MINIMIZATION OF SINK MARK DEFECTS IN INJECTION MOLDING PROCESS

Naveen Reddy Pothula[1], S.Vamshi Krishna[2], K.Charantej[3], Bodugam Siddartha[4]

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

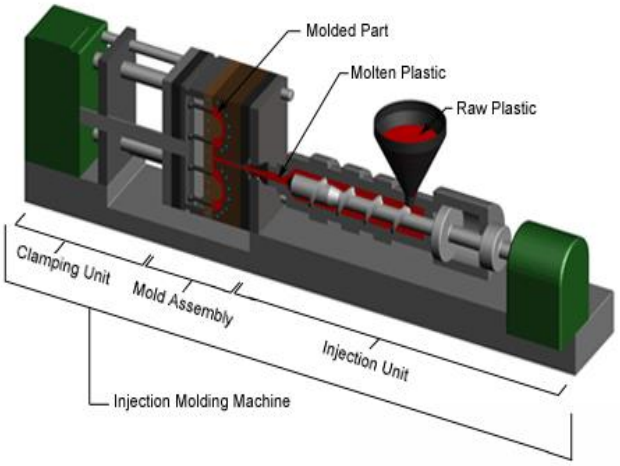
Abstract - Because of overwhelming interest in plastic items, plastic businesses are developing in a speediest rate. Plastic infusion forming starts with shape making and in assembling of complex shapes. Ideal setting up of infusion forming process factors assumes an imperative part in controlling the 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. PROCESS CYCLE

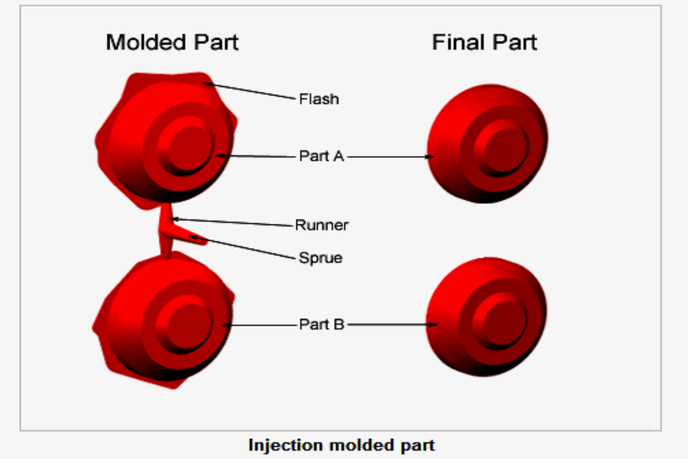
The procedure cycle for infusion forming is short, normally between 2 seconds and 2 minutes, and comprises of the accompanying four phases:

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 clasping 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 clipping 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 clasping 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.



Injection molded part

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.

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

Min. mold thickness (in.)	1.18	7.9	31.5
Platen size (in.)	2.95 x 2.95	40.55 x 40.55	122.0 x 106.3

IV. TOOLING

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

MOLD BASE

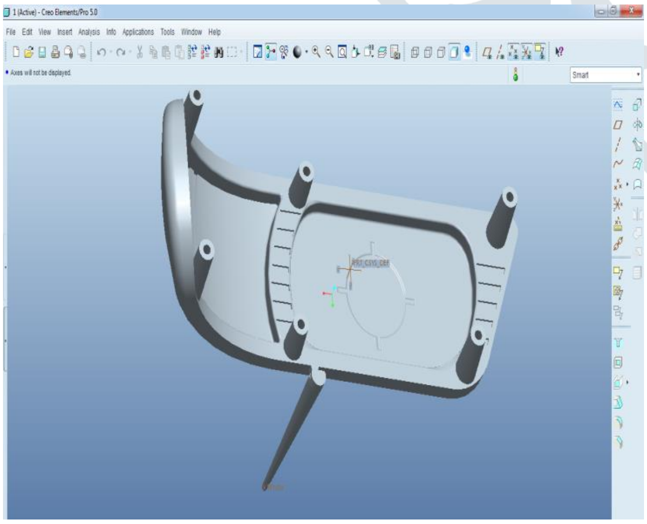
The shape center and form depression are each mounted to the shape base, which is then settled to the platens inside the infusion shaping machine. The front portion of the shape base incorporates a help plate, to which the form depression is connected, the sprue bushing, into which the material will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

MOLD CHANNELS:

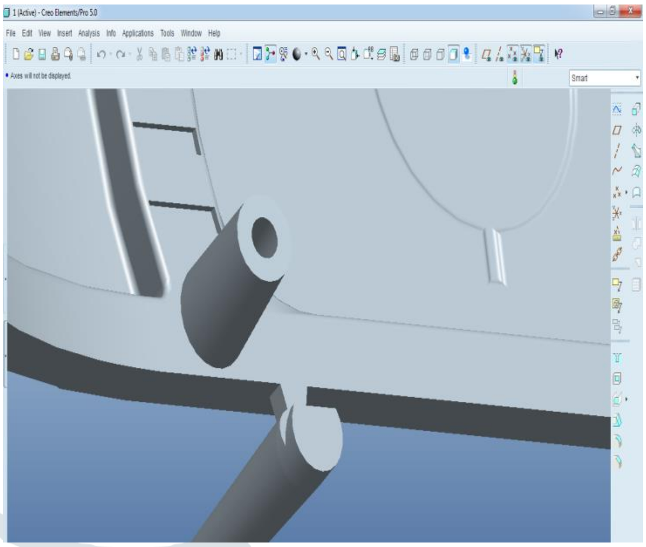
All together for the liquid plastic to stream into the form depressions, a few channels are incorporated into the shape outline. To start with, the liquid plastic enters the shape through the sprue. Extra channels, called sprinters, convey the liquid plastic from the sprue to the greater part of the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

MOLD DESIGN

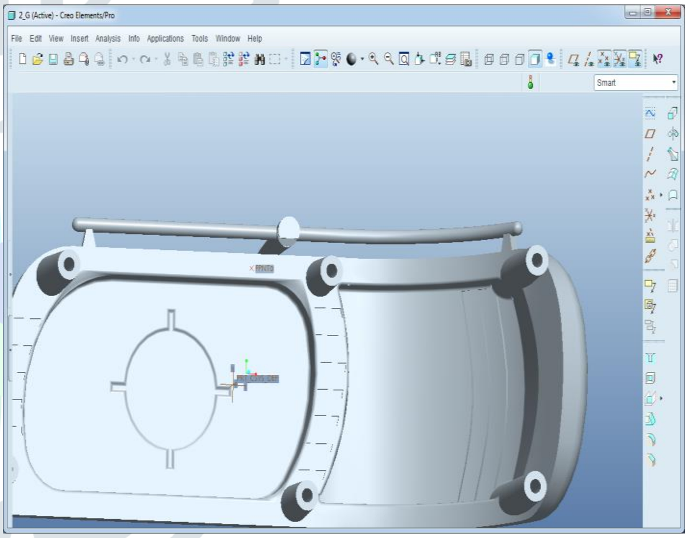
MODELS OF HEAD LAMP WITH DIFFERENT GATE LOCATIONS



SINGLE GATE



TWO GATES



THREE GATES

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.

Material name	Abbreviation	Trade names	Description	Applications
Acetal	POM	Celcon, Delrin, Hostaform, Lucel	Strong, rigid, excellent fatigue resistance, excellent creep resistance, chemical resistance, moisture resistance, naturally opaque white, low/medium cost	Bearings, cams, gears, handles, plumbing components, rollers, rotors, slide guides, valves
Acrylic	PMMA	Diakon, Oroglas, Lucite, Plexiglas	Rigid, brittle, scratch resistant, transparent, optical clarity, low/medium cost	Display stands, knobs, lenses, light housings, panels, reflectors, signs, shelves, trays
Acrylonitrile Butadiene Styrene	ABS	Cycolac, Magnum, Novodur, Terluran	Strong, flexible, low mold shrinkage (tight tolerances), chemical resistance, electroplating capability, naturally opaque, low/medium cost	Automotive (consoles, panels, trim, vents), boxes, gauges, housings, inhalors, toys
Cellulose Acetate	CA	Dexel, Cellidor, Setilithe	Tough, transparent, high cost	Handles, eyeglass frames
Polyamide 6 (Nylon)	PA6	Akulon, Ultramid, Grilon	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque/white, medium/high cost	Bearings, bushings, gears, rollers, wheels
Polyamide 6/6 (Nylon)	PA6/6	Kopa, Zytel, Radilon	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque/white, medium/high cost	Handles, levers, small housings, zip-ties
Polyamide 11+12 (Nylon)	PA11+12	Rilsan, Grilamid	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque to clear, very high cost	Air filters, eyeglass frames, safety masks
Polycarbonate	PC	Calibre, Lexan, Makrolon	Very tough, temperature resistance, dimensional stability, transparent, high cost	Automotive (panels, lenses, consoles), bottles, containers, housings, light covers, reflectors, safety helmets and shields

POSSIBLE DEFECTS :

Defect	Causes
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

VI. COST DRIVERS

Material Cost:

The material cost is dictated by the heaviness of material that is required and the unit cost of that material. The heaviness of material is plainly a consequence of the part volume and material thickness.

Production Cost:

The generation cost is fundamentally ascertained from the hourly rate and the process duration. The hourly rate is corresponding to the extent of the infusion shaping machine being utilized, so it is essential to see how the part configuration influences machine choice.

Tooling Cost:

The tooling cost has two fundamental segments - the form base and the machining of the cavities. The cost of the form base is essentially controlled by the span of the part's envelope. A bigger part requires a bigger, more costly, form base.

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/cm^3)	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

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