ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Optimizing Injection Moulding Production Rate

Savara Chiranjeevi
P.G (M.Tech. Machine Design)
Student In Department of Mechanical Engineering,
Sarada Institute of Science Technology &
Management (Approved By A.I.C.E.T., Affiliated To
J.N.T.U, Kakinada, A.P) Srikakulam.

Mr. B.Ashok Kumar, M.Tech
Assistant Professor,
Department of Mechanical Engineering,
Sarada Institute of Science Technology &
Management(Approved By A.I.C.E.T., Affiliated To
J.N.T.U, Kakinada, A.P) Srikakulam.

ABSTRACT:

For plastic components manufacturing process used is Injection moulding. While doing this manufacturing process we have to face some problems in filling process, clamping, cooling, and amount of material to inject into the cavity area. Due to the above problems there is wastage of material, time, poor component quality.In this study, we are conducting mould flow analysis on the plastic component power box by using plastic adviser. In this study we are conducting mould flow analysis by varying processing conditions for different materials.

Processing conditions used in this project are Material melting temperature, mould temperature, Maximum machine Injection pressure, velocity/pressure switch-over by volume, injection time, Machine clamp open time. Materials used are Polypropylene (P.P), and Acrylonitrile butadiene styrene (ABS). In this study we are avoiding above problems faced in the injection moulding processes.

Results are getting in this project are Actual Injection Time, Actual Injection pressure, Weld lines, Air Traps, shot Volume, Estimated cycle time, Filling clamping force, Packing Clamp Force. By using above result we can optimize Injection moulding manufacturing process. This project is done in the Navya polymers, Hyderabad. Software's used in this project are Pro/Engineer and plastic advisor.

Key words:

Optimizing injection moulding process, Mould flow analysis, processing conditions, Pro/Engineer, plastic adviser.

1.Inroduction :

Injection Moulding:

Injection moulding is a manufacturing process for producing parts from both thermoplastic and thermosetting plastic materials. Material is fed into a heated barrel, mixed, and forced into a mold cavity where it cools and hardens to the configuration of the mold cavity.

After a product is designed, usually by an industrial designer or an engineer, molds are made by a moldmaker (or toolmaker) from metal, usually either steel or aluminium, and precision-machined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels.

2.Equipment:

Injection molding machines consist of a material hopper, an injection ram or screw-type plunger, and a heating unit. They are also known as presses, they hold the molds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can exert.

This force keeps the mold closed during the injection process. Tonnage can vary from less than 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations.

The total clamp force needed is Determined by the projected area of the part being molded. This projected area is multiplied by a clamp force of from 2 to 8 tons for each square inch of the projected areas.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

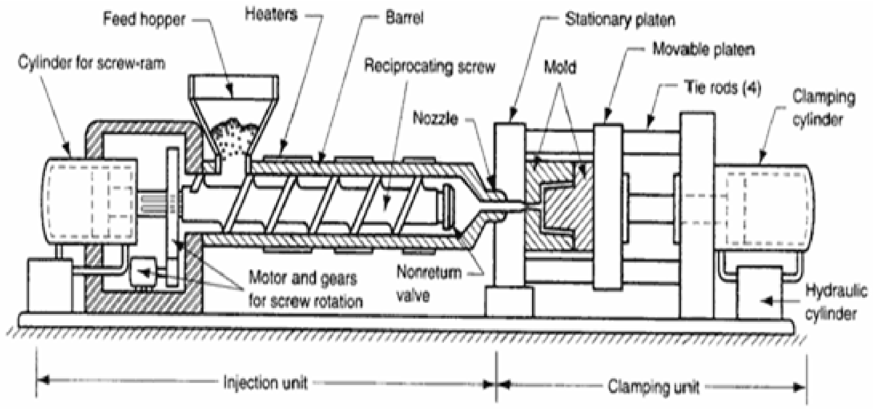


Fig 1.1 Injection molding machine

3.Injection process:

With Injection Molding, granular plastic is fed by gravity from a hopper into a heated barrel. As the granules are slowly moved forward by a screw-type plunger, the plastic is forced into a heated chamber, where it is melted. As the plunger advances, the melted plastic is forced through a nozzle that rests against the mold, allowing it to enter the mold cavity through a gate and runner system. The mold remains cold so the plastic solidifies almost as soon as the mold is filled.

4. Applications:

Injection molding is used to create many things such as milk cartons, containers, bottle caps, automotive dashboards, pocket combs, and most other plastic products available today. Injection molding is the most common method of part manufacturing. It is ideal for producing high volumes of the same object. Some advantages of injection molding are high production rates, high tolerances are repeatable, wide range of materials can be used, low labor cost, minimal scrap losses, and little need to finish parts after molding. Some disadvantages of this process are expensive equipment investment, running costs may be high, and parts must be designed with molding consideratio .

5. INJECTION MOLDING CYCLE AND PROCESS:

The injection molding process occurs cyclically. Typical cycle times range from 10 to 100 seconds and are controlled by the cooling time of the thermoplastic or the currying time of the thermosetting plastic. The plastic resin in the form of pellets or powder is fed from the hopper and melted.

In a reciprocating screw type injection molding machine, the screw rotates forward and fills the mold with melt, This is called the hold time. Eventually the gate freezes, isolating the mold from the injection unit, the melt cools and solidifies. Next the screw begins to rotate and more melt is generated for the next shot. In the soak time the screw is stationary and the polymer melts by heat conduction from the barrel to the polymer. The solidified part is then ejected and the mold closes for the next shot.

6.Trouble shooting for injection molding process:

1.Black Spots, Brown streaks.
2.Blisters (Air Entrapment).
3.Brittleness.
4.Bubbles.
5.Short Shot (Incomplete Filled Parts).
6.Sink Marks.
7.Splay Marks, Silver Streaks.
8.Sprue Sticking.
9.Surface Finish (Low Gloss).
10.Surface Finish (Scars, Wrinkles).
11.Undersized Part.
12.Valve Pin Does Not Close.
13.Voids.
14.Warping, Part Distortion.
15.Weld Lines.

7. Black spots, brown streaks Description

Black spots and brown streaks appear as dark spots or streaks in the molded part and are usually caused by thermal damage to the melt.

Possible Solutions:

- Check the material for contamination.
- Decrease the melt temperature.
- Decrease the overall cycle time.
- Purge and/or clean the screw and barrel.
- Decrease the screw speed. High screw speeds may cause the material to degrade.

8. Weld lines
Description

Weld Lines are created when two or more melt flow fronts meet possibly causing a cosmetically visible line. It can also create a weakened area in the finished molded part especially with filled resins.
Possible Solutions

- Increase injection pressure.
- Increase injection speed.
- Increase injection hold.
- Increase melt temperature.
- Increase mold temperature.
- Make sure part contains no sharp variation in cross-sections.

9. MOULD FLOW ANALYSIS:

Mould flow, 3D solids-based plastics flow simulation that allows plastics part designers to determine the manufacturability of their parts during the preliminary design stages and avoid potential downstream problems, which can lead to delays and cost overruns. Following are the benefits:

- Optimize the part wall thickness to achieve uniform filling patterns, minimum cycle time and lowest part cost Identify and eliminate cosmetic issues such as sink marks, weld lines and air traps.

- Determine the best injection locations for a given part design Mould flow analysis gives you the ability to maintain the integrity of your product designs. It provides you the tools to quickly optimize part designs and check the impact of critical design decisions on the manufacturability and quality of the product early in the design process.

There is no need to:

- Compromise the aesthetics of your design concept for manufacturability;

- Go through a lengthy trial and error process to find the most suitable material to produce the part with the highest possible quality and the lowest possible cost

- Find out during trial runs that the produced part has visual blemishes, such as sink marks, weld lines, air traps or burn marks.

10. GIVEN VALUES IN MOULD FLOW ANALYSIS

• Case 1

- Material used : Generic P.P

• Properties:

- Melt Density: 0.72828g/cm3

- Solid Density: 0.89163g/cm3

- E= 1340Mpa

- Poissons ratio = 0.392

- Recommended processing conditions:

- Mould Surface temperature = 40 C

- Melt Temperature = 240 C

• Present Processing conditions of the machine:

- Mould Surface temperature = 30 C

- Melt Temperature = 180.00 deg.C

- Maximum machine injection pressure = 120 Mpa

- Automatic Injection time = 10sec

- Machine clamp/open time = 10 Sec

- Mould Flow Report:

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

10. Result:

Release Level:	5.0

bottom-part	
Part Name:	bottom-part
Part Revision:	11
Material Supplier:	Generic Default
Material Grade:	Generic PP
Max Injection Pressure:	180.00 MPa
Mold Temperature:	40.00 deg.C
Melt Temperature:	240.00 deg.C
Model Suitability:	Part model was highly suitable for analysis.

Filling Analysis	bottom-part
Moldability:	Your part will be extremely difficult to fill and part quality may be unacceptable. View the Confidence of Fill plot and use the Dynamic Adviser to get help on how to improve the filling of the part.
Confidence:	Low
Injection Time:	8.16 sec
Injection Pressure:	43.51 MPa
Weld Lines:	Yes
Air Traps:	Yes
Shot Volume :	842.97 cu.cm
Filling Clamp Force:	155.51 tonne
Packing Clamp Force Estimate @20%:	(8.70)MPa 111.76 tonne
Packing Clamp Force Estimate @80%:	(34.81)MPa 447.05 tonne
Packing Clamp Force Estimate @120%:	(52.21)MPa 670.58 tonne
Clamp Force Area:	1259.54 sq.cm
Cycle Time:	24.97 sec

Cooling Quality	bottom-part
Cooling Quality:	Your part will have large problems cooling and may cause problems with ejection.
Surface Temperature Variance Range	-17.56 deg.C to 11.86 deg.C
Freeze Time Variance Range	-7.51 sec to 29.56 sec

Sink Mark Analysis	bottom-part
Sinkability:	Less than 1% of your model was found to be prone to sink marks

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

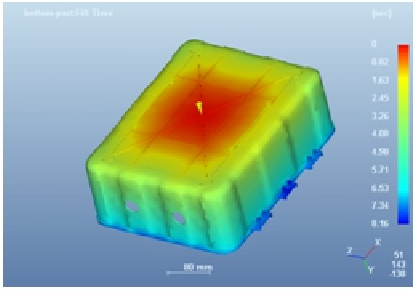


Fig.4.1.1.2Fill time

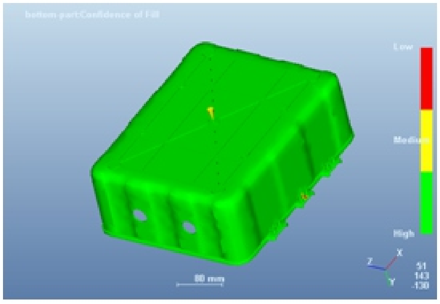


Fig 4.1.1.3.Confidence of fill

Injection pressure

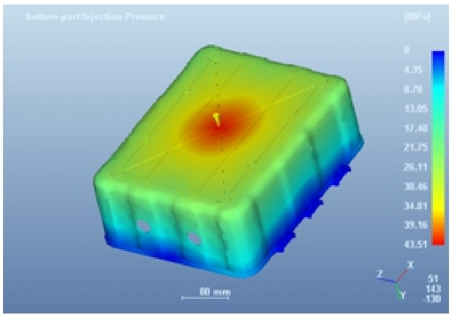
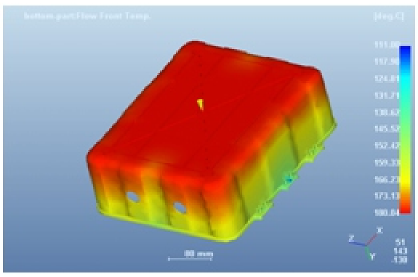


Fig4.1.1.4.Injection pressure



Weld lines

12.CONCLUSION:

In the production of components in injection molding process by changing processing parameters by trial and error method, the company is incurring loss in material, time and power. In this thesis, the above problem is rectified by taking software support of plastic advisor which is a module in Pro/Engineer. In this software, the component can be checked for filling by given processing parameters before going to manufacturing.

In this the bottom part of the power box is analyzed for mould flow using materials Generic P.P and ABS plastic. For both the materials, by changing processing parameters by seven times, the production of the component is good. In the trial and error method nearly 50 components are wasted but by using this mould flow analysis only 7 components are wasted. The percentage of reduction of wastage is 86%.

13. SCOPE OF THE FUTURE :

We can using this mould flow analysis the reduction of wastage is increased.And also by using this analysis, the exact processing parameters for production can be determined and increase the production rate.

14. REFERENCES:

1.Injection Mould Design byR.G.W. Pye
2.Injection Molding Handbook by Tim A. Osswald, Lih-Sheng Turng, Paul J. Gramann
3.Materials Science And Engineeringby William D. Callister
4.Injection Molding Machines by Friedrich Johannaber
5.Injection Molding Handbook by Dominick V. Rosato, Donald
6.Plastic Injection Molding by Douglas M. Bryce
7.Injection Mold Design Engineering by David O. Kazmer
8.Injection Molding by GerdPötsch, Walter Michae
9.Plastic Part Design for Injection Molding by Robert A. Malloy
10Injection Molding by Musa R. Kamal, Avram I