International Journal for Ignited Minds (IJIMIINDS)

Design of Injection Mould Tool for Rubber Seal Used in Water Heater

Karthik G L[a] & Dr. Ramegowda D[b]

[a]*PG Student, Dept. of PG studies, Govt. Tool Room & Training Centre, Karnataka, India.*
[b]*Principal, Dept. of PG studies, Govt. Tool Room & Training Centre, Karnataka, India.*

ABSTRACT

The rubber products are having high demend in this modern world, these rubber are processed by various methods depends upon the type of the rubber used. Injection moulding has more advantages comprise reduced labour cost better dimensional control and shorter cure times than transfer mould and compression mould. So, our choice is injection moulding process for moulding given component called seal. Injection moulding is a plastic and rubber manufacturing process, where in plastic and rubber that have been compressed into plastic pellets are fed into an injection moulding machine. These machines mould the pellets into the correct shape and add additives to give them the correct color and texture. This paper presents the conceptual design of rubber injection mould. The method represented for the design of four-cavity injection mould. The material used for the production of the component is EPDM. The technique is incorporated to produce a good quality component considering the ease of manufacturability and Positive ejection of the component within the minimum possible time and cost. Any product to be manufactured invariably requires machines and tool. Tool design and development is a specialized and critical area. The tool design should match the machine specification and should be accurate and economical for successful life of a component or product.

Keywords *- Injection mould, EPDM (ethylene propylene diene monomer), Ejection system.*

1. INTRODUCTION

The rubber products manufacturing industry has been growing very rapidly in recent years. This growth will be increased by the tendency to substitute rubber for plastics, which is appearing throughout the world. Injection moulding has more advantages comprise reduced labour cost better dimensional control and shorter cure times. This process is improved for elastomer compound. The conceptual design of injection moulding part is a highly iterative process with consideration of customer needs, part design requirements, material selection, and mold design features, mold making processes, moulding equipment and production economics.

2. INJECTION MOULDING

Moulding is a cyclical process, each cycle comprises several operations like, feeding, melting, and homogenization of polymer grains inside the plasticizing cylinder mould closing, injection under pressure of melt in mould's cavities and cooling or heating of polymer inside the mould, mould opening and ejection of moulded piece.

In fig 1, shows time influence for each parts of cycle. It is necessary to realize, that rubber injection moulding cycle is several times longer than for thermoplastics.

During injection moulding process, melt is subjected to more severe processing conditions than during compression or transfer moulding. Values of temperatures, pressures, and shear stresses are higher, though cure times are shorter in rubber compound. Control over process variables can be more precise.

The cycle time can be minimized by independently controlling barrel temperature, screw speed, mould temperature and injection pressure. That is the reason why the injection moulding process should be improved and understood.

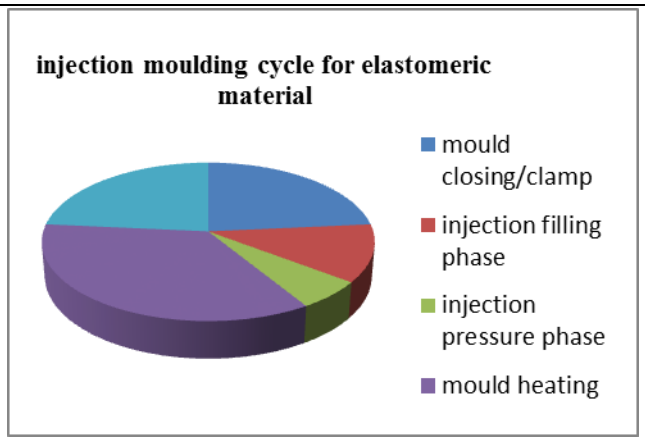

Fig 1: Injection Molding Cycle.

- Mould close and clamp, (few seconds depends on machine speeds)
- Injection Fill (speed) phase, (few seconds)
- Switch over and Pack (pressure) phase, (few seconds)
- Heating time (40 to 60% of cycle time)

Factors to be considered during design of any molding tool

- Design and material of components
- Selection of injection moulding machine
- Number of cavities
- Type of tool
- Selection of parting line
- positioning of core and cavity
- Ejection system
- Cooling elements
- Tool life

All above factors has to be considered during the designing which affects directly or indirectly. In order to control the processing temperature, pressure, melt velocity, filling time, molding material, parting line selection, feed system, mould cooling, ejection system etc. by proper consideration of the factors greater control over the process and process parameters can be obtained.

3. EPDM MATERIAL

The actual properties of a particular EPDM material depend on the blend ratio of the three major constituents (ethylene- propylene, diene and peroxide curing). Ethylene (45% to 85%) gives the higher the loading possibilities of the polymer, better mixing, and extrusion. Diene gives serve as cross linking when curing with sulphur and resin; with peroxide cures, the diene (or third monomer) functions as a coagent, which provides resistance to unwanted tackiness, creep, or flow during end use (2.5% to 12%). Peroxide curing these polymers gives a higher crosslink density compared with their amorphous counterpart. The amorphous polymer is also excellent in processing. This is very much influenced by their molecular structure. It is excellent resistance to temperature extremes, sunlight, moist. It has good resistance to ketones, ordinary diluted acids and alkali [6, 7]. It used as washers, belts, electrical insulation and solar panel heat collectors. It has greater flexibility even at lower temperatures and resistant to cracking, good UV stability EPDM seals are used in solar heaters to effectively prevent air, moisture, water and dust from entering into the heaters. EPDM rubber is an excellent material to protect solar heaters from damage [8]. EPDM material is filling melt chamber of range of temperature is 80-90 degree Celsius to start its melt by heating catridge around the melt chamber as shown in fig3 then injected into mould of temperature range is in the range of 150-155 degree Celsius to get curing of component of material based on its properties. Mould temperature maintained by temperature controller is SELEC TC513 with importance of better curing time by controlling over process variables is barrel temperature, mold temperature can be more precise.

4. METHODOLOGY

Components are modeled using the solid works 2013 software. Component has a Hallow cylindrical structure with dimensions as follows: Outer diameter 58.87 mm, Inner diameter 53.95 mm and length 54.36 mm as a molded part was used. Component has continuous ribs as shown in the fig 2 and other details of model are given below.

Component Name: rubber seal
Material: EPDM
Shrinkage: 1.5-2%
Moulding type: four Cavity injection mould tool
Tonnage required: 100 tonnage capacity
Density: 1.5 g/cm^3
Projected Area of component: 1682.665 mm2 (From CAD Model).

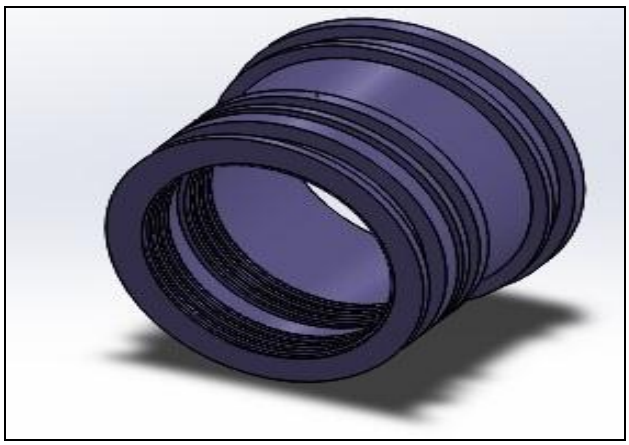

Fig 2: Component 3D Model.

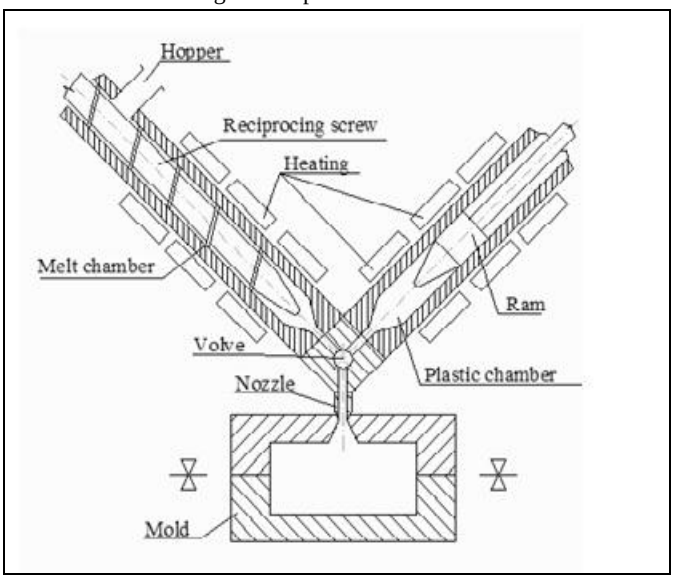

Fig 3: Schematic of rep rt9 Vertical Injection Mould Machine.

Selection of injection moulding machine is depends on clamping force required to clamping the mould of number of cavity by considering of total weight of component. Here clamping force is about 50.48 tons for given rubber component but sometime selection of capacity of clamping force of machine changed by availability with considering of given obtained clamping force is less than capacity of available machine. Generally better choice of rubber injection mould machine is REP RT9 as shown in fig 3.

5. BASIC DESIGN OF INJECTION MOULD TOOL

This section describes the design aspects and other considerations involved in designing the mould to produce four components.

Three design concepts had been considered in designing of the mould and are as follows:

- Two-Plate Mould (concept-1) with single cavity as shown in fig 4.

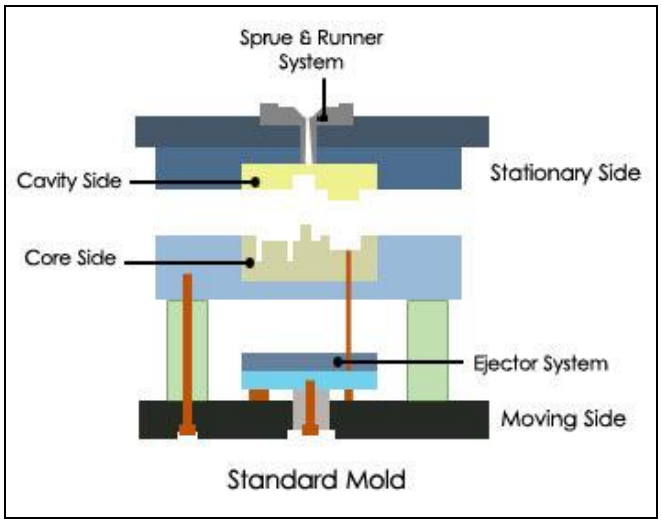

Fig 4: Standard Two Plate Mould (Concept-1).

Since the number of components is more it may not be applicable.

- Two-Plate mould (concept-2) with four cavity and side core actuated by hydraulic system as shown in fig 5.

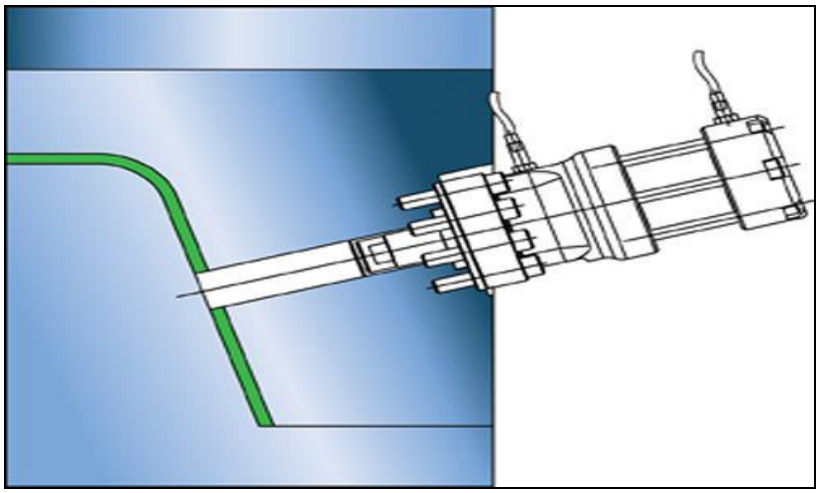

Fig 5: Hydraulic Actuation System.

This method not applicable due to higher cost of hydraulic system.

- Two-Plate mould (concept-3) four cavities with finger cam actuation as shown in fig 6.

In designing of the mould for given component as shown in fig 2, the third design concept had been applied. Various design considerations had been applied in the design. The mould is designed based on the platen dimension of the plastic injection machine used. There is a limitation of the machine, which is the maximum area of machine platen is given by the distance between two tie bars. Therefore, the maximum width of the mould plate should not exceed this distance.

6. BASIC DESIGN OF INJECTION MOULD TOOL

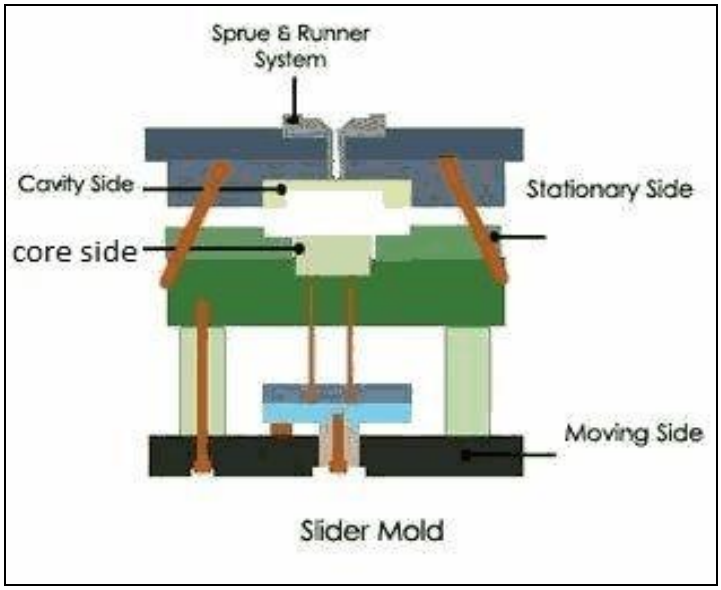

Fig 6: Standard Two Plate Mould (Concept-3).

This consists of two halves fastened to the two platens of the moulding machine's clamping unit. When the clamping unit is opened, the mold halves separate. The parting surface is the surface shared by the two mould halves. A cooling system is required for the mould. This consists of an external pump connected to passageways in the mould, through which water is circulated to remove heat from the hot rubber.

Here, ejection system has been removed to reduce the cost of given mould. Component is taken out of the core as shown in fig 11 by manual dragging. Manual dragging of component is will not affected the inner surface of the component region of its elastic properties, given four cavity injection mould as shown in fig 7 to be made for producing given seal components.

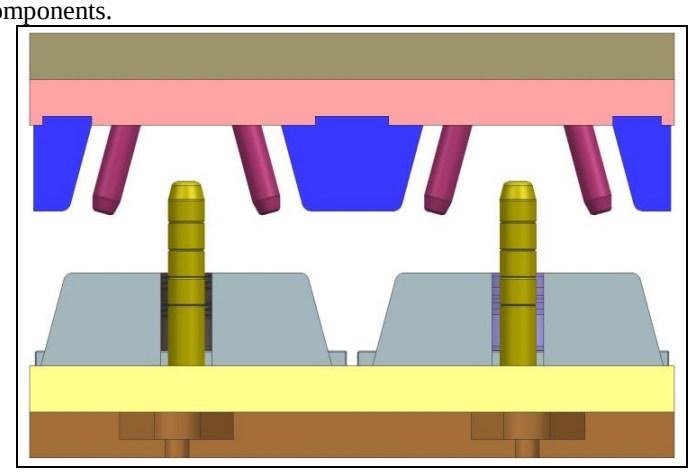

Fig 7: 3D Model of Our Injection Mould with Finger Cam Actuation Method.

Generally cavity is impression for filling of material in the side of fixed half. Here given component have number of ribs on outer surface and inner surface of ribs is affected to easy ejection by general type of mould. Because of outer surface of ribs on component is not going with moving half during opening. Overcome of above difficulties, we have to use two side cores for outer surface of given component called seal and adopting finger cam actuation for actuating side cores during ejection.

6.1 Core and Cavity

Mould separate into two halves to form the core and the cavity, which permit the part to be extracted. Generally cavity is impression for filling of material in the side of fixed half. Here given component have number of ribs on outer surface and inner surface of ribs is affected to easy ejection by general type of mould. Because of outer surface of ribs on component is not going with moving half during opening. It stick with fixed half, getting a chances of breaking of ribs on outer and threads on inner surface of component called grommet during opening of mould of moving half side consists of core plate with core as shown in fig 10 and fig 12.

Overcome of above difficulties, we have to use side cores for outer surface of component as shown in fig 10 on either side of core with core plate to create the impression in the side of moving half of mould as shown in fig 13.

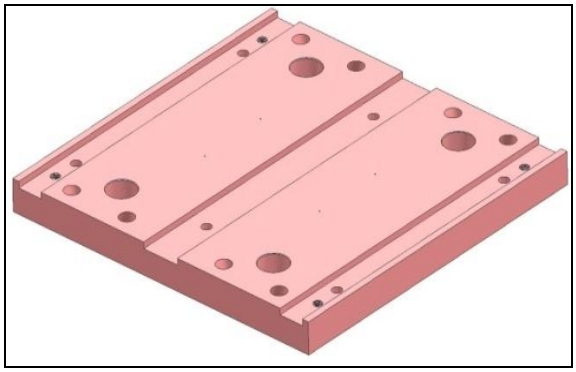

Fig 8: Cavity Plate.

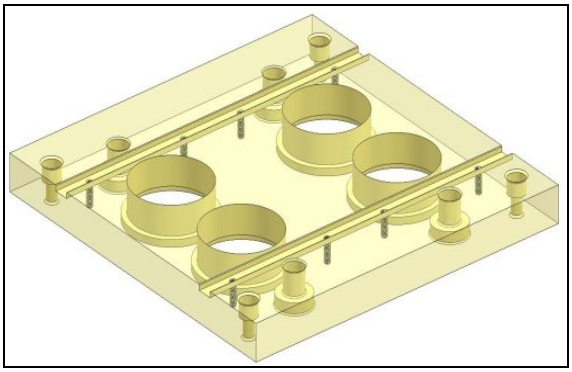

Fig 9**:** Core Plate.

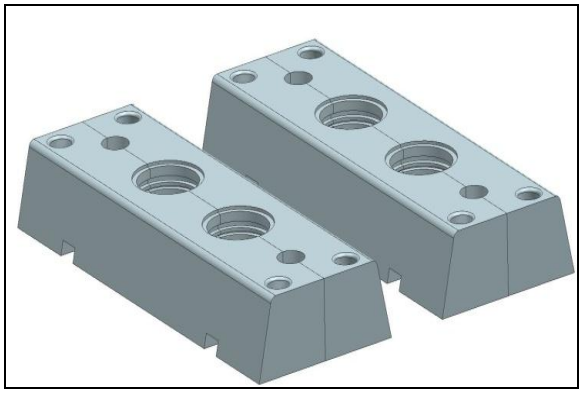

Fig 10: Side Core.

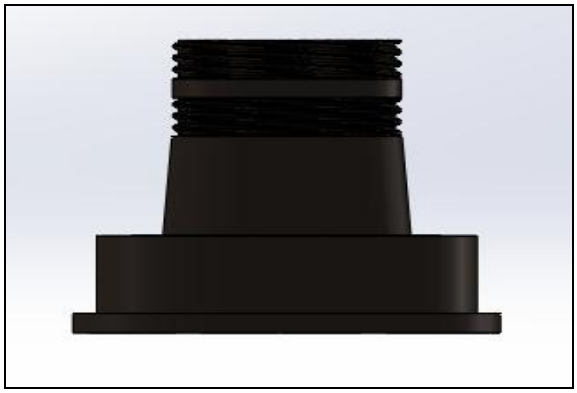

Fig 11: Core.

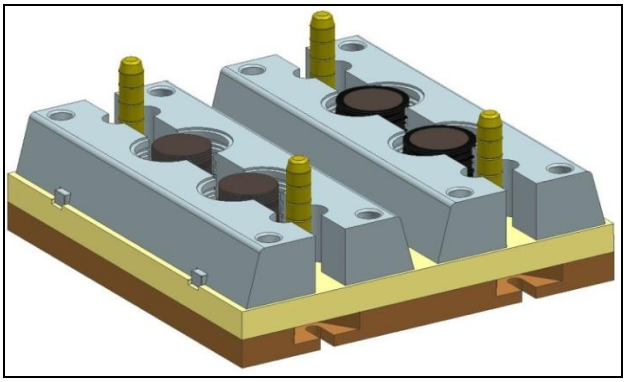

Fig 12: Moving Half Side.

6.2 Gate

A small opening which connects the runner and cavity is called as gate. The gate is of pin point, leaving a very negligible mark on the surface of the component. When designing injection mold, size and location of the gate is one of the most important considerations for correct moulding of the part. Incorrect gate positioning can result in uneven filling, over packing, and dimensional instability of the part. Incorrect selection of the gate size can result in an inability to fill the part, inability to thermally shut off the gate, dimensional instability or internal stresses in the part.

The most common gate type is direct pin point gating, which offers the simplest construction and high reliability and due to complexity of component design number of gating is provided in each ribs.

7. EJECTION SYSTEM

An ejection is very much required in order to eject the cured component from the tool without causing any damage to component. The design of ejection system is one of the major factors, how efficiently the tool will be in production. Side core is getting away from core followed the actuation of finger cam as shown in fig 13 during

mould opening in such a way that natural shrinkage of moulding causes the component will stick onto and remain on core or moving half side as shown in fig 12. Then component is taken out of the core by manual dragging. Manual dragging of component is not affected to inner surface of component reason of its elastic properties.

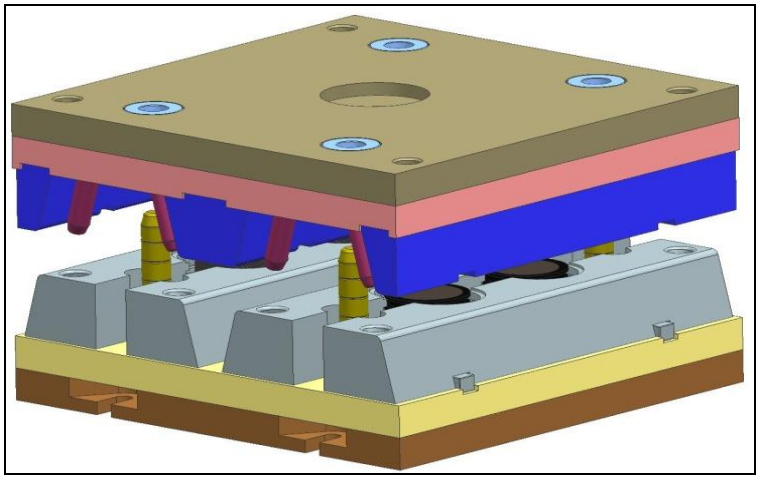

Fig 13**:** Mould Opening With Finger Cam Actuation Method.

8. CONCLUSION

The work deals with the Concept of designing of injection mould tool for rubber seal. Quality product at considerable cost can be achieved by injection mould tool with finger cam actuation. By proper supply of temperature and pressure will overcome the effects on the part being produced.

REFERENCES

1. Technical directory on design and tooling for plastics, CIPET.
2. R.G.W. Pye, Injection mould design, New Delhi, 4th Edition, 2000
3. G R Nagpal, Tool Engineering and Design, Khanna Publishers, 2006.
4. GT&TC standard data hand book