Design and Development of an Injection Mold Tool for Top Cover Bearing used in Automobiles

Ravi Kumar P[1], Kuldeep Kora[2]

[1]M-Tech Tool Engineering Scholar, Government Tool Room and Training Centre (GTTC), Mysuru
[2]Tool Design Engineer, Government Tool Room and Training Centre (GTTC), Mysuru

---***---

Abstract - *Injection Molding is a very old and advanced manufacturing technique in which heated molten polymer is forced into mold opening under huge pressure. The heated plastic in the mold then cools down, solidifies to form desired shaped part. Top cover bearing is a part used to accommodate in automobile suspension systems. Material selected is Nylon 6 with 30% glass filled (black). The grade of the material is SGR 30 and the shrinkage factor of 0.6%. The component and tool designs are done with the help of Unigraphics (NX 11.0). Mold flow analysis is carried out using the software Easy flow advance NX 11.0. Finally, manufacturing, tool trials and corrections were performed.*

Keywords*: *Injection Molding, Top cover bearing, Nylon 6, Shrinkage factor, Easy flow advance

1. INTRODUCTION

One well-liked and effective method for making plastic objects is the use of injection molding. Typically, this method uses molten polymer thermoplastics as the substrate. That is the order in which they originated and cooled. Infusion molding plastics are used in nearly every functional produced item in today's society, from food containers to vehicles.

Top cover bearing is used in the suspension assembly of the automobiles. Investigating the Nylon-6 substance that is used in the piece of equipment is the first step. The initial stage involves a thorough analysis of the substance specs and part dimensions (from 3D CAD). Choice of separating lines, drawing, shrinkage expense, optimal gate placements, runner layout, extraction of cores and cavities, ejected units, and ventilation systems are all included in design.

2. OBJECTIVES

- Study of the component and design
- Tool Design Calculations
- Investigation of mold flow
- Solid component of the tool

3. METHODOLOGY

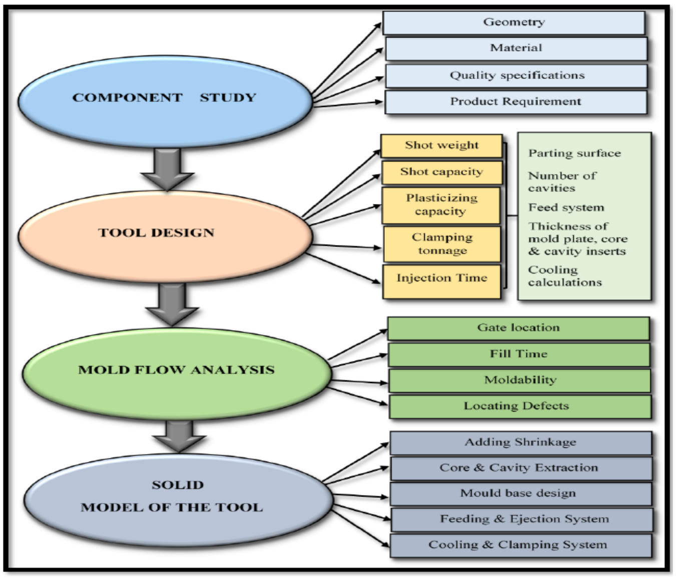

Fig -1: Flow chart of Methodology

4. COMPONENT STUDY AND 3D MODELLING

The part is called a top cover bearing. Nylon-6 makes up the component substance, which has 30% glass packed in it. The part has a black color and a SGR 30 rating. The part has a volume of 298.35 cm^3. The total mass of the components is 406.23 grams. The part has a density of 1.35 grams/cubic centimeter and an approximate volume reduction of 0.6%. The melting point is 285°C. Part that is produced is utilized in cars/automobiles.

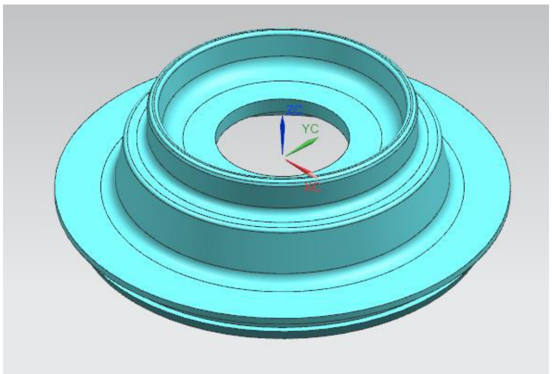

Fig -2: Top cover bearing 3D model

5. TOOL DESIGN CALCULATIONS

This section outlines a methodical approach to obtaining theoretical values for specific mold properties. Pointing out that the real values is much greater and takes into safe consideration, because the gained properties are not directly used in the mold.

5.1 Calculation of molding weight

Shot weight (Sw)

Weight of component (w) = Material density x Volume
w = 1.35 x 298.35
w = 402.77 g
Incorporate the mold mass into the formula and multiply by the multiplication factor (M.F), therefore,
Shot weight of component (Sw) = Weight of component x M.F
Sw = 402.77 x 1.05 = 422.90g
Sw = 423g

5.2 Clamping force/clamping tonnage

Clamping force = Projected area with feed system x Injection pressure
Projected area with feed system is 256 cm^2.
For processing Nylon 6 with 30% glass filled to produce an engineering part, the injection pressure required is 1650 kg/cm^2.
The cavity pressure is calculated to be 1/2 of the injection pressure because Nylon 6 with 30% glass fill has good flow ability and is also the least viscous material.
C_f = Projected area with feed system x 1/2 Injection pressure
= 256 x [1/2 x 1650] = 211200 kgf
C_f = 220 T

5.3 Capacity to plasticize (Pc)

Shot Capacity

Shot capacity = Swept volume x Material density x Constant
Where, Constant = correction factor for percent volume expansion of the plastic at the molding temperature for Nylon = 0.35 (semi-crystalline materials).
The screw-type is normally rated in terms of swept volume of the injection cylinder = 900 cm^3. Density of the material is 1.35 g/cm^3.
Shot capacity (g) = 900 x 1.35 x 0.35 = 425.25 g
Shot capacity of the machine = 425 g

5.4 Calculation of number of cavities

Calculating the quantity of cavities from machine capacity
Ns = Shot capacity of the machine / Shot weight of the component
Shot/Machine capacity of the machine for Nylon-6 30% glass filled is 425 grams. Shot weight of the component is 423 grams.
Ns = 425 / 423 = 1.005
Ns = 1 Cavity
Depending on shot capacity, a single cavity mold can be accepted.

5.5 Design of feed system

Diameter of sprue = 10 mm
Diameter of runner = 6 mm
Diameter of the Tunnel gate = 1.547 mm
No. of gates used = 4 numbers

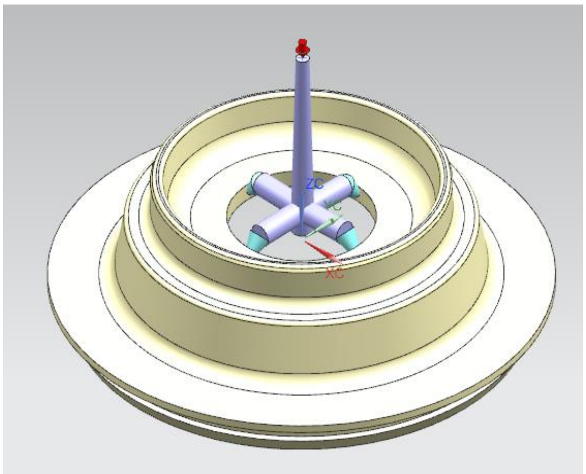

Fig -3: Part with feed system

5.6 Determination of working length of finger cam

L = [Ms / Sin θ] + [2c / Sin 2θ]
Where, Ms = splits movement to eject the component positively = 32.5 mm
θ = angle of finger cam = 25°
c = clearance between hole and the finger cam in the splits = 0.5 mm
L = [32.5 / Sin 25] + [(2 x 0.5) / Sin 2(25)]
L = 78.2 mm

5.7 Length of side core (Ls)

Length of side core = Fitting length + Working length + radius of finger cam
Where,
Fitting length = (Diameter of finger cam x 2) + Factor of Safety = (22 x 2) + 10 = 54 mm
Working length = [Slider stroke (Undercut + 5mm)] / Sin (Finger cam angle) = [42+10] / Sin 25 = 123.04 mm
Radius of the finger cam = 11 mm
Therefore,
Length of side core = 54 + 123.04 + 11 = 188.04 mm
Hence length of side core (Ls) is 188.04 mm

5.8 Injection time (Ti)

The injection time taken after the calculation is 6.25 s (value from the analysis report).

5.9 Cooling calculations

The solidifying time taken after calculation is Ts = 40 s (value from the mold flow report).

5.10 Cycle time estimation

Injection Time, Ti = 6.25 s
Solidifying Time, Ts = 40 s
Mold opening and closing time, Toc = 2Td = 2 x 5 = 10 s
Where, Td = machine cycle time = assumed as 5 s
Ejection Time (Te) = 2 s approximately
Mold lubrication time T_L = 3 + (ns + nc) + (nc - 1)
Number of side cores (ns) = 2
Number of cavities (nc) = 1
T_L = 3 + (2 + 1) + (1 - 1) = 6 s
Therefore, total cycle time = Ti + Ts + Toc + Te + T_L
= 6.25 + 40 + 10 + 2 + 6 = 64.25 s
Total cycle time = 65 s
From the above calculations the cycle time is 65 seconds.

5.11 Determination of number of shots/hr.

Number of shot/hr. = Total time / Total cycle time
= 3600 / 65
= 55 shots/hr.
Hence, number of shots/hr. is 55.

6. ANALYSIS OF MOULD

It focuses on the flow characteristics of plastics. After the hole is filled by molten plastic, the flow characteristics are measured. Plaster flow estimation is done in NX Simple Fill Basic and Molded Flow Consultant.

The application Mouldex3D controls NX Easy-Fill Plus. Early in the goods innovation period, designers may test the mold ability of plastic piece designs thanks to this included mold simulator tool. In order to optimize gate number/locations, choosing materials, component design, operational criteria, or product choice, programmers may additionally perform adjustments in advance. Autodesk's Molded Flow Adviser is a small, inexpensive device that simulates design throughout element manufacture. Thanks to the electronic samples that are produced, time to market and costs can be drastically decreased. The adviser makes it easier to add CAD data from several widely used programs, such as Solid Works®, and it speeds up item training.

6.1 Reports from the Flow Analysis

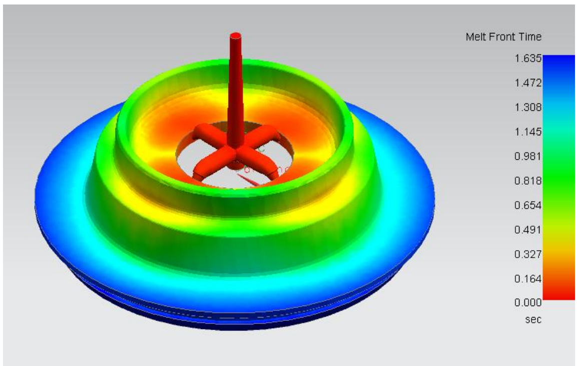

Fig -4: Melt Front Time

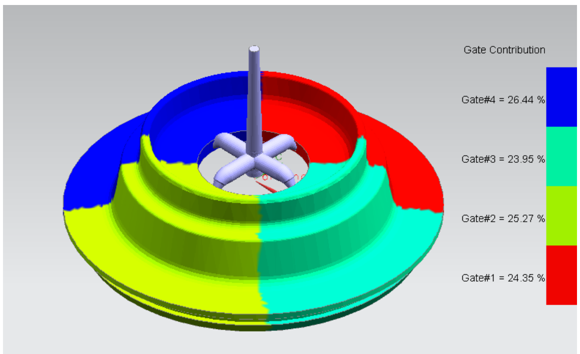

Fig -5: Gate Contribution

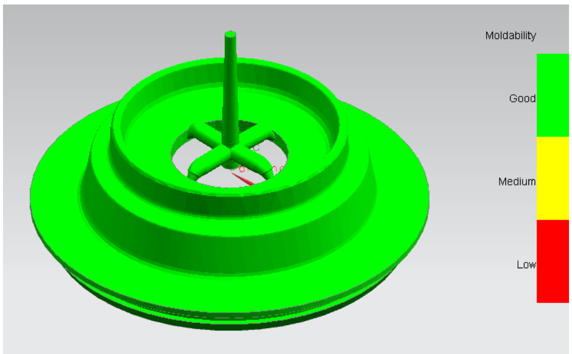

Fig -6: Moldability

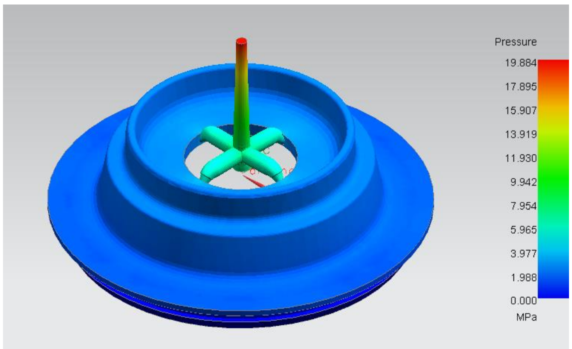

Fig -7: Pressure

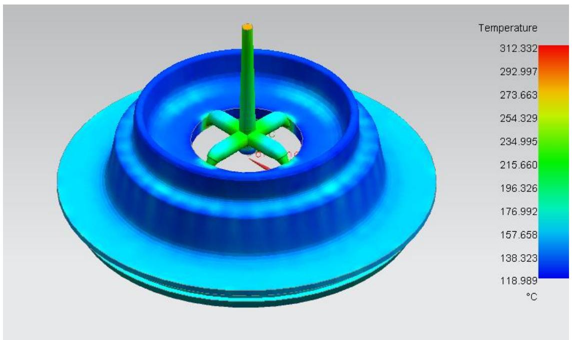

Fig -8: Temperature

6.1.1 Weld line

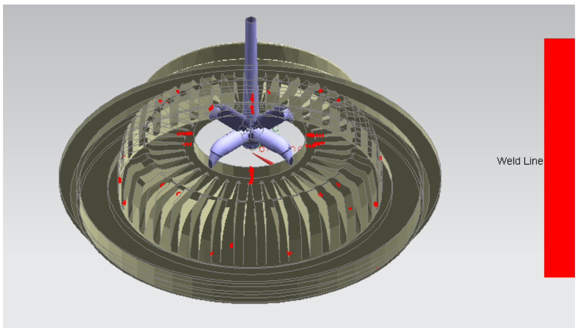

Fig -9: Weld line

The part with less weld marks and a workable design is depicted in the image. Raise the liquid warmth or the container's heat to prevent weld streaks. Loading time ought to be shortened. Increase the rate of injection.

6.1.2 Air Trap

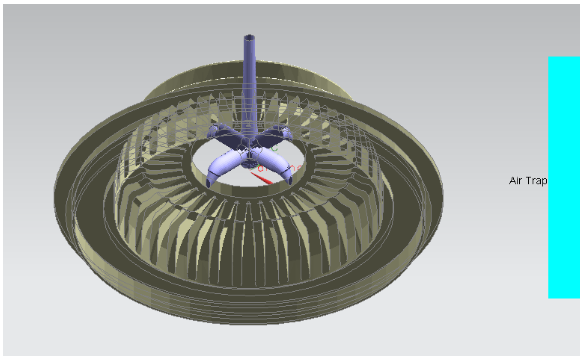

Fig -10: Air trap

The sites where wind trap may occur are indicated by the air capture result. The presence of air-traps is indicated by the component's blue color. Escapes should be arranged in a circular pattern to prevent air trapping. The baking time ought to be increased. Move the entry way to a new location. Reduce the speed at which you inject. Modify the valves location or size.

7. TOOL DESIGN

A filling mold is a device made up of multiple components that are used to mold & cool hot material into a certain part configuration. The tool's components as they appear in below figure.

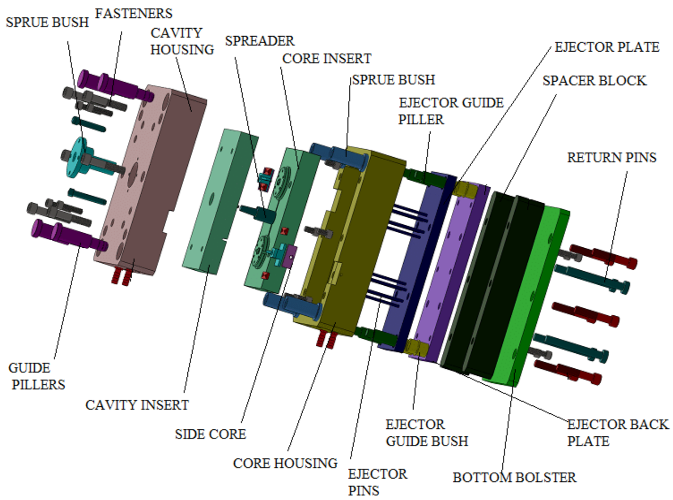

Fig -11: Exploded view of 2-plate mold

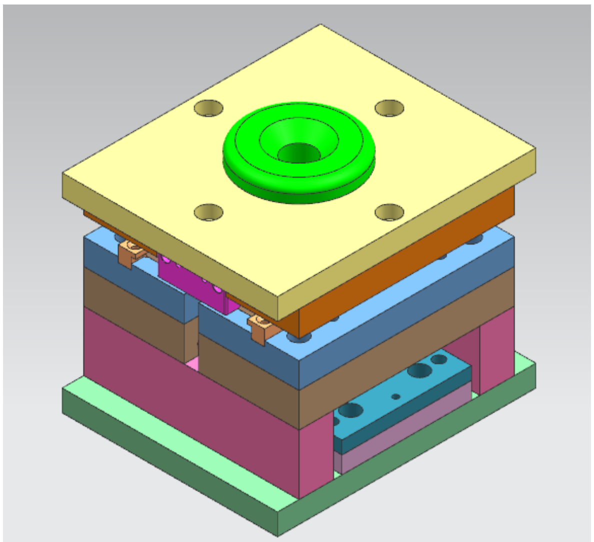

Fig -12: 3D isometric view of mold tool

8. MANUFACTURING & ASSEMBLY OF TOOL

8.1 Fabrication of tool

Part designs with an adequate dimensional range are created for every single element after the final layout is complete. Upon receipt of the drawings, the making department must determine whether to buy or manufacture them. Conventional parts such as bolts & nuts are obtained from marketplaces, while pieces that cannot be constructed internally are contracted. Other components that can be manufactured internally utilizing the existing facilities are retained. After deciding which pieces need to be made internally, the manufacturing branch starts development.

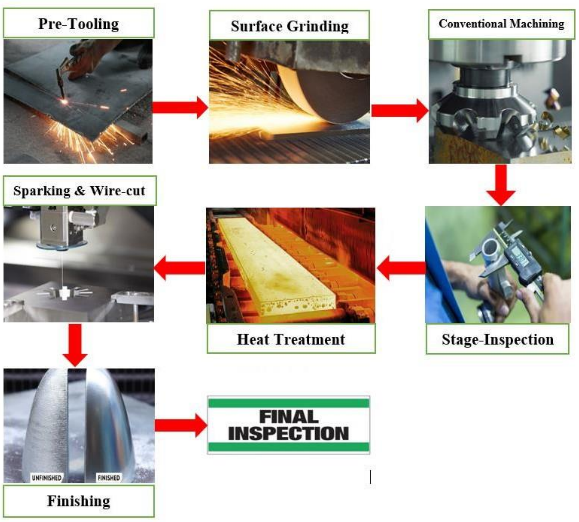

Fig -13: Manufacturing process planning

MOULD ELEMENTS	OPERATIONS	MACHINE USED
Top & Bottom Clamping Plate	Facing	Mild steel VMC
	Side Facing	Mild steel VMC
	Plate Grinding	Surface Grinding Machine
	Slot Milling	Mild steel VMC
	Drilling & Counter Drilling	Mild steel VMC
Cavity Housing	Facing	Mild steel VMC
	Slot & Pocket Milling	Mild steel VMC
	Drilling & Counter Drilling	Mild steel VMC
	Plate Grinding	AARTI H4 SGM
Core Plate	Facing	Mild steel VMC
	Slot & Pocket Milling	Mild steel VMC
	Drilling & Boring	Mild steel VMC
	Plate Grinding	AARTI H4 SGM
Core Back Plate	Facing	Mild steel VMC
	Slot & Pocket Milling	Mild steel VMC
	Drilling & Counter Drilling	Mild steel VMC
	Plate Grinding	AARTI H4 SGM
Spacer blocks	Facing	Mild steel VMC
	Drilling	Mild steel VMC
	Plate Grinding	AARTI H4 SGM
Cavity & Core Inserts	Facing	Mild steel VMC
	Drilling	Mild steel VMC
	Plate Grinding	AARTI H4 SGM
	Lapping	Manual
	Polishing	Manual
	EDM	Oscarmax Sparking EDM
	WEDM	DK7225 DM Wire EDM

Fig -14: Machining operations & machine use

8.2 Assembly of the tool

The construction work performed by the toolmaker is quite skilled. A good tool requires the ability of the toolmaker to maneuver each part into its designated spot in accordance with the layout. The two most crucial factors to take into account when assembling are attention and hygiene. The pieces will be destroyed if they are not handled carefully, which will result in wasted time and increase the price of the mold.

9. TOOL TRIALS

Following construction and assembly, the tool verified that the shape and sizes of the component were manufactured in accordance with the customer's specifications. Try-out is the process of subjecting a production tool to real-world operational requirements on an injection molding machine.

The mold has been cleared of oil and dirt. Once the mold's halves are closed, the product flows by force via the equipment's nozzle and into the supply line. The material is transported to the mold's cavities by the feed mechanism. The substance is kept under stress until the part is dried. The part can then be ejected after the tool has been exposed and the mechanism that releases it assembly is activated.

The item is immediately examined after the test runs and then forwarded to the molding quality control division for the initial trial observation report, where grade assurance comments are provided and the components is evaluated in all parameters. All the components are submitted for assembly and inspection if each size is acceptable.

For the first several shots, less force is used to inject the substance. Due to the recent die, it is essential to check for substance leaks and ensure that both the core & cavity are suitable for the hot operating circumstances. Both the engagement pressure & the applied flow are being steadily maintained. On the first attempt, defective components that were not filled were made.

9.1 Defects occurs in mold

It is frequently difficult to identify the root reason behind molding errors due to the multitude of elements that influence injection formed item manufacture. Multiple reasons can lead to a single error. The reasons may have started with the properties of the stuff, the circumstance of the process, the molding machine, or another mold. Before trying to fix any errors, it's crucial to thoroughly understand the different kinds of errors and their root causes and fixes for them.

The component has the following defects that have been observed:

- Weld lines
- Air-traps & Air bubbles
- Flow lines
- Cracks
- Flash
- Ejector pin marks

Table -1: Inspection report

Sl No.	Check Item	Check Method	Frequency	Result
1	Flow Lines	Visual	100%	Less & Acceptable
2	Cracks	Visual	100%	Free from cracks
3	Flash	Visual	100%	Acceptable
4	Ejector pin marks	Visual	100%	Acceptable
5	Weld lines	Visual	100%	Free from weld lines
6	Air traps & Air bubbles	Visual	100%	Acceptable

9.2 Finished Part

Fig -15: Part's core side

Fig -16: Part's cavity side

10. CONCLUSION

In this covert work, it is described the layout of a polymer injection molding used to create the top bearing cover, an item found in cars. An examination of the element material is conducted in order to ascertain its physical, chemical, and electrical characteristics as well as its mold ability. The machine that is most suited to operate the mold process is selected through estimates. The NX Easy Fill Plus & Flow Advisor program was used to do the study. Before trying to fix a tool that doesn't function well on the first try, it's crucial to thoroughly understand the many kinds of errors and the root cause fixes for them. Then, the tool is constructed after going through the stages of production.

11. SCOPE FOR FUTURE WORK

- If there are multiple kinds of bearing parts manufacturing. The core & cavity frames can be changed to reuse the same mold base in the event that another piece is delivered.
- The outcomes of the mold flow study can be utilized to enhance the mold design. Further research can be done to optimize the settings and boost the mold's productivity.
- Finite Element Analysis program can be utilized to predict tool longevity and perform stress test and endurance analyses on the mold's component elements.

REFERENCES

[1] Injection mould design, R.G.W Pye, 4th Edition, Affiliated East-West Press Pvt Ltd.

[2] Practical guide to injection moulding, Arburg Vannessa Good-ship, 2nd Edition, Smithers Group Company.

[3] Mould theory, Handbook, GTTC-Mysore.

[4] Understanding injection mould design, Herbert Rees, Hanser Publishers 2001.

[5] Polymer science & technology, Joel R Fried, University of Cincinnati.

[6] Selecting injection moulds, Herbert Rees Bruce Catoen, Hanser Gardner Publications, Cincinnati.

[7] ASM handbook, Failure, Causes & Analysis, Volume-15.

[8] Srinivasa G, L.G. Sannamani, Dr. S. N. Ravi Shankar, Influence of Gating System for Electrical Panel Part Component by Simulation Flow Analysis using NX-Easy Fill Advanced Software, ISSN 2582-7421, Vol 3, no 9, pp 2148-2154, September 2022.

[9] Abhilash V, L.G. Sannamani, Dr. S.N. Ravi Shankar, Design and Analysis of an Injection Mould Tool for Power Board Housing Used in Washing Machine, ISSN 2582-7421, Vol (2) Issue (8) (2021) Page 1423-1428.

[10] Vinayak S. Patil, L. G. Sannamani, Dr. S. N. Ravi Shankar, Design and Simulation Study of Mixer Jar Lid Component using NX-11.0 and Autodesk Moldflow Adviser Software, ISSN 2582-7421, Vol (2) Issue (8) (2021).

[11] Hongbo Fu, Hong Xu, Ying Liu, Zhaogang Yang, S. Kormakov, Daming Wu and Jingyao Sun, Overview of Injection Molding Technology for Processing Polymers and their Composites, ES Mater. Manuf., 20 May 2020.

[12] Prekshith Y. P, Dr. S. N. Ravi Shankar, L. G. Sannamani, Flow Analysis for an Injection Mould Tool for the Component ION Base Plate, ISSN 2582-7421, Vol (2) Issue (8) (2021).

[13] Technical Directory on Design and Tooling for Plastics, Central Institute of Plastics Engineering and Technology.

[14] E-Profit-Servo-201908.pdf - Electronica Plastic Machines manual.

[15] G.T.T.C mould design manual.

[16] DME standard manual.