International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 3, No. 4, October 2014

Research Paper

CONCEPTUAL DESIGN OF TWO PLATE INJECTION MOULD TOOL FOR FIVE PIN DAIMLER REGULATOR

Kiran Tom Thomas[1]* and Ramesh Babu K[1]

*Corresponding Author: Kiran Tom Thomas, ✉ kirantomthomas@yahoo.co.in

The paper consists of designing a two plate injection mold tool for five pin regulator. The required component study was done before the design. The components drawing are carefully scrutinized to extract the maximum possible amount of information. Solid modeling of components is done using Creo-Parametric 2.0 considering all the critical dimensions. Proper material selection and proper combination of alloys is selected for manufacturing of mould.

Keywords: Five pin regulator, Creo-parametric 2.0 software, Solid modeling

INTRODUCTION

Plastic injection moulding is one of the most common methods of converting plastics from the raw material form to an article of use. This process is most typically used for thermoplastic materials which may be successively melted, reshaped and cooled. Injection moulded components are a feature of almost every functional manufactured article in the modern world, from automotive products through to food packaging. This versatile process allows us to produce high quality, simple or complex components on a fully automated basis at high speed with materials that have changed the face of manufacturing technology.

Injection moulding technology continually develops, with major milestones including the introduction of the first thermoplastic materials, the reciprocating screw design, engineering materials, the introduction of microprocessors for machine control; computer aided engineering flow simulation software, and recently the application of expert systems for optimized machine setup. This work presents a vision of injection moulding for the next millennium to address current industry needs, and then describes some needed developments to convert that vision to reality. The ultimate aim is a machine that produces no scrap material and increased product quality with reduced labour skill requirements,

1 Department of P.G. Studies, G.T&T.C, Mysore, Karnataka, India.

low energy consumption, and minimal maintenance. Nowadays plastic consumption is more than metallic products due to its ease of production and high performance. Worldwide plastic consumption is at least 125,000 million pounds (by weight). About 36% is processed by extruders, 32% is processed by injection moulding, 10% by blow moulding, 6% by calendars, 5% in coating, 3% in compression moulding, 2% in power form, and 6% using other processes these percentages do not correlate with the number of machines used. Major advantages of using plastics include formability, consolidation of parts, and providing a low cast to performance.

OBJECTIVE OF THE PROJECT

The prime objective is to design the Injection Mould tool, produce good quality economically.

- Analyse if the part is mouldable, detect and fix problematic zones. The study of selected materials has been done, to know its physical and mechanical properties associated with moulding material and moulding characteristics that influence tool design.
- Apply a shrinkage that corresponds to the part material, geometry and moulding conditions.
- Make conceptual design of two platemould.
- Populate a mould assembly with standard components such as mould base, ejector pins, sprue bush, screws, fittings and other components creating corresponding clearance holes.
- Check for any fouling of ejector pin holes, cooling holes, screw holes.
- The design of injection mould tool is done using “creo-parametric 2.0” software.
- Before manufacturing the tool, the mould ability and quality of the component is checked using Moldflow Analysis and correction if needed is implemented in the design.
- Creating manufacturing drawings of each component of the tool for manufacturing.
- The mould flow advisor analysis is carried out to set injection location, confidence of fill and possible type of gates for the component rmance ratio.

PROBLEM FORMULATION

The work deals with Design, flow analysis and manufacturing of two plate injection mould tool for five pin regulator. This is a component used in the trucks as connectorhousing for connecting various pins.The costumer requirement of component for every month is 1000 components. The Injection moulding machine available in the firm is 70 tonnage capacities. For this optimized design was done.

SCOPE OF THE PROJECT

The scope of the study for this project consists:

- Study of component.
- Conceptual design and calculation.
- Core and cavity extraction.
- Analysis for the component.
- Preparation of assembly and mould base drawings.
- Preparation of detailed drawing of core and cavity with all other mould elements.

- Tool manufacturing, assembly and inspection.
- Try-out and rectifying the defects if found.

METHODOLOGY

Methodology is a systematic approach for the realization of total task. It consists of the following detail:

- Study of the component: The study of the component is the most important and the first step for the designer. The component drawings are carefully scrutinized to extract the maximum possible amount of information. The important information available is the critical dimensions, line of draw, parting line, suitable ejection system, and required side core.
- Solid model of the component: Solid modelling of component is done using “creo-parametric 2.0” considering all the critical dimensions.
- Step by step design Calculations: It is carried out to determine the various design parameters that determine the final mould clamp force required during injection, number of cavities, wall thickness of inserts, guide pillar design, design of feeding system, cooling calculations.
- Core and Cavity extraction: Extraction is done by providing proper shrinkage, tolerance is provided to the dimensions to which a cavity and core should be manufactured in order to produce a part of desired shape and size. The usual way to decide on the amount of shrinkage is to consult data supplied by the material manufacturer. While designing shrinkage is provided depending on the type of plastic material to be chosen for injection moulding. A thicker piece will have a higher shrinkage value compared to thinner section.
- Solid modelling of the tool: 3-D modelling of the entire mould is done using “creo-parametric 2.0” . The required dimensions are determined by calculation, which is used during modelling of the tool.
- Mould flow analysis: Moldflow analysis is carried before tool manufacturing to determine input parameters to the injection moulding machine etc.
- Moldflow advisor analysis is carried before tool manufacturing to select best gate location and confidence of fill.
- Tool try-out and troubleshooting: After the tool is manufactured and assembled, the tool is tried to see that component produced is true to the geometry and dimensions specified by the customer. Try-out is a procedure where the tool is subjected to actual working condition and the performance of the tool is noted. After the tool has been tried out, the component is thoroughly inspected for various defects. If any defects are found, it is suitably reworked.
- Cost estimation: Mould cost estimation is crucial especially for small and medium batch of production runs where the cost of a mould represents a significant percentage of the product development cost. It is the probable cost of an article before the manufacturing starts. By compiling statement of the quantities of the material required and production time required, the probable cost is computed.

CALCULATION OF THE 3D MODEL

Part Details

Name of the component: Daimler Regulator

Material: PA 66 With 33% GF(ZYTEL 70G33L BK031)

Shrinkage: 0.2-1.1%

Volume of component: 18.037 cm^3

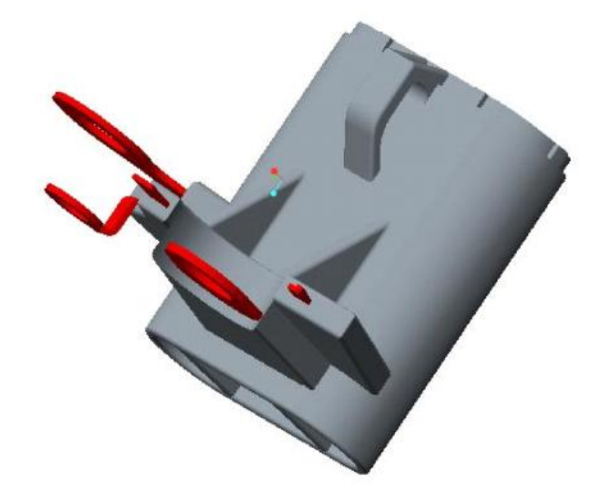
Density of material: 1.39 g/cm^3

Weight of the component: 23.27 g

Number of cavities: Single cavity

Projected area of component: 101.28 cm^2 (from CAD model).

Figure 1: Component Details



Weight of Moulding

Actual weight of component, (W)

$W = ° \times V$ ...(1)

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

W = Actual weight of the component, g

° = Density of plastic material, gm/cm^3 = 1.39 g/cm^3

V = Volume of the component, cm^3 = 18.037 cm^3 (CAD model).

$W = 1.39 \times 18.037$

$W = 25.07$ g

Total weight = W x Number of cavities

Total weight = 25.07 x 1 = 25.07 g

The weight of the sprue and the runner related to the moulding must not generally be neglected. This should be considered in the formula while determining the moulding weight. The moulding weight should be substituted in the formula and multiplied with the multiplication factor (M.F).

Total weight of single component with feed system = 25.07 x 1.05 = 26.325 g

Clamping Tonnage

Clamping tonnage required = Total Projected area of the mould × Cavity pressure x no. of cavities ...(2)

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

Injection pressure required for processing Polyamide 66 with 33% glass filled to produce an engineering part is 1000 kg/cm^2 (maximum).

1/2 Of injection pressure, as cavity pressure for easy flow materials, 1/3 of injection pressure, as cavity pressure for viscous materials. polyamide 66 with 33% glass filled has good flow-ability, hence 1/2 of the injection pressure, may be assumed as the cavity pressure.

Tonnage required for the component =

Total projected area x 1/2 Injection

pressure x number of cavities

= 101.28 × (1/2 × 1000) x 1

Tonnage required for the component = 50640 Kg

Factor of safety of 1.3 (30% of actual tonnage) = 65,832 Kg

Minimum machine tonnage required = 65.832 tonnes = 645.81 N

It is suggested that the available machine is Mathmann 70 T Machine.

Mathmann 70 T Machine is selected.

Plasticizing Capacity (p_s)

Plasticizing capacity of the machine is calculated as follows,

Rated plasticizing capacity of the material is:

$$Plasticizing\ rate = with\ Material\ PS \times \frac{Q_{PS}}{Q_{PP}} \quad ...(3)$$

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

Plasticizing rate of polystyrene = 16.6 g/sec

q_a = Total heat of polystyrene = 57 cal/g

q_b = Total heat of polyamide 66 with 33% glass filled = 135 cal/g

$$P_S = \frac{16.6 \times 57}{135}$$

P_S = 7.00 g/s = 25.2 kg/hr

Plasticizing capacity of the machine for polyamide 66 with 33% glass filled is 25.2 kg/hr;

Machine capacity is 25.2 kg/hr; therefore machine can be used safely. (Machine data)

Total Weight of material required per hour = 1.98 kg/hr;

Machine Plasticizing capacity for PA = 25.2 kg/hr.

Machine selection is safe.

Shot Capacity (SC)

The capability of machine is normally expressed in cubic centimeters of swept volume the injection cylinder. The shot is, therefore, the mass of this volume of plastic melt at the plasticizing temperature and pressure. Thus,

Shot capacity (*kg*) = *Swept volume* x *Density of material* x *Cont.* ...(4)

where, Constant = correction factor for percent volume expansion of the plastic at the moulding temperature for PA = 0.9 (Crystalline materials).

The screw type is normally rated in terms of swept volume of the injection cylinder = 67.9 cm^3 (from Machine Specification).

Density of material = 1.39 g/cm^3

Shot capacity (g) = 67.9 x 1.39 x 0.9 = 85 g

Shot capacity of the machine with PA is 85 g. Since the shot weight of the component is 26.325 g, the design is safe and production of the component can be carried out without any restrictions.

Determination of Number of Cavities

Calculation of Number of Cavities Based on Shot Capacity

N_s = 0.80 × (*Shot capacity*/*Weight of component*) ...(5)

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

Shot Capacity of the machine for PA is 85 g

= 0.80 x (85/26.325) N_s = 2.58 = 2 cavities

Depending on shot capacity the mould can be designed to accommodate two cavities. But since the part is of very complex shape single cavity is selected.

Calculation of Number of Cavities Based on Plasticizing Capacity

$$N_p = \frac{0.80 \times Ps \times Ts}{Ws} \quad ...(6)$$

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

Ps = Rated plasticizing of polyamide 66 with 33% glass filled in grams per hour

Ts = Cycle time in seconds

Ws = Weight of the component in grams

= (0.80 x 25.2 x 48)/26.325 = N_p = 36.7 = 36 cavities

Depending on plasticizing capacity the mould can be designed to single cavity. Hence the design is safe.

Based on Clamping Force

Based on production rate the number of cavities can be determined

Clamping force = 65.832 tonnes

Projected area = 101.28 cm^2

Cavity pressure = 750 kg/cm^2

Factor of safety = 1.05-1.2 for the injection moulding machine N_c = 2.47 = 2 cavities

From the above calculation it is clear that a mould with single cavity is a safe design and Mathmann 70 T Machine of can accommodate easily for Single cavity, which also suits customer specification.

Calculation for Wall Thickness of Core/Cavity Inserts

Insert wall thickness, ¢,

$$¢ = \sqrt[3]{\frac{CPd^4}{Ey}} \text{ mm} \quad ...(7)$$

C = Constant based on ratio of cavity length to depth = 0.140

P = Cavity pressure = 1000 kg/cm^2

d = Depth of cavity wall = 2.5 cm

E = Modulus of elasticity = 2.1 × 10^6 kg/cm^2

y = Permissible deflection for the insert = 0.015 cm

$$¢ = \sqrt[3]{\frac{0.140 \times 1000 \times 2.5^4}{2.1 \times 10^6 \times 0.015}} = 0.416\ cm = 4.2\ mm$$

Minimum wall thickness of core/cavity inserts, *t* = 15 mm. For safe design minimum required wall thickness is 15 mm. Here we get 4.2 mm. Hence the design is safe.

Design of Guide Pillar

Guiding diameter of the guide pillar, d_p

$$d_p = \sqrt{\frac{4 \times Q}{® \times N_p \times f_s}} \quad ...(8)$$

Q = Side thrust

N_p = Number of guide pillars = 4 numbers

f_s = Working shear stress for the guide pillar material, kg/mm^2

Side thrust,

$$Q = d_i \times h \times P_c \quad ...(8.1)$$

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

d_i = Height of the core, mm

h = Maximum side of the core, mm

P_c = Pressure in the cavity, kg/cm^2

Q = (2.8 x 3.5 x 1000) = 9800 Kg

Substituting the value of the side thrust induced, we get the minimum diameter of the guide pillar, *d*,

$$d_p = \sqrt{\frac{4 \times 13720}{® \times 4 \times 1400}} = 1.492\ cm = 15\ mm$$

For manufacturing the guide pillar diameter is considered as 24 mm.

From the above calculation it is found that 15 mm is required. For ease of manufacturing and assembly of tool we have considered 24mm diameter guide pillar.

Feeding System Design

Runner Design

The runner diameter is calculated by the following formulae

$$D = \frac{\sqrt{W} \times \sqrt[4]{L}}{3.7} \quad ...(9)$$

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

where,

W = Weight of the component = 23.27 g

L = Length of the runner = 30 mm

$$= \frac{\sqrt{23.27} \times \sqrt[4]{30}}{3.7}$$

Substituting the values in Equation Diameter of the runner.

D = 3.05 mm

Therefore diameter of the runner = 3.05 mm

Gate Design

According to the size and shape in this design, submarine gate is employed to feed the component.

To find gate width

$$w = \frac{h \times \sqrt{A}}{30} \quad ...(10)$$

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

where,

h = Constant = 0.7 for polyamide 66 with 33% glass filled material.

A = Total surface area of the cavity = 10128 mm^2

$$W = \frac{0.7 \times \sqrt{10128}}{30} = 2.34\ mm$$

To find Gate depth

Gate depth, h_g = Avg width of gate/Avg thickness of component

where,

Avg width of gate = 1.29 mm

t = thickness of the component = 2 mm

Substituting these in the equation we have,

h_g = 1.29/2 = 0.645 mm

h_g = 0.645 mm is selected

Mould Cooling Calculations

Heat to be transferred from mould per hour (Q):

$Q = n$ x m x q_b ...(11)

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

where,

Q = Heat to be transferred per hour (cal/hr)

m =Mass of the plastic material injected into the mould per shot (g) = 26.325 g

n = number of shots per hour (75 shots/hr)

(Number of shots is taken from Machine data)

q_b = Heat content of plastic material, for polyamide 66 with 33% glass filled =130 cal/g

Q = 75 x 26.325 x 130 = 256.668 KCal/hr

But in practice heat is removed by three ways

- Conduction
- Radiation
- Convection

It is found in practice, that approximately 50% of the total heat input is carried away by the water cooling systems in moulds. Therefore amount of heat removed by cooling water is,

Q_d = 0.5 x Q = 0.5 x 256.668

= 128.334 KCal/hr

Amount of Water to be Circulated per Hour to Dissipate Geat (mw)

Amount of water to be circulated to remove 50% of Heat is calculated as

$$m_w = \frac{Q \times 0.55}{k\,(T_{out} - T_{in})} \quad ...(12)$$

[Reference: Technical Directory on Design and Tooling for plastics, CIPET]

where,

K = Thermal conductivity of water

K = 0.65 for direct cooling

K = 0.5 for indirect cooling

T_{out} = Outgoing water temperature °C

T_{in} = Incoming water temperature °C

S_W = Specific heat of water

m_w = Amount of water required to remove 50% of heat.

Assuming a reasonable temperature difference of T_{out}–T_{in} = 5 °C for water

$$= \frac{269.78}{0.65 \times (5)} = 39.48\ kg/hr$$

= 0.658 lt /min

DISCUSSION

From the above design calculations following details are assumed for manufacturing the tool:

- Total weight of single component with feed system is 26.325 g.
- Minimum machine tonnage required is 65.832 T. It is suggested that the available machine is Mathmann 70 T Machine.
- Total Weight of material required per hour is 1.98 kg/hr and machine plasticizing capacity for PA is 25.2 kg/hr, hence machine selection is safe.
- Shot capacity of the machine with PA is 85 g since the shot weight of the component is 26.325 g, the design is safe and production of the Component can be carried out without any restrictions.
- Based on shot capacity, clamping tonnage and plasticizing capacity number of cavities are selected as one.
- Amount of water circulated per hour to dissipate heat is 0.658 litres/minute.

REFERENCES

1. Glanvill A and Denton E (1965), *Injection Mould Design Fundamentals*, Industrial Press.
2. Herbert Rees (2002), *Mould Engineering*, 2nd Edition, Hanser Gardener Publications.

3. Nagpal G R (2006), *Tool Engineering and Design*, Khanna Publishers.
4. Peter Jones (2008), *The Mould Design Guide*, Smithers Rapra Technology Ltd.
5. Pye R G W (2000), *Injection Mould Design*, 4th Edition, Affiliated East-West Press Pvt. Ltd.
6. Rohinton K Irani, Srinivas Kodiyalam and David O Kazmer (1992), "Runner System Balancing for Injection Molds using Approximation Concepts and Numerical Optimization", Advanced Design Engineering Group, GE.
7. Whelan A (1982), *Injection Moulding Materials*, Applied Science Publishers.