MINIMIZATION OF SINK MARK DEFECTS IN INJECTION MOLDING PROCESS

Naveen Reddy Pothula¹, S.Vamshi Krishna², K.Charantej³, Bodugam Siddartha⁴

¹Assistant Professor in the Department of Mechanical Engineering VITS (N6), Karimnagar, Telangana, India-505001, reddy.n52@gmail.com
²Assistant Professor in the Department of Mechanical Engineering VITS (N6), Karimnagar, Telangana, India-505001, sravas.34@gmail.com
³Assistant Professor in the Department of Mechanical Engineering VITS (N6), Karimnagar, Telangana, India-505001, charantej.kalva@gmail.com
⁴Student of B.tech IV year in the Department of Mechanical Engineering VITS (N6), Karimnagar, Telangana, India-505001, bodugamsiddartha@gmail.com

Abstract - Because of overwhelming interest in plastic items, plastic businesses are developing in a speediest rate. Plastic infusion forming starts with shape making and in assembling of complex shapes. Ideal setting up of infusion forming process factors assumes an imperative part in controlling the nature of the infusion shaped items. It is all the most critical to control property abandons like sink marks. Sink marks are fundamentally a "planned in" issue and henceforth it is to be gone to amid outlines stages. Attributable to specific conditions and imperatives, now and then, it is somewhat overlooked amid configuration stages and it is relied upon to be dealt with by disintegrates with just guideline to 'do the best'. Treatment of various handling factors to control deserts is a mammoth errand that costs time, exertion and cash. Form Flow examination is an intense recreation device to limit the sink checks and to foresee the generation time required at the most minimal conceivable cost. Confirmation utilizing reenactment requires substantially less time to accomplish a quality outcome, and with no material expenses, as contrasted and the traditional experimentation strategies on the creation floor.

In this theory, a near examination has been performed by taking diverse process parameters and single door, two entryway and three entryway areas to limit the sink checks in the assembling of a set out light toward Alto Car plastic segment. Light is to be transmitted obviously, so blow openings, and sink checks and weld lines must be maintained a strategic distance from when material is filling in the form. Displaying is done in Pro/Engineer and Mold Flow Analysis is done in Plastic Advisor in Pro/Engineer.

Index Terms - Plastic Infusion, Sink Marks, Temperature control, Injection Pressure, Cost time, Cost Reduction, Less Material Wastage, ABS plastic

I. INTRODUCTION

Infusion shaping is the most generally utilized assembling process for the manufacture of plastic parts. A wide assortment of items is fabricated utilizing infusion shaping, which differ significantly in their size, multifaceted nature, and application. The infusion forming process requires the utilization of an infusion shaping machine, crude plastic material, and a shape. The plastic is liquefied in the infusion forming machine and after that infused into the shape, where it cools and hardens into the last part.
II. PROCESSES CYCLE

The procedure cycle for infusion forming is short, normally between 2 seconds and 2 minutes, and comprises of the accompanying four phases:

1. Clamping - Prior to the infusion of the material into the form, the two parts of the shape should first be safely shut by the clamping unit. Every 50% of the shape is connected to the infusion forming machine and one half is permitted to slide. The using pressurized water controlled clamping unit pushes the shape parts together and applies adequate power to keep the form safely shut while the material is infused. The time required to close and cinch the form is reliant upon the machine - bigger machines (those with more noteworthy clamping powers) will require additional time. This time can be assessed from the dry process duration of the machine.

2. Injection - The crude plastic material, more often than not as pellets, is bolstered into the infusion forming machine, and progressed towards the shape by the infusion unit. Amid this procedure, the material is liquefied by warmth and weight. The liquid plastic is then infused into the form rapidly and the development of weight packs and holds the material. The measure of material that is infused is alluded to as the shot. The infusion time is hard to figure precisely because of the unpredictable and changing stream of the liquid plastic into the shape. Be that as it may, the infusion time can be evaluated by the shot volume, infusion weight, and infusion control.

3. Cooling - The liquid plastic that is inside the form starts to cool when it reaches the inside shape surfaces. As the plastic cools, it will set into the state of the coveted part. Be that as it may, amid cooling some shrinkage of the part may happen. The pressing of material in the infusion organize enables extra material to stream into the form and diminish the measure of noticeable shrinkage. The shape cannot be opened until the point that the required cooling time has passed. The cooling time can be accessed from a few thermodynamic properties of the plastic and the most extreme divider thickness of the part.

4. Ejection - After adequate time has passed, the cooled part might be launched out from the shape by the discharge framework, which is joined to the back portion of the form. At the point when the shape is opened, a component is utilized to drive the part out of the form. Power must be connected to launch the part in light of the fact that amid cooling the part psychologists and clings to the shape. To encourage the launch of the section, a shape discharge operator can be showered onto the surfaces of the form cavity before infusion of the material. The time that is required to open the shape and discharge the part can be evaluated from the dry process duration of the machine and ought to incorporate time for the part to fall free of the form. Once the part is launched out, the shape can be cinched closed for the following shot to be infused.

III. MACHINE SPECIFICATIONS

Infusion shaped parts can shift enormously in estimate and in this way require these measures to cover a vast range. Therefore, infusion shaping machines are intended to each oblige a little scope of this bigger range of qualities. Test particulars are appeared beneath for three changed models (Babyplast, Powerline, and Maxima) of infusion shaping machine that are fabricated by Cincinnati Milacron.

| Model  | Clamp force (ton) | Shot capacity (oz.) | Clamp stroke (in.) |
|--------|------------------|---------------------|-------------------|
| Babyplast | 6.6              | 0.13 - 0.50        | 4.33              |
| Powerline | 330             | 8 - 34              | 23.6              |
| Maxima   | 4400            | 413 - 1054          | 133.8             |
IV. TOOLING

The infusion shaping procedure utilizes molds, ordinarily made of steel or aluminum, as the custom tooling.

MOLD BASE

The shape center and form depression are each mounted to the shape base, which is then settled to the platens inside the infusion shaping machine. The front portion of the shape base incorporates a help plate, to which the form depression is connected, the sprue bushing, into which the material will spill out of the spout, and a finding ring, so as to adjust the shape base with the spout.

MOLD CHANNELS:

All together for the liquid plastic to stream into the form depressions, a few channels are incorporated into the shape outline. To start with, the liquid plastic enters the shape through the sprue. Extra channels, called sprinters, convey the liquid plastic from the sprue to the greater part of the holes that must be filled.

MOLD DESIGN

MODELS OF HEAD LAMP WITH DIFFERENT GATE LOCATIONS

In addition to runners and gates, there are many other design issues that must be considered in the design of the molds. Firstly, the mold must allow the molten plastic to flow easily into all of the cavities.

V. MATERIALS

There are many sorts of materials that might be utilized as a part of the infusion shaping procedure. Most polymers might be utilized, including all thermoplastics, some thermosets, and a few elastomers.
POSSIBLE DEFECTS:

| Defect      | Cause                     |
|-------------|---------------------------|
| Flash       | Injection pressure too high, Clamp force too low |
| Warping     | Non-uniform cooling rate  |
| Bubbles     | Injection temperature too high, Too much moisture in material, Non-uniform cooling rate |
| Unfilled sections | Insufficient shot volume, Flow rate of material too low |
| Sink marks  | Injection pressure too low, Non-uniform cooling rate |
| Ejector marks | Cooling time too short, Ejection force too high |

LUBRICATION AND COOLING

Clearly, the shape must be cooled all together for the creation to happen. On account of the warmth limit, minimal effort, and accessibility of water, water is utilized as the essential cooling operator.

POWER REQUIREMENTS

| Material     | Specific gravity (g/cm³) | Melting point (°F) | Melting point (°C) |
|--------------|--------------------------|--------------------|--------------------|
| Epoxy        | 1.12 to 1.24             | 248                | 120                |
| Phenolic     | 1.34 to 1.95             | 248                | 120                |
| Nylon        | 1.01 to 1.15             | 381 to 509         | 194 to 265         |
| Polyethylene | 0.91 to 0.965            | 230 to 243         | 110 to 117         |
| Polystyrene  | 1.04 to 1.07             | 338                | 170                |

INFUSION MOLDING PROCESS CONDITIONS:

1. Temperature Control
2. Pressure Control
3. Molding Cycle
4. Holding Time
5. Cooling Time

VI. COST DRIVERS

Material Cost:
### VII. RESULTS TABLE

#### SINGLE GATE

|                | CASE 1   | CASE 2   | CASE 3   |
|----------------|----------|----------|----------|
| CONFIDENCE     | MEDIUM   | MEDIUM   | MEDIUM   |
| FILL TIME (Secs) | 19.58    | 4.13     | 4.11     |
| INJECTION PRESSURE (MPa) | 5.70     | 4.32     | 10.36    |
| WELD LINES     | YES      | YES      | YES      |
| AIR TRAPS      | YES      | YES      | YES      |
| CYCLE TIME (Secs) | 624.16   | 609.05   | 590.00   |
| QUALITY PREDICTION | MEDIUM   | MEDIUM   | LOW      |
| SINK MARKS     | 16 % of your model was found to be prone to sink marks. | 16 % of your model was found to be prone to sink marks. | 16 % of your model was found to be prone to sink marks. |

#### TWO GATES

|                | CASE 1   | CASE 2   | CASE 3   |
|----------------|----------|----------|----------|
| CONFIDENCE     | MEDIUM   | MEDIUM   | MEDIUM   |
| FILL TIME (Secs) | 4.12     | 4.13     | 4.13     |
| INJECTION PRESSURE (MPa) | 13.31    | 10.72    | 8.31     |
| WELD LINES     | YES      | YES      | YES      |
| AIR TRAPS      | YES      | YES      | YES      |
| CYCLE TIME (Secs) | 609.02   | 609.04   | 609.06   |
| QUALITY PREDICTION | MEDIUM   | MEDIUM   | LOW      |
| SINK MARKS     | 18 % of your model was found to be prone to sink marks. | 18 % of your model was found to be prone to sink marks. | 18 % of your model was found to be prone to sink marks. |

#### THREE GATES

|                | CASE 1   | CASE 2   | CASE 3   |
|----------------|----------|----------|----------|
| CONFIDENCE     | MEDIUM   | MEDIUM   | MEDIUM   |
| FILL TIME (Secs) | 4.11     | 4.12     | 4.09     |
| INJECTION PRESSURE (MPa) | 10.77    | 8.50     | 6.35     |
| WELD LINES     | YES      | YES      | YES      |
| AIR TRAPS      | YES      | YES      | YES      |
| CYCLE TIME (Secs) | 609.01   | 609.03   | 609.01   |
| QUALITY PREDICTION | HIGH     | MEDIUM   | LOW      |
| SINK MARKS     | 17 % of your model was found to be prone to sink marks. | 17 % of your model was found to be prone to sink marks. | 17 % of your model was found to be prone to sink marks. |

### VIII. CONCLUSION

In this theory, the ideal procedure parameters and the ideal number of entryways required to fill the part head light with minimum imperfections is investigated. The quantity of entryways taken is one, two and three. The procedure parameters considered in three cases, Case-1: Max Injection Pressure: 180MPa, Mold Temperature: 40 deg C, Melt Temperature: 200 deg C, Case-2: Max Injection Pressure: 200MPa, Mold Temperature: 60 deg C, Melt Temperature: 230 deg C and Case-3: Max Injection Pressure: 230MPa, Mold Temperature: 80 deg C, Melt Temperature: 300 deg C.

THE MATERIAL IS ABS PLASTIC.

By watching the investigation comes about, utilization of single door is better since 16 % of model was observed to be inclined to sink marks when single entryway is utilized yet when two entryways or three doors are utilized 18% and 17% of model was observed to be inclined to sink stamps separately. At the point when the procedure parameters are viewed as, taking Case-1 (i.e) Max Injection Pressure: 180MPa, Mold Temperature: 40 deg C, Melt Temperature: 200 deg C is better since in three door area, the quality expectation is high for case 1 than different cases. So it can be inferred that to limit sink checks just single entryway and utilizing process parameters Max Injection Pressure: 180MPa, Mold Temperature: 40 deg C, Melt Temperature: 200 deg C is better. Be that as it may, considering the quality expectation, fill time and process duration utilizing three entryway areas is better. In any case, the drawback of utilizing three door areas is that there will be more wastage of material.
IX. REFERENCES

1. Plastic Injection Molding: Manufacturing Process Fundamentals by Douglas M. Bryce

Injection Mold Design: A Design Manual for the Thermoplastics Industry by R G W Pye

2. The Mold Design Guide by Peter Jones Injection trim - Society of Plastics Engineers Injection trim of plastic segments by John BownRosato, D. Infusion Molding Handbook. Second Edition, Chapman and Hall, London, 1995.

3. Lam, Y.C., Jin, S. Enhancement of the door area for plastic infusion forming. Diary of Injection Molding Technology, 2001

4. Sahputra, I.H. Examination of two stream investigation programming for infusion forming apparatus plan. Procedures of the International Conference on Industrial Engineering and Engineering Management, 2-4 Dec. 2007, Singapore.

5. Saman, A.M., Abdullah, A.H., Nor, M.A.M. PC reproduction opportunity in plastic infusion shape advancement for car part. Worldwide Conference on ComputerTechnology and Development, 13-15 Nov. 2009, Kota Kinabalu, Malaysia