ISSN (e): 2250 – 3005 || Vol, 03 || Issue, 8 || August– 2014 ||
International Journal of Computational Engineering Research (IJCER)

Case Study on Injection Moulding Windsor 650 Machine Parameters on Wall Thickness Variation Defect

K. Raghavendra Kasyap[1] and M Subba Rao[2]
[1] M. Tech Student in MITS, Madanapalle
[2] Assistant Professor in Mechanical Engineering Department, MITS, Madanaplle

ABSTRACT:
In present scenario, injection moulding is most suitable manufacturing process for developing a plastic product. While developing a plastic product through injection moulding Windsor 650 machine, various defects are occurred internally and externally. Optimization of various parameters may reduce the defects of products and effect on the quality of the product. The purpose of this paper is studying the effect of various injection moulding Windsor 650 machine process parameters on Wall thickness variation defect. The Wall thickness variation is the most common defect which will form internally. To reduce the Wall thickness variation defect the various process parameters of injection moulding Windsor 650 machine are optimized by using Design of Experiments method.

KEYWORDS: *Design of Experiments, Injection moulding, Optimization, Process parameters, Wall thickness variation defect.*

I. INTRODUCTION

Injection moulding machine is most commonly used to producing plastic products. This process is most practical and cost effective to produce plastic products. During producing a product using injection moulding process various defects such as warpage, shrinkage sink marks and weld lines can be occurred. The optimization of injection moulding process parameters is very important to reduce these defects [1]. The process parameters for each defect are different, because of that while decreasing one defect may cause to increase another various defects [2]. So optimization of various process parameters is very essential for minimization of all the defects to get the quality product [3]. Common defects in injection moulding process are classified in to two ways. They are,

[1] Dimensional related
[2] Attribute related

Dimensional related defects can be considered by correcting the mould dimension. But attributed related defects are generally depends on process parameters of Injection moulding [4]. This paper presents the case study on injection moulding Windsor 650 parameters on wall thickness variation defect. Cavity pressure is the most common parameter in injection moulding process to compensate the shrinkage during cooling stage [5]. The mould temperature is one of the most efficient parameters in injection moulding to reduce warpage and shrinkage defects [6]. Holding time is another major effecting factor which influence the quality of the product produced. While temperature and pressure are constant some defects still arise due to effect of time [7].

II. TYPES OF WINDSOR INJECTION MOULDING MACHINE

There are three Windsor injection moulding machines which are most commonly used machines in the industries to produce plastic components. They are
[1] Windsor sprint 180 machine.
[2] Windsor sprint 350 machine.
[3] Windsor sprint 650 machine.

Windsor sprint 180 machine: This type of machines is used for manufacturing a wide range of plastic products. This machine used to develop a plastic product which is small in size.
Example: Ribs, Bushes, vane plugs, pen caps etc.

Windsor sprint 350 machine: This type of machines is used for manufacturing a wide range of plastic products. This machine is used to develop a plastic product which is medium in size.
Example: Battery covers, Boxes etc.

Windsor sprint 650 machine: This type of machines is used for manufacturing a wide range of plastic products. This machine is used to develop a plastic product which is large in size.
Example: Battery containers, Buckets, Large plastic products.

III. WINDSOR SPRINT 650 MACHINES

The Windsor sprint 650 machine is one of the most commonly used injection moulding machine to produce a plastic product which is greater in size.

Procedure of Windsor sprint 650 machine

Like remaining processes, this machine is also simple to produce a wide range of variety of the products. The plastic granules are in the form of pellets are pre-heat in the dryer for 3 hours up to certain temperature. After preheating the pellets then poured in to the injection moulding machine through hopper.

In this machine there are mainly three stages for developing a plastic product,
They are
[1] Injection stage.
[2] Clamping stage.
[3] Ejection stage.

Injection stage: In this stage the plastic granules comes through the hopper and feed in the form of pellets. A melting zone is there in this stage for melting the raw material at certain temperature and converts the pellets granules into liquid form. A reciprocating screw is located in the injection stage for inject the raw material in to the next stage. While screw moves backward the melted raw material moves forward. Now, the screw moves forward the raw material inject in to next stage through nozzle of the screw at certain temperature and certain speed. Injection moulding machine uses moulds to manufacture a plastic product. There are many components in the mould but it is split in to two halves. They are, mould core and mould cavity. When the mould is closed, the space between the mould core and mould cavity forms the part cavity. Multiple-mould is sometimes used, in which the two mould haves form several identical part cavities.

Clamping stage: When injecting the material in to the mould the two parts of mould must be securely closed by clamping stage. The mould core is attached to the injection chamber and mould cavity is attached to cooling chamber. While the material is inject in to the mould, the clamping stage pushes the two mould core and cavity together at certain force to close the mould.

Ejection stage: After some time, the cooled part is to be ejected from the mould by ejection system. While open the mould, a mechanism is used to push the plastic product out of the machine. Once the part is ejected, the mould will closed for the next shot of injection operation.

Defects in the product: Many factors can affect the quality of the plastic products during injection moulding process.

There are many defects that are formed internally and externally of the injection moulding plastic products. They are
[1] Short fill.
[2] Silver streak.
[3] Colour variation.
[4] Sink marks.
[5] Weld lines.
[6] Surface defects.
[7] Flashes.
[8] Warpage and
[9] Wall thick ness variation

1. Short fill

This means occurring of the short weight on the product. It is caused by insufficient filling of raw material in the mould.

Remedies of short fill are

[1] Increase the short weight of the product.
[2] Increase the injection pressure.
[3] Decrease the cylinder temperature.
[4] Increase the injection speed.
[5] Decrease the rotation speed and reduce the back pressure of the screw.

2. Silver streak

It contains the moisture particles with in the raw material. It is in the form of silver colour which is to be formed on the product surface and which follow the flow direction of the raw material in the cavity.

It is caused by moisture occurred in the raw material.

Remedies of silver streak are

[1] Pre – heat the raw material in the dryer for up to certain temperature for removing moisture particles.
[2] Decrease the cylinder temperature and increase the injection pressure.
[3] Increase the injection speed.
[4] Increase the rotation speed and adjust the back pressure of the screw.
[5] Ensure the proper melting duration of the raw material.

3. Sink marks

It is a local surface depression that typically occurs in the moulding with thicker sections or at locations above ribs, bosses and internal fillets.

Remedies of the sink marks are

[1] Increase the hold on time, feeding time and cooling time.
[2] Increase the injection pressure.
[3] Extend the hold pressure duration.
[4] Increase the screw forward time.
[5] Decrease the melt temperature and
[6] Decrease the mould temperature.

4. Weld lines

It is occurred when melt flow front collide in a mould cavity. It is very common and difficulty injection moulding defect to eliminate.

Remedies of the weld lines are

[1] Increase the injection pressure.
[2] Increase the injection speed.
[3] Increase the cylinder temperature.
[4] Make the position of where the weld line occurs more close to gate.
[5] Change the gate position.
[6] Change the part thickness.

5. Colour variation

It is occurred when insufficient mixing up of raw - material with master batch. Remedy is to take care of mixing proportions with accurate weight proportions.

6. Surface defects

The surface defects are mainly caused by either mould is heavy hot or in the form of heavy cold. The surface defects are mainly occurred at gates.

Remedies for surface defects are

When mould is hot

1. Cold the mould near gates.

When mould is cold

[1] Increase the mould temperature.
[2] Increase mould pressure and
[3] Increase the injection speed.

7. Flashes

It is caused when excessive injection pressure and insufficient clamping force.

Remedies of flashes are

[1] Decrease the injection pressure.
[2] Decrease injection temperature.
[3] Adjust the mould parting.
[4] Increase the clamp force.

8. Warpage

It is caused when sharp variation occurs in the wall thickness of the plastic product.

Remedies of the Warpage are

[1] Increase the melt temperature.
[2] Decrease the volume of the product.
[3] Decrease the injection pressure.
[4] Decrease the injection time.
[5] Adjust the gate sizes.
[6] Adjust the part designs.

Iv. Windsor 650 Machine Parameters on Wall Thickness Variation Defect:

The wall thickness variation is a common defect on injection moulding product. This is mainly occurred at internally of the plastic product**.** Hold on Pressure, Hold on speed, Hold on time and position of the raw material is the most efficient parameters of the wall thickness variation which will effect on the quality of the product.

Hold on Pressure: The hold on pressure holds the pressure against cooling the plastic in the cavity image while solidifies. It is one of the efficient parameter of injection moulding with range 50-80 bars. If hold on pressure increases, it result in over flash on the product. If hold on pressure decreases, it result in shrinkage.

Hold on Speed: It is the common parameter which causes the wall thickness variation defect in the injection moulding plastic product. Range of hold on speed is 20-40%. If hold on speed increases then shrinkage is avoidable on the product. If hold on speed decreases short fill occurs on the product.

Hold on Time: It is the common parameter which causes the wall thickness variation defect in the injection moulding plastic product. Range of hold on time is 5-8sec. If hold on time increases, it result in increasing the weight of the product. If hold on time decreases then short fill occurs on the product.

Position of raw material: It is the quantity of raw material which will feed for one stroke of injection rammer. It is the common efficient parameter in the injection moulding plastic product. If increasing and decreasing the position of raw material, it will affect in weight of the product and varying the wall thickness of the product.

v. CONCLUSIONS

This study is mainly focused on reduce the wall thickness variation defect in injection moulding product by optimizing various process parameters. Hold on pressure, hold on speed, hold on time and position of the raw material considered as most efficient parameters on the quality of the injection moulding product. By optimizing these parameters we can reduce the wall thickness variation defect in injection moulding Windsor 650 machine.

REFERENCES

[1] R. Hussin, R. M. Saad, Razaidi Hussin, and M. S. I. M. Dawi, "An optimization of plastic injection moulding parameters using taguchi optimization method", *Asian Transactions on Engineering,* November 2012, Volume 02 Issue 05.

[2] A. Akbarzadeh, and M. Sadeghi, "Parameter study in plastic injection moulding process using statistical methods and IWO algorithm", *International journal of Modeling and Optimization*, June 2011, vol.1, No.2.

[3] T. Eenzurumulu, and B. Ozcelik, "Minimization of warpage and sink mark index in injection - moulded thermoplastic parts using Taguchi Optimization method", *Materials and design* 27, 2006 853-861.

[4] D. Mathivanan, M. Nouby and R. Vidhya, "Minimization of sink mark defects in injection moulding process – Taguchi approach", *International Journal of Engineering, Science and technology,* 2010 vol. 2, NO. 2.

[5] M. Kurt, O.S. Kamber. Y. Kaynak, G. Atakok and, O. Girit, "Experimental investigation of plastic injection moulding: Assessment of the effects of cavity pressure and mould temperature on the quality of final products", *Materials and design* 30, 2009, (3217 - 3224).

[6] Ko-Ta chiang, "The optimal process conditions of an injection – moulded thermoplastic part with a thin shell feature using Fuzzy logic: A case study on machining the PC/ABS cell phone shell', *Materials and Design* 28 2007 1851 – 1860.

[7] M.C. Huang, and C.C. Tai, "The effective factors in the warpage problem of an injection – moulded part with a thin shell feature' *Journal of Materials Processing Technology* 110, 2001, 1 – 9.

[8] L.M Galantucci and, R. Spina, "Evaluation of filling conditions of injection moulding by integrating numerical simulations and experimental tests", *Journal of Materials Processing Technology* 141, 2003, 266 – 275.
[9] H. Oktem, T. Enzurumulu, and Ibrahim uzman, "Application of Taguchi optimization technique in determining plastic injection moulding parameters for a thin – shell part", *Materials and Design* 28, 2007, 1271- 1278.
[10] J. Cheng, Y. Feng. J. Tan, and W. Wei, "Optimization of injection mould based on Fuzzy mould ability evaluation", *Journal of Materials Processing Technology* 208, 2008, 222 – 228.
[11] B. Ozcelik, "Optimization of injection parameters for mechanical properties of specimens with weld line of polypropylene using Taguchi method", *International Communications in Heat and mass Transfer* 38, 2011, 1067 – 1072.
[12] Y.K Shen, C.W. Wu, Y.F Yu, and H. Wei- Chung, "Analysis for optimal gate design of thin-walled injection moulding", *International Communications in Heat and Mass Transfer* 35, 2008, 728 – 734.
[13] B. Farshi, S. Gheshmi, and, E. Miandoabchi, "Optimization of injection mouding process parameters using sequential simplex algorithm", *Materials and design* 32, 2011, 414 – 423.