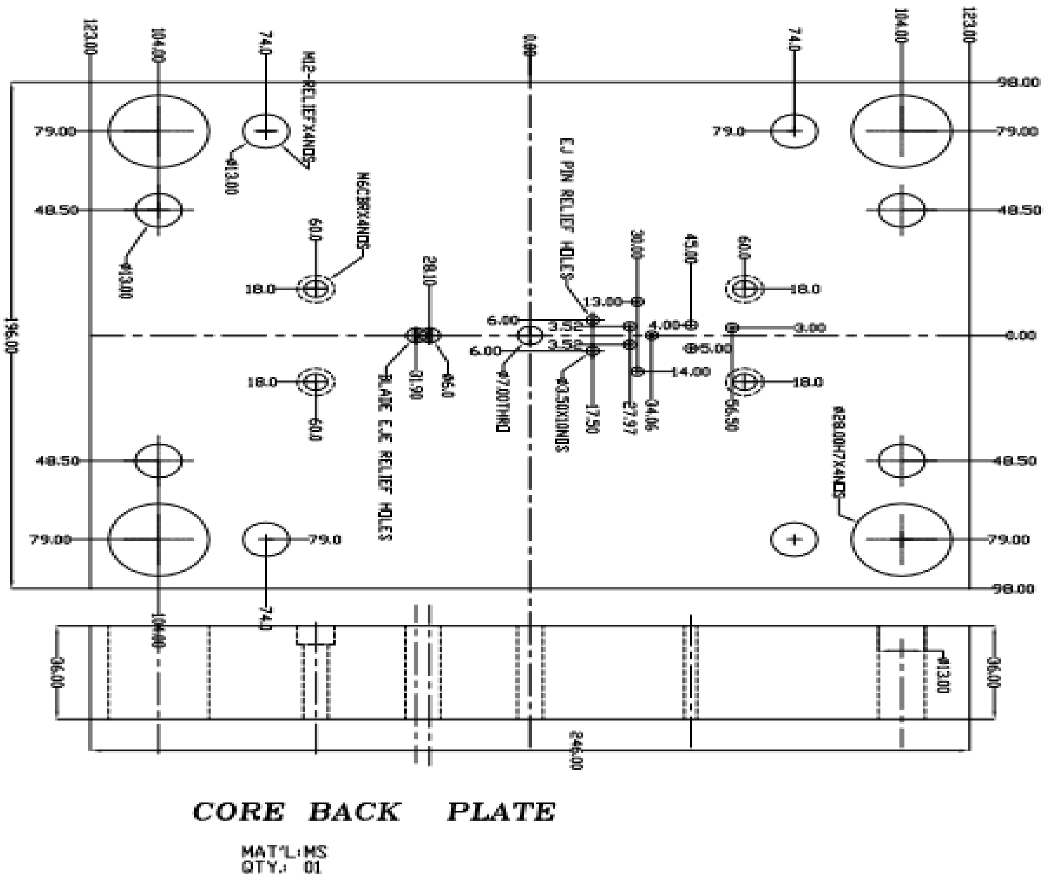
Drawing-8



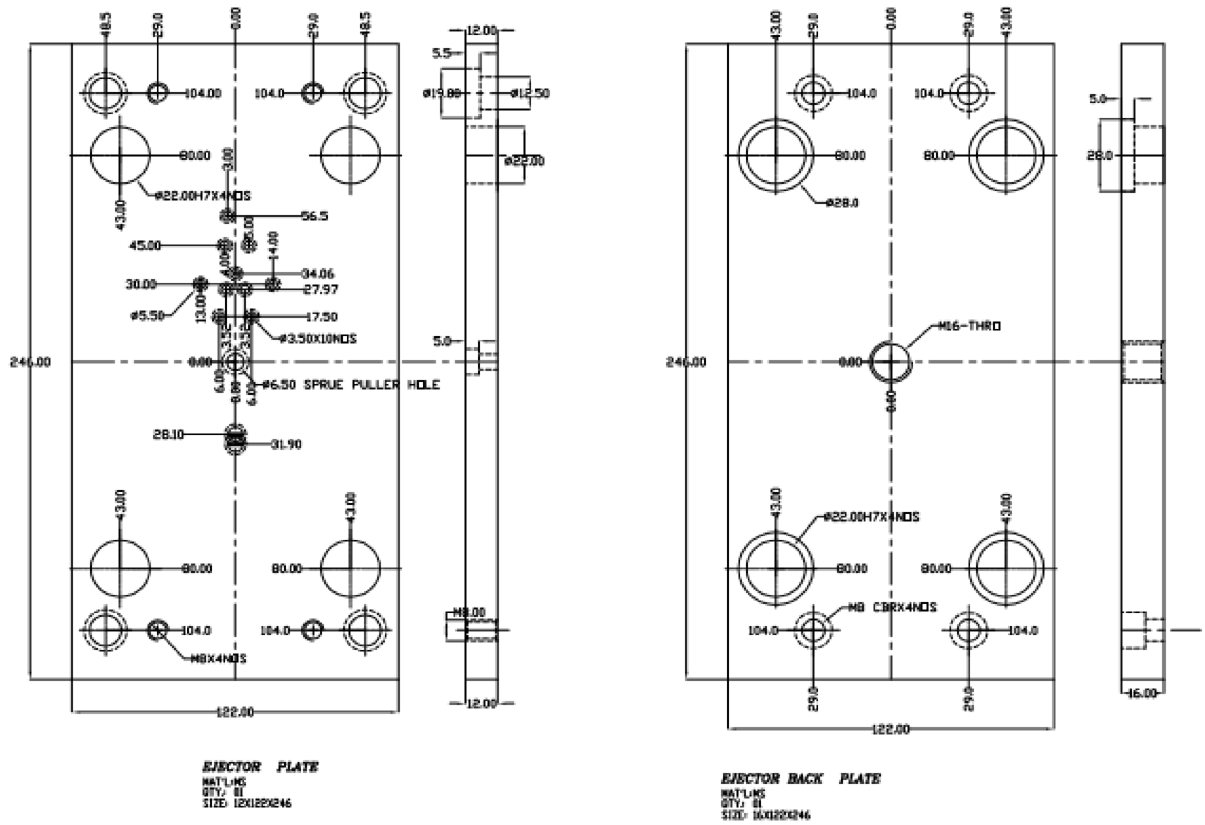
Drawing-9



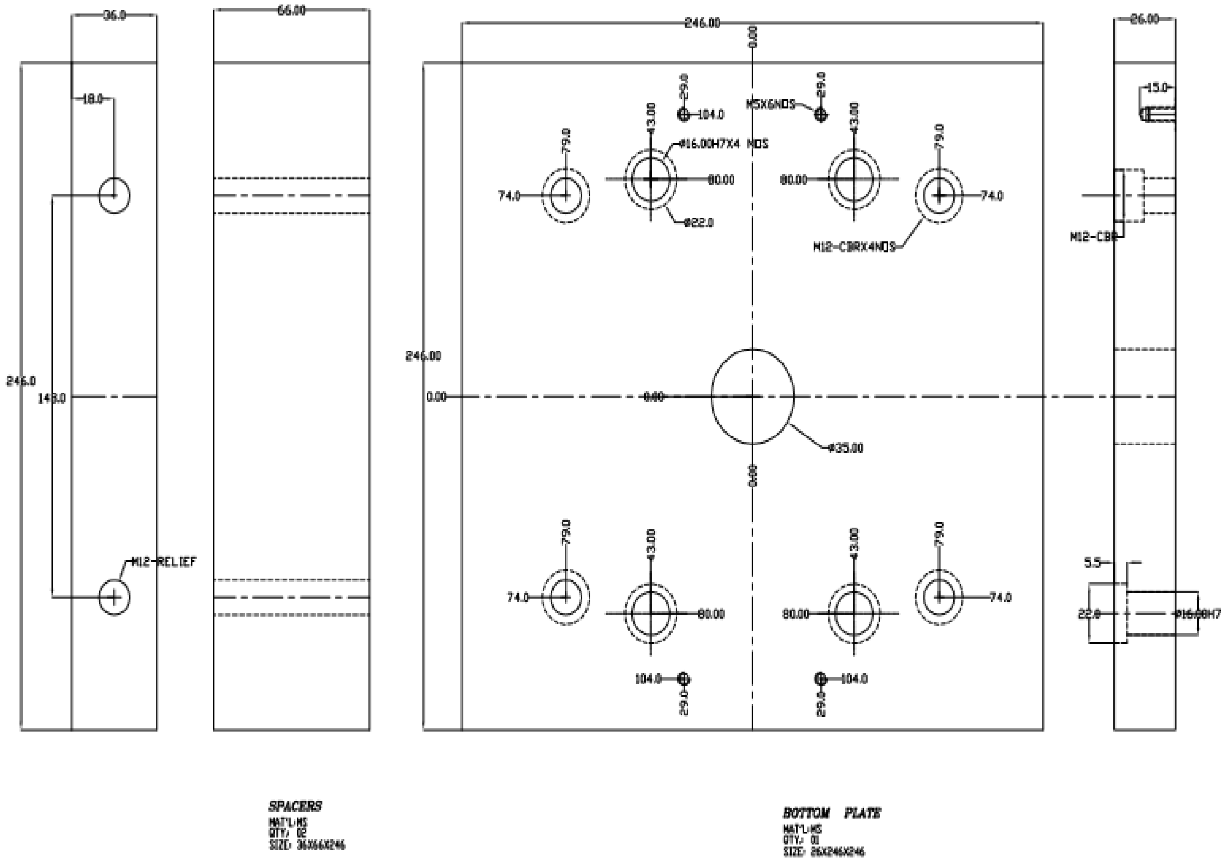
Drawing-10



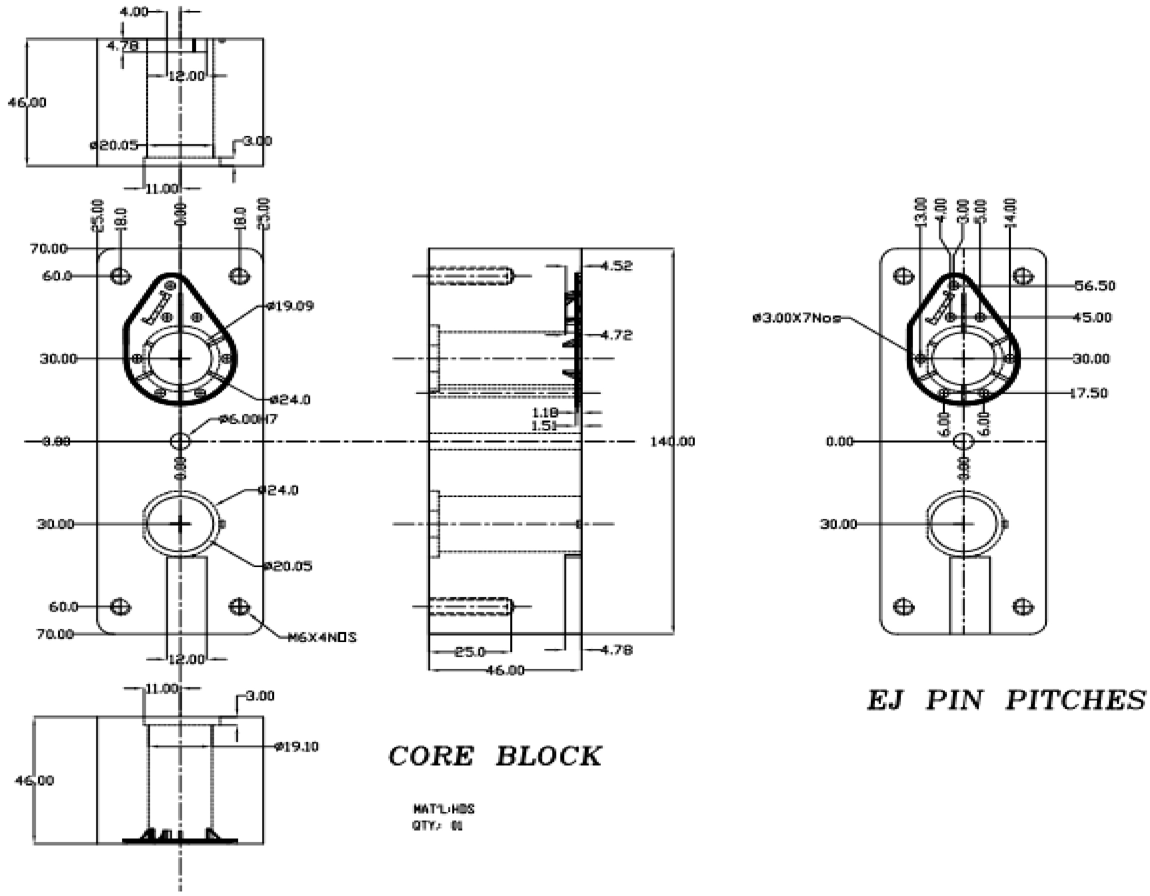
Drawing-11



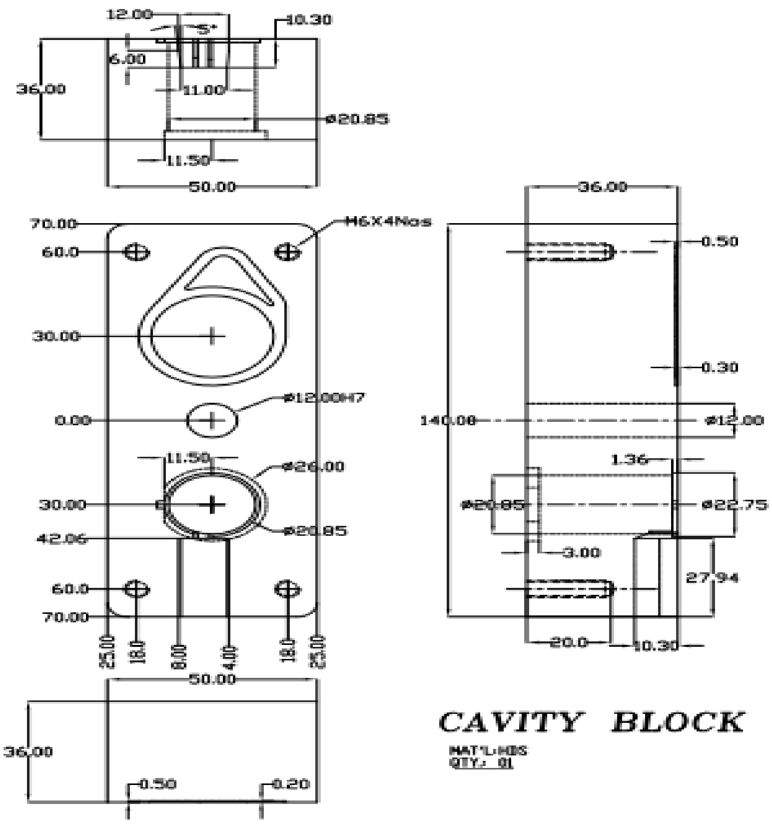
Drawing-12



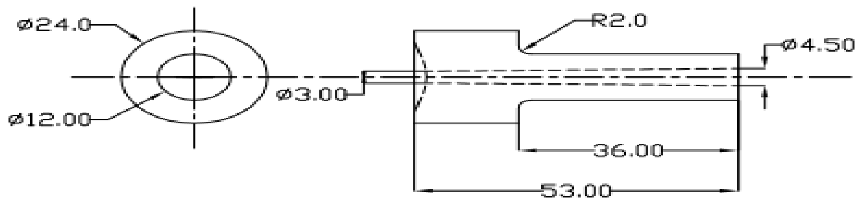
Drawing-13



Drawing-14

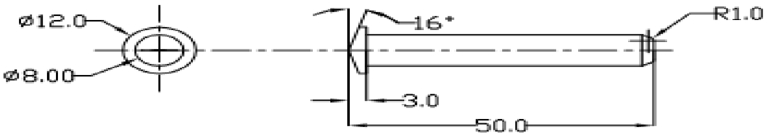


Drawing-15



SPRUE BUSH

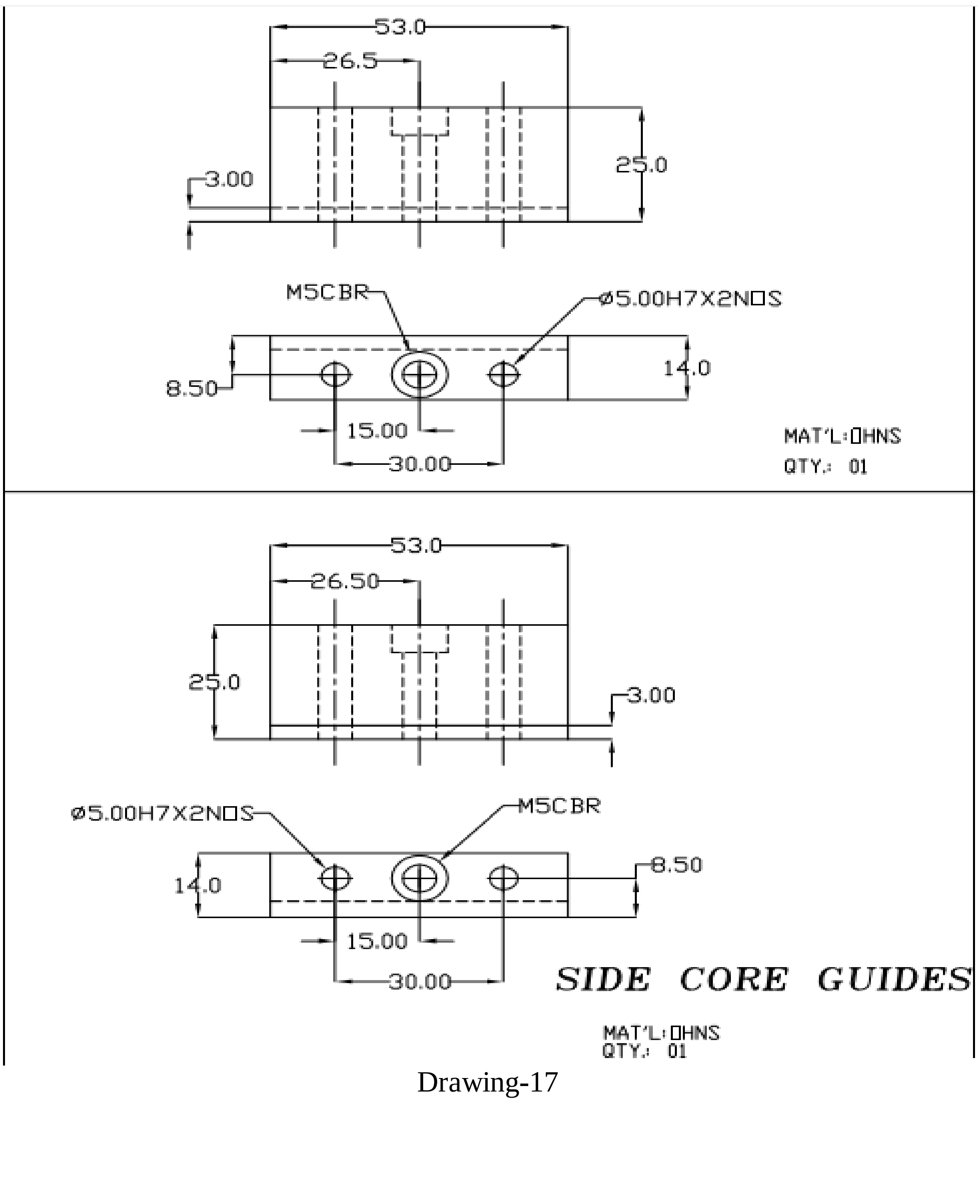
MAT'L: OHNS
QTY.: 01



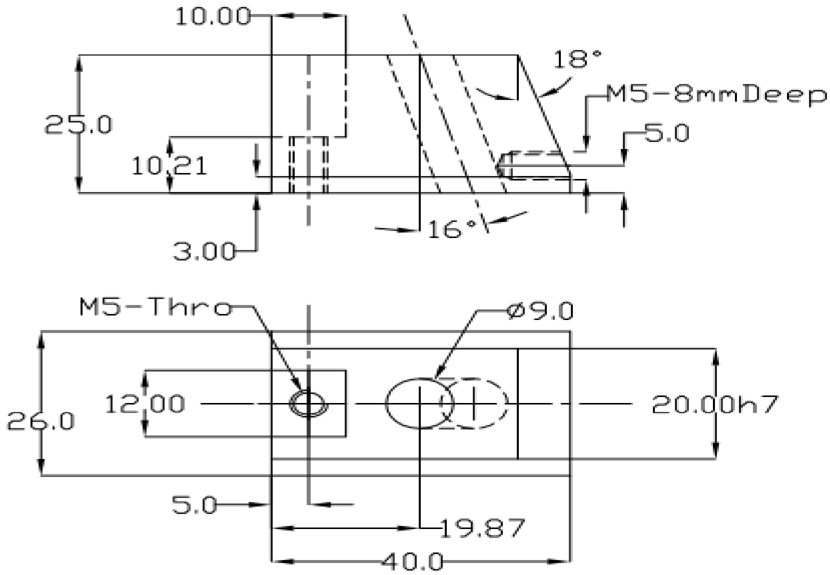
FINGER CAM

MAT'L: OHNS

Drawing-16



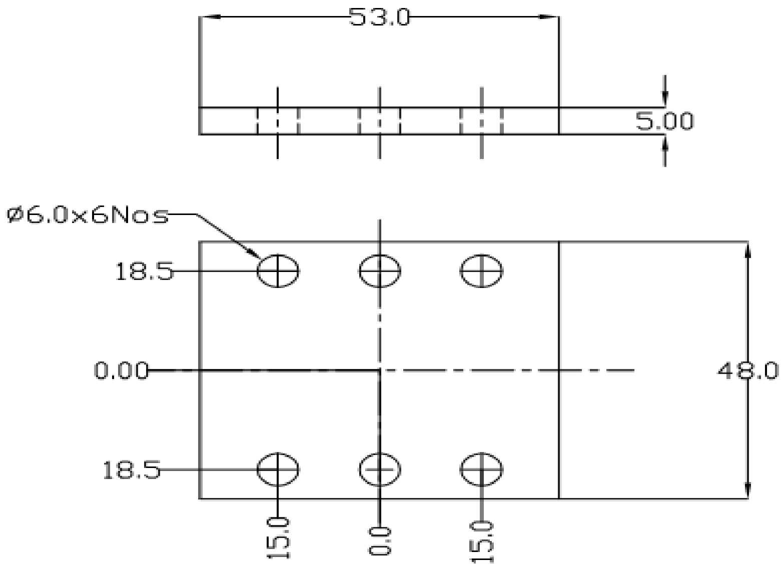
Drawing-17



SIDE CORE HOLDER

MAT'L:OHNS
QTY.: 01

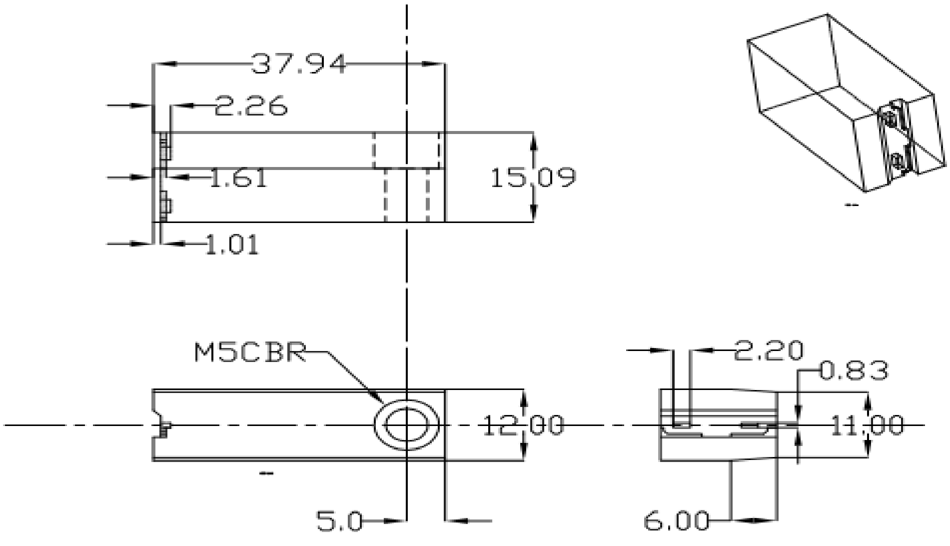
Drawing-18



WEAR PLATE

MAT'L:OHNS
QTY.: 01

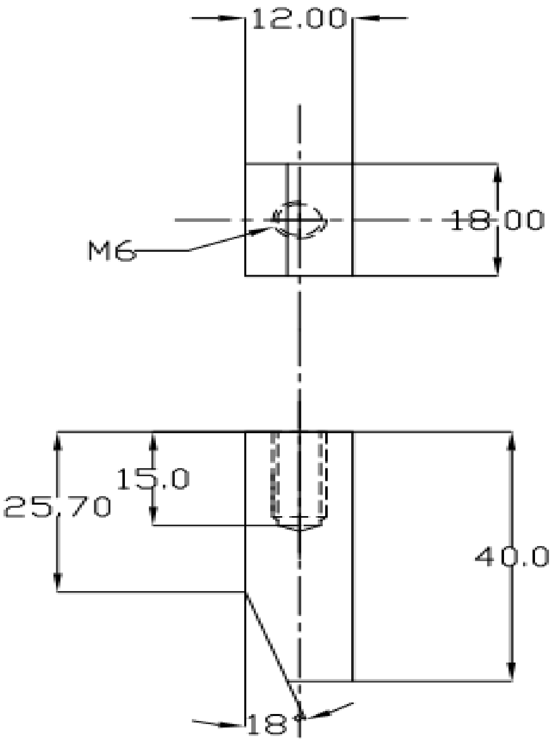
Drawing-19



SIDE CORE INSERT

MAT'L:HDS
QTY.: 01

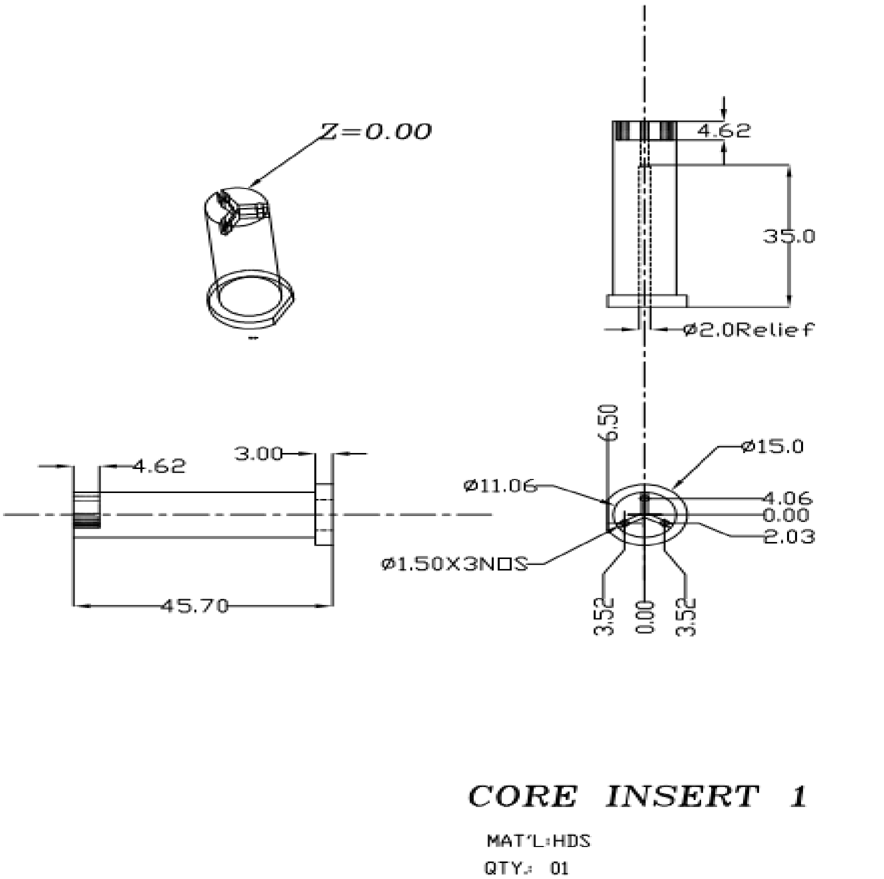
Drawing-20



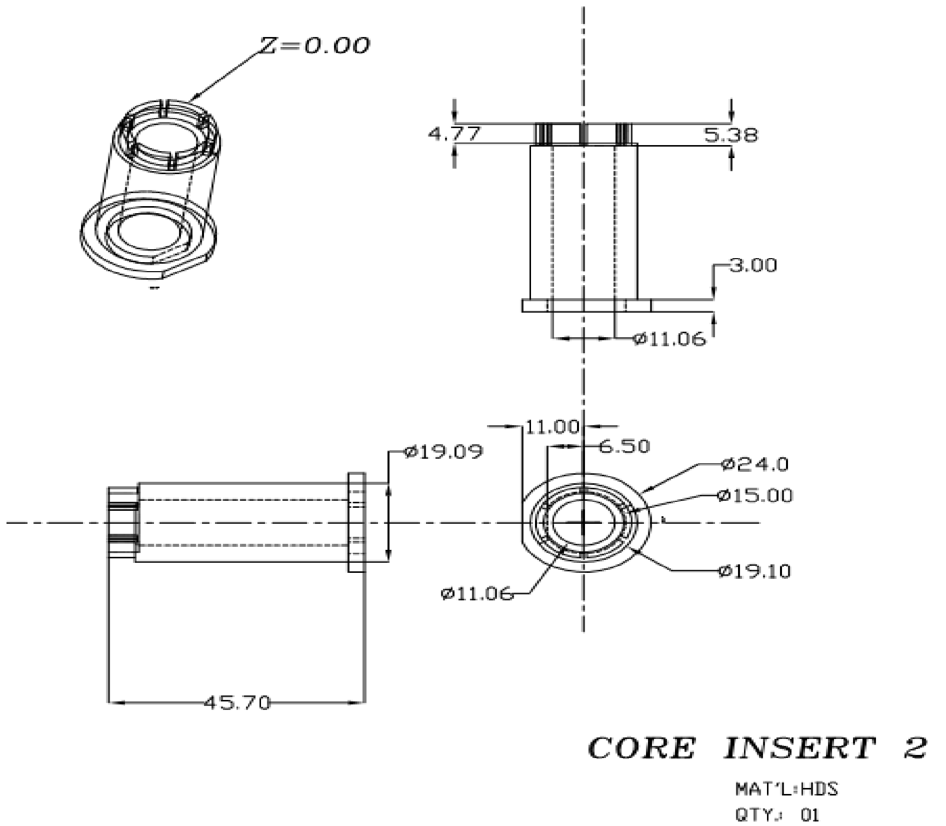
WEDGE

MAT'L:□HNS
QTY.: 01

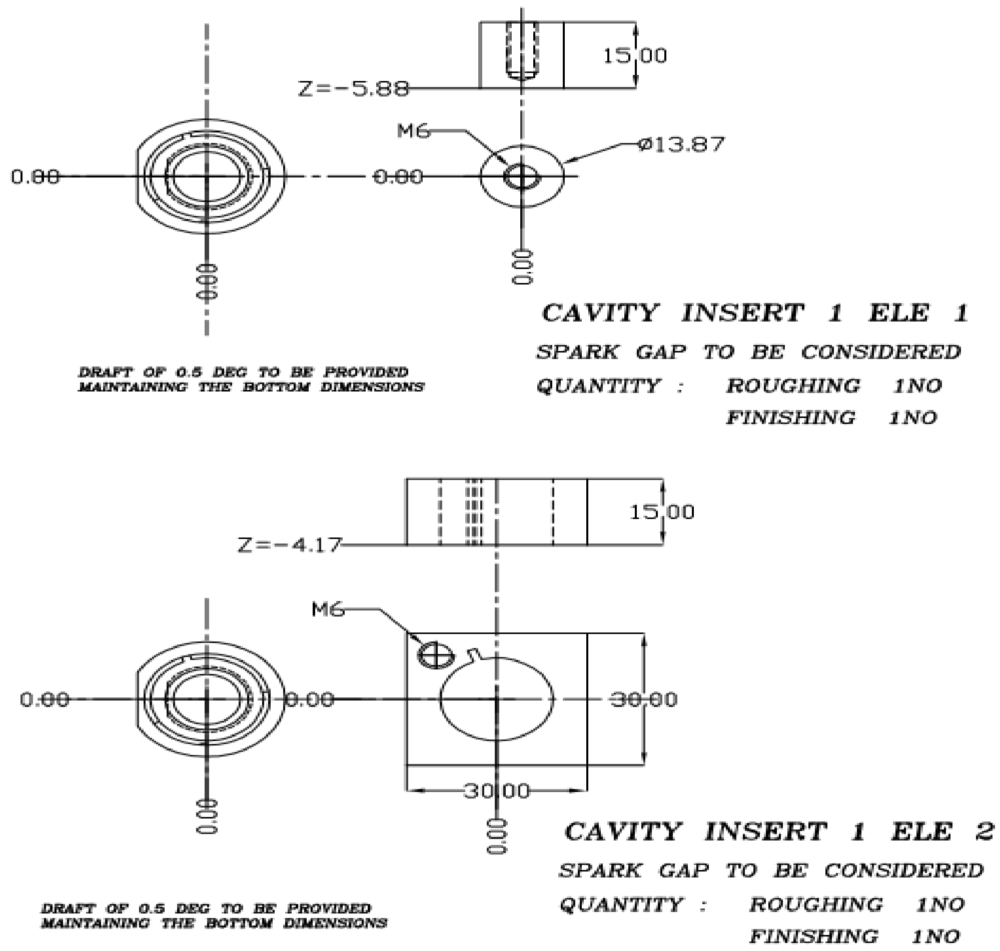
Drawing-21



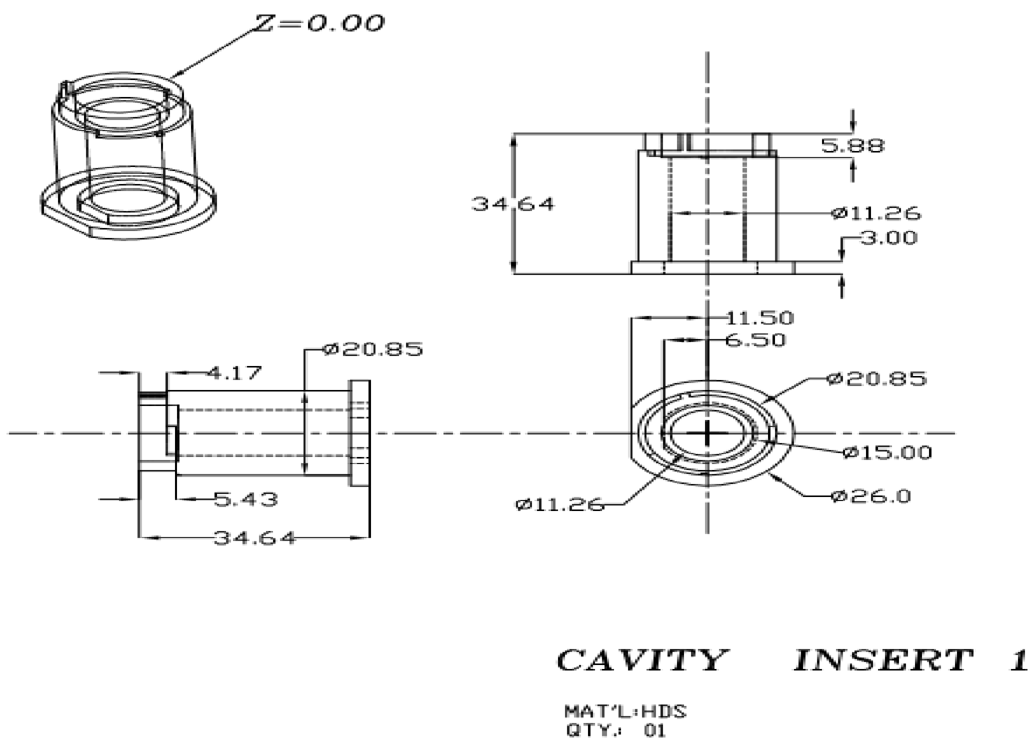
Drawing-22



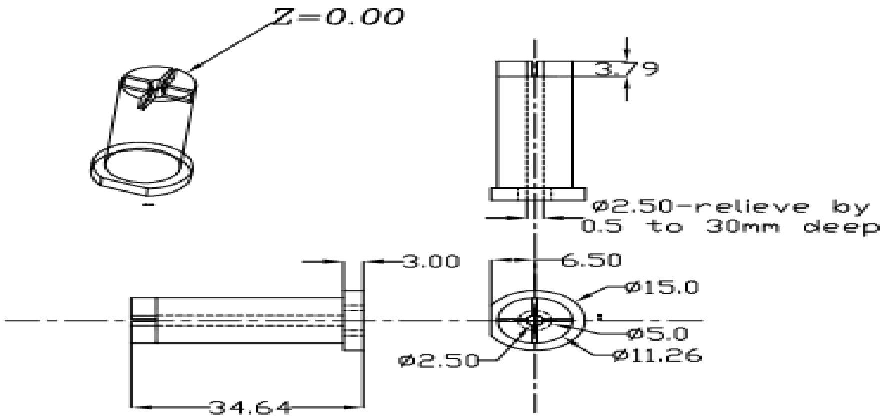
Drawing-23



Drawing-24

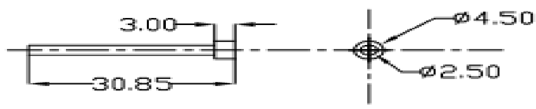


Drawing-25



CAVITY INSERT 2

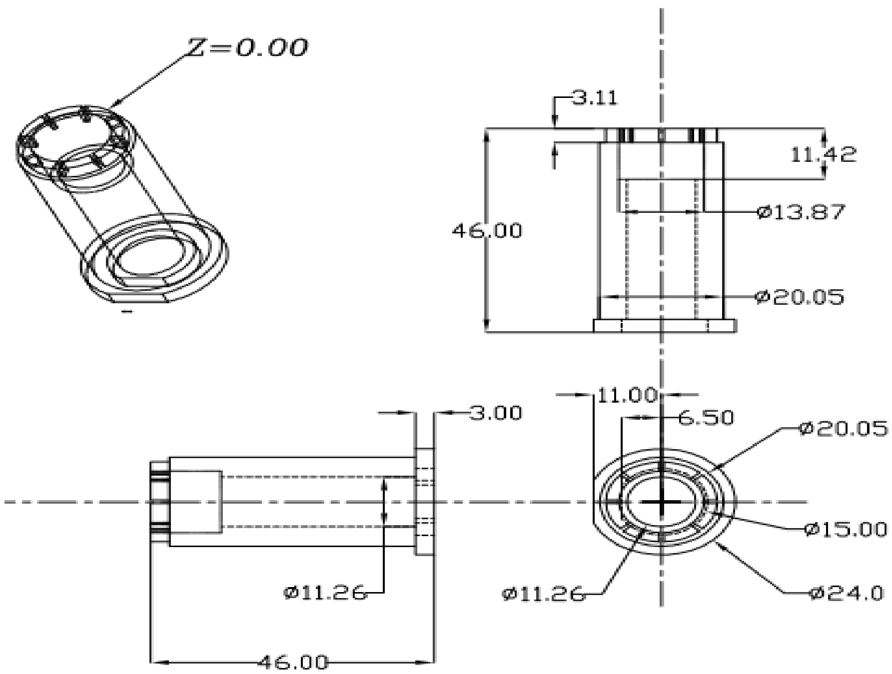
MAT'L:HDS
QTY.: 01



CAVITY PIN

MAT'L:HDS
QTY.: 01

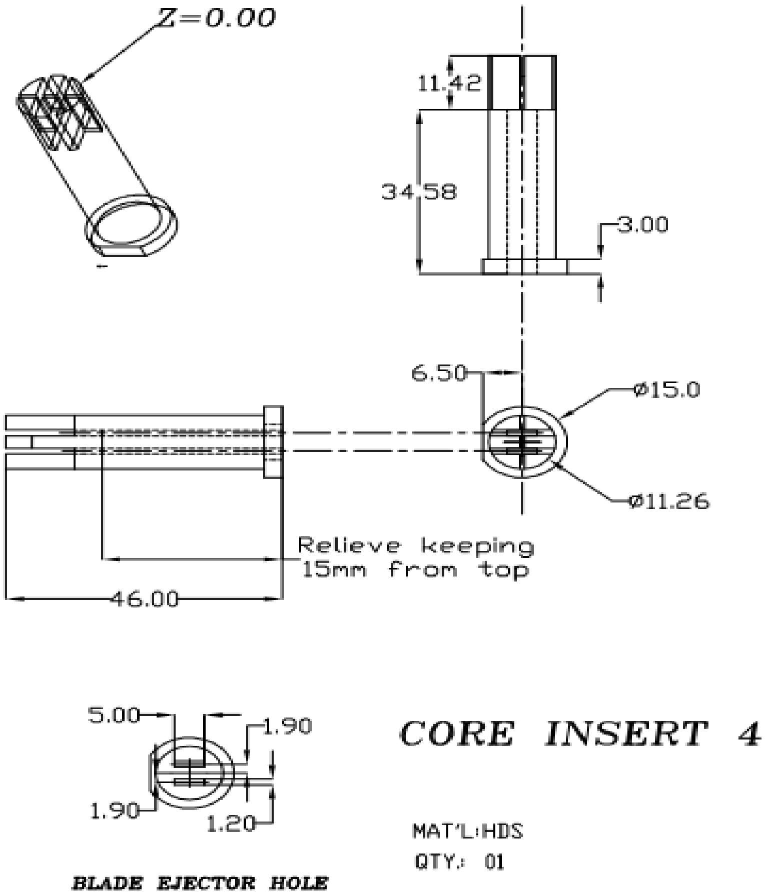
Drawing-26



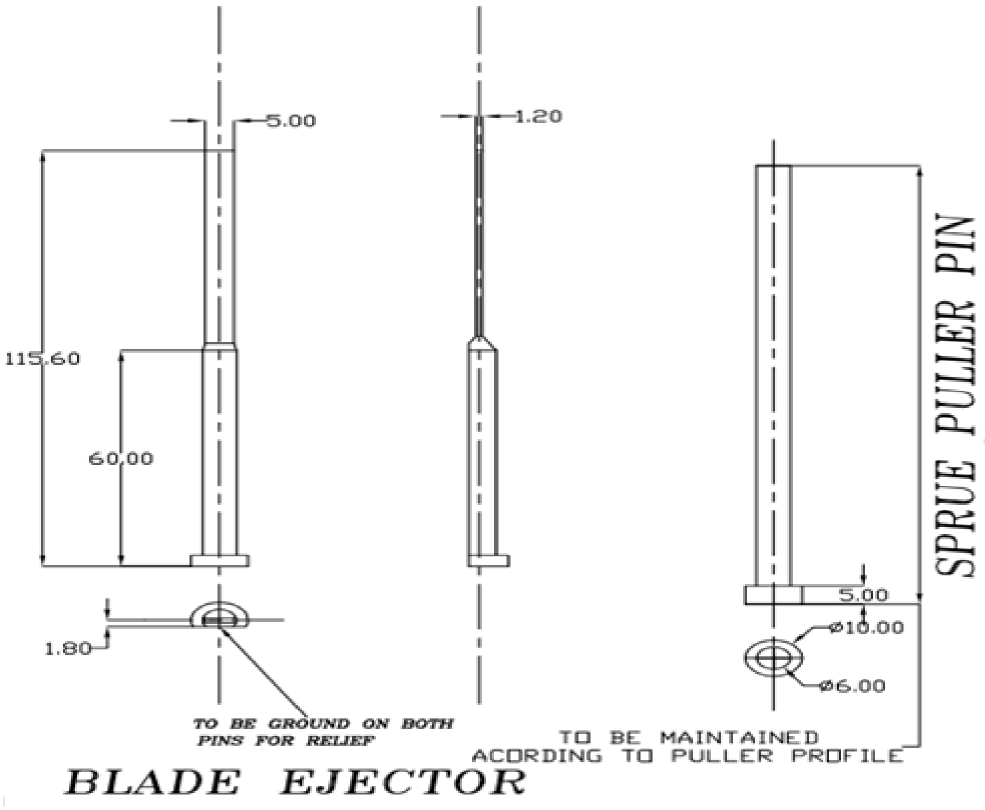
CORE INSERT 3

MAT'L:HDS
QTY.: 01

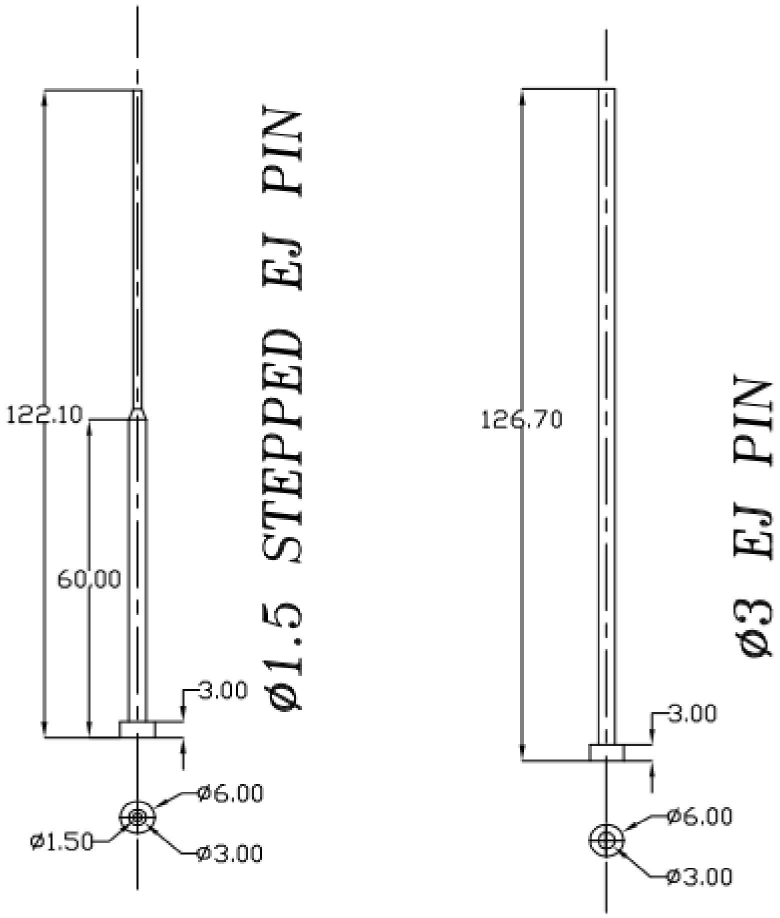
Drawing-27



Drawing-28



Drawing-29



Ø1.5 STEPPED EJ PIN
122.10
60.00
3.00
ø6.00
ø1.50
ø3.00
Ø3 EJ PIN
126.70
3.00
ø6.00
ø3.00

Drawing-30

ABOUT THE BOOK

Injection moulding, one of the most popular commercial manufacturing techniques in plastic industry, is an automated, highly cost-effective, precise and competent manufacturing technique having ability to produce complex design products. The design of an injection mould is an integral part of the plastic injection moulding technique which affects the quality of the final product. This book is a stepwise guide to design, manufacturing and validation of an injection mould for 'Rotor and Cover' of a plastic component used in a particular model of a two wheeler. It is very useful for researchers and the people who are working in the area of tool design and manufacturing.

ABOUT AUTHOR

Dinbandhu Singh was born in Sohagpur, a small village in Gopalganj District, Bihar, India. He did his schooling from Gita Niketan Awasiya Vidyalaya, Kurukshetra, Haryana. He is an M. Tech in Tool Engineering from R.V. College of Engineering (2011) and B. Tech (2009) in Mechanical Engineering from G. Pulla Reddy Engineering College (Autonomous), Kurnool, Andhra Pradesh. His teaching career started at Al-Habeeb College of Engineering & Technology, Hyderabad, Telangana (then Andhra Pradesh) and later worked at various reputed institutions across the country. Presently, he works as an Assistant Professor in Department of Mechanical Engineering at Vidya Vihar Institute of Technology, Maranga, Purnea, Bihar. He has more than 06 years of teaching experience. His research interests are focused on Material Sciences/Composite Materials. He has published/presented/contributed more than 10 research papers in various international journals and conferences of their repute. He is associated with many academic bodies & NGOs in different capacities. He can be mailed at dinosingh@hotmail.co.uk

ISBN 978-81-930928-3-5

9 788193 092835