

Troubleshooting_Injection



Understanding Trouble Status

In order to solve problem more quickly& easily,
it is critical to understand the status of the site in the early stages.

1. Since when? (Production time / Time of occurrence)

- 1) Has this problem occurred since the beginning of the development? : Mold/Design/Resin – Should be experienced
- 2) No problem at the beginning of the mass production : What has been changed? – Lot deviation, Change in conditions, Decrepit equipments (mold etc.)
- 3) No problem over a year, this is a recent problem – Lot deviation, Seasonal factor

2. Where?

- 1) Whether it occurs continuously in a certain area
- 2) Whether it occurs in random areas
- 3) Whether it occurs near the gate / mid part / side ends / parts which have complicate shapes

3. How often?

- 1) Whether it occurs periodically
- 2) Whether it occurs irregularly without any rate
- 3) Whether it occurs at the beginning / after the machine has been stopped (Lunch break, etc.)
- 4) Whether it occurs continuously once it arises

4. Others

- 1) Scrap / Facility changes / Other circumstantial changes
- 2) Condition reproducibility (Cushion, Fill time) / History of similar defects

Types of Defects with Injection Molding

- Types of Defects with Injection Molding

- ① Appearance : Most defects occur on the appearance

- Unmolded(Short shot), Mold flash(Burr), Sink mark, Flow mark, Silver streak, Cloudy surface, Weld-line, Bubble, Black streak/Brown Striation, Crazing/Crack, Jetting, Warpage, and so on.

- ② Size defects

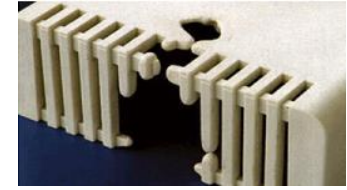
- Problems occurring during assembly (length, angle, degree of warpage, etc.)

- ③ Strength(Physical Properties)

- Loss of Tensile/Impact/Flexural Strength led by Carbonization, Weld-line, Crazing/Crack, etc.

- Identify the cause of defects by comprehensive observation of injection conditions / resin / mold / injection machine

Short shot



Source: <http://www.miconmould.com/plastic-mould-defects-burnt-streaks-9.html>

- Short shot is the problem which a product is injected smaller than the cavity shape.
- Improvement method varies depending on the cause.

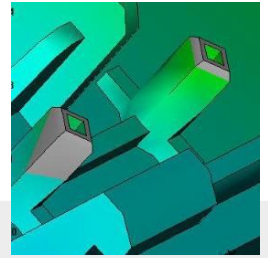
(1) Short shot with insufficient injection volume

- Might occur when the material solidifies too fast
- Reduce the flow length or injection pressure by increasing number of gates or changing the gate positions
- Increase injection pressure, injection speed, injection temperature, mold temperature
 - Yet the problem is not resolved, check if there is a difference between the actual and the set temperature
- Increase gate or runner size
- Check if the backflow prevention ring (check ring) is leaking
 - Block the gate and inject it to check the remaining amount of cushion

(2) Short shot with gas trap

- Install sufficient gas vents and maintain these clean to increase efficiency
- Decrease injection speed of the area in order to make the gas escape

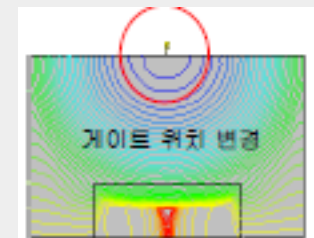
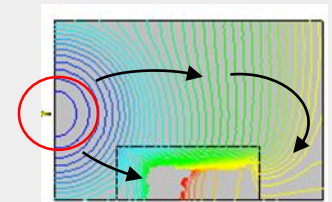
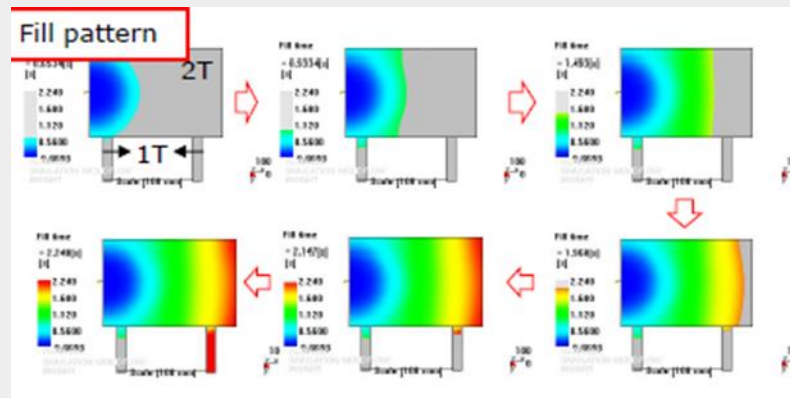
Hesitation (Flow congestion)



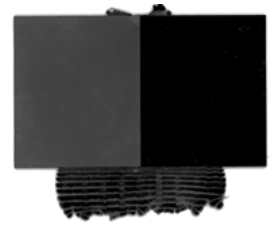
➤ If there are thin structures near the gate, or at the position that is perpendicular to the flow direction, congestion of flow might occur. Short shot occurs when resin is solidified before pressure transfer.

(1) Short shot at the thin structures near the gate

- Due to high resistance at thin areas, they are filled after the main part is filled
- Reduce thickness deviation between the defect area and the main part
- Make the thin area as the end of filling in order to transfer the pressure easily
- If it is the problem of runner system, resize the runner system.
- Increase material's flowability (Increase the temp.)



Mold flash (Burr)



- A phenomenon in which an unnecessary film is attached due to the leakage of resin along the parting line or the gas vent.
- Improvement method varies depending on the cause.

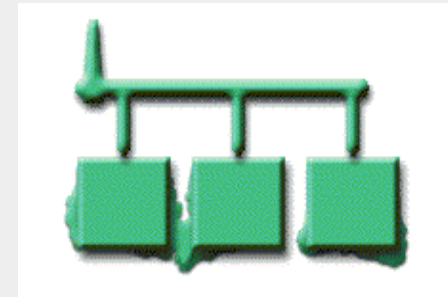


(1) Burr at the parting line

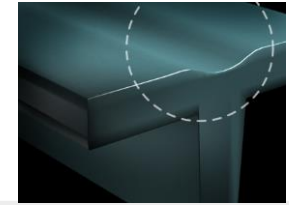
- If the inner pressure of mold becomes excessive, and leads clamping force decrease, the mold is opened and causes burr.
- Increase mold temperature or resin temperature to decrease injection pressure.
- Need to apply suitable multi-stage dwelling pressure and injection speed for its gate shape.
- Mold modification is required if it is due to decrepit mold.

(2) Burr at the gas vent

- Usually occurs at the end of a decrepit mold, when a low-viscosity material is used.
- Increase the viscosity by decreasing mold temp. or resin temp.
- Check mold deformation and strength, especially at defects
- Check whether the appropriate vent depth that fits to the material is applied.



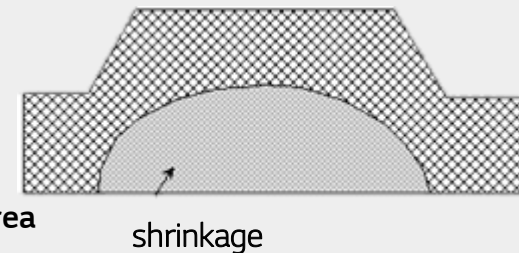
Sink mark



➤ Defects shrinking on the thickness direction, especially at ribs or thick areas on the surface of the product.

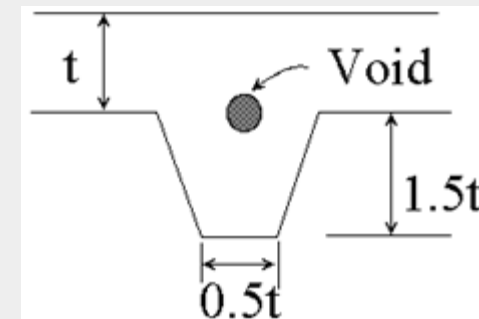
(1) Issue of dwelling pressure

- Increase dwelling time and pressure
- Set adequate amount cushion to deliver sufficient dwelling pressure
- Decrease the mold temp. to increase the thickness of the solidification area



(2) If there are many thick parts in the mold design

- Set thickness under 4mm
- Widen gate/runner to delay the gate seal
- Set gate at the thick point
- Reduce the thickness of unnecessary parts such as the rip, and add R
- Uniform thickness of the product is recommended



(3) Mold cooling

- Make sure the cooling system is properly designed.
- Cooling line should be installed where the heat dissipation is difficult (or thick)

Flow mark



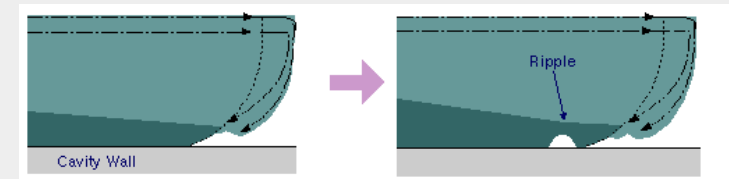
Source: <http://fimmtech.com>

- Poor flow marks on the surface of the gate or at the end area.
- Improvement method varies depending on the cause.

(1) Difference between the solidification speed and the speed of the resin tip

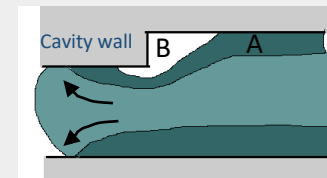
(Usually when material viscosity is high)

- Increase mold & injection temp. to increase the flowability.
- Increase injection pressure/injection speed to prevent unstable flow



(2) Rapid change of thickness of the product, or speed

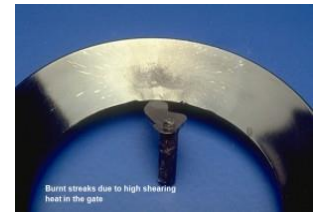
- The filling of skin layer B is slower than A, and the area B is filled later.
This results defective flow mark.
- Change the gate position to prevent abrupt changes in thickness of the flow, or add R
- Apply multi-stage injection speed to reduce abrupt changes in speed



(3) Uneven temperature

- Check mold cooling line and fix if needed

Silver streak



- Silver stripe shaped defects which appear to be popped on the product
- Improvement method varies depending on the cause.

(1) Undried Material

- Dry for an appropriate time and temp. which material supplier has provided on the data sheet
- Purge the resin to check if the resin is foamed, or check the moisture content of the resin

(2) Excessive gas / Decomposition of material

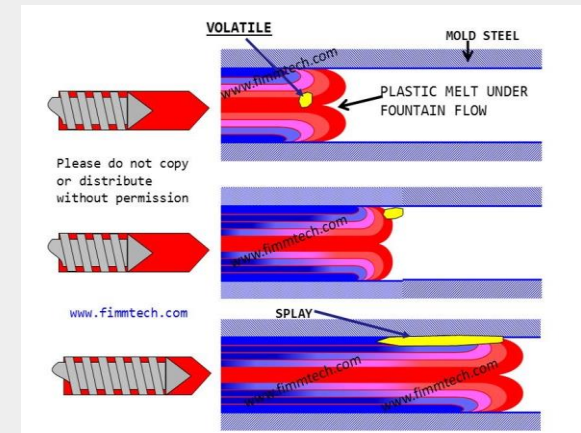
- Adjust plasticization condition : Adjust injection temp. / back pressure / screw RPM, check hot runner temp.
- Set weighing delay time, increase mold temp.
- Apply gas vent, check if excessive release agent was used, check pellet shape.

(3) Cold slug (Occurred by re-melting of solidified material)

- Install cold slug well
- Check String

(4) Abrupt changes in thickness

- Adjust multi-stage injection speed



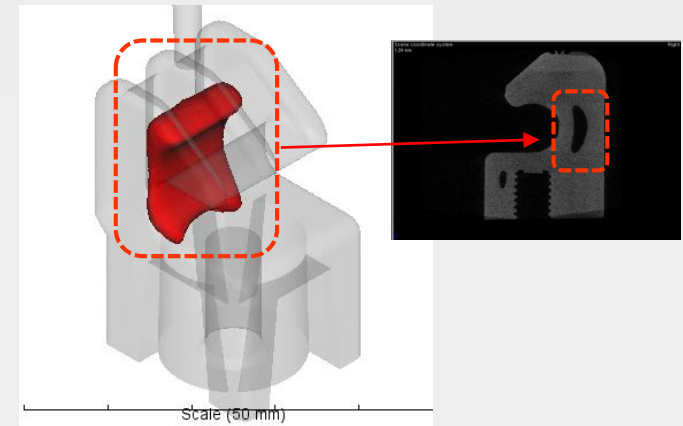
Source: <http://www.miconmould.com/plastic-mould-defects-burnt-streaks-9.html>

Void (Air trap)

➤ Air bubbles on the surface or the inside of the product

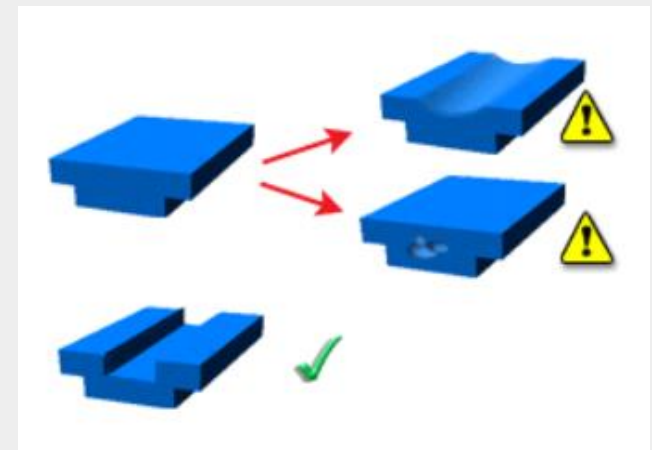
(1) Void by gas / moisture

- Set up the air vent at the defect area properly
- Enlarge the size of sprue & gate and increase mold temperature for pressure transfer
- Lower injection temp. to reduce gas
- Check whether there are any unevenness on cooling lines

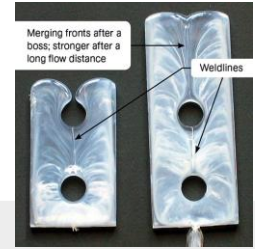


(2) Void due to shrinkage of molded products

- Shrinkage difference caused by cooling rate difference between skin & core layer generates void
- Usually occurs at thick products over 4mm thickness
- High temp. and low & long dwelling pressure can help prevent shrinkage
- Mold change required to get rid of thick parts



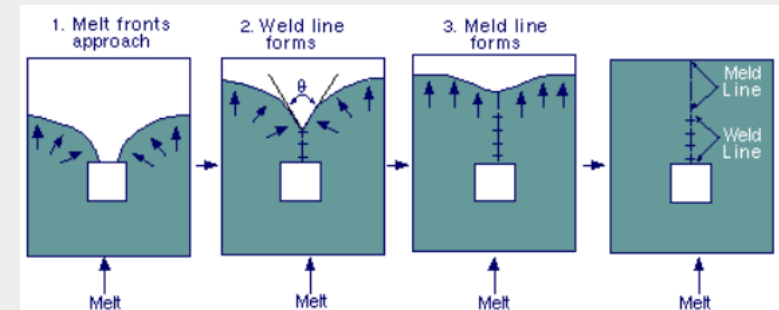
Weld-line



- A thin line which occurs at the junction of different flow fronts due to multi-gate system or obstacles such as a hole
- It is difficult to perfectly eliminate by simply adjusting injection conditions. However, it is possible to soften the visibility and strength of the weld-line.

(1) In case of the existence of obstacles such as holes.

- Change the position of the gate or obstacle to set a wider contact angle.
 - Move the weld-line to the side where it is not significantly related to the appearance of the final product.
- Increase injection temp., mold temp. and injection speed in order to increase flowability
- Set up the gas vent where weld-line appears
- Maintain the uniform thickness of the product
- * Other utilities (Increase mold temp.)
 - If E-Mold, RHCM, HIT are used, rapid heating can be set.

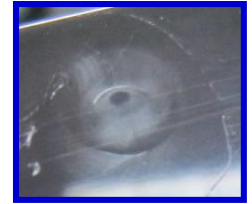


Source: <https://www.ptonline.com/> <http://www.imtechdesign.com/>

(2) In case of applying multi-point gate

- Appropriate valve gate control can reduce weld-line

Gate mark



➤ Refers to defects such as gas mark, flow mark, and halo occurring around the gate

(1) High shear rate near the gate

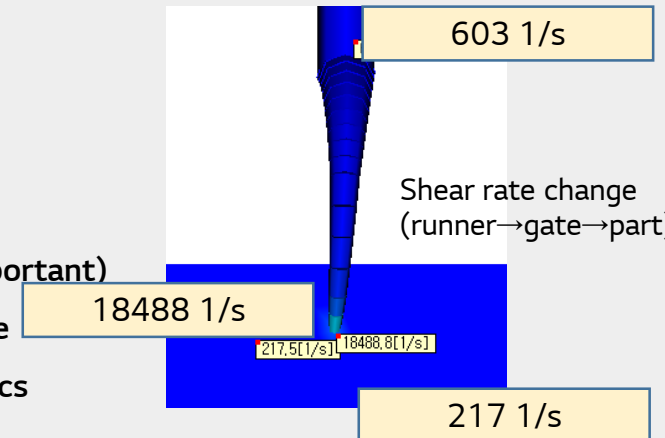
- Phase separation or migration of the additives with lower molecular weight due to high shear rate
- Gas mark due to high shear frictional heat
- Increase flowability by increasing injection temp., mold temp.
- Apply multi-stage injection speed to lower shear rate in narrow areas

$$\dot{\gamma} = \frac{4Q}{\pi r^3}$$

r : pipe radius
Q : flux

(2) Mold issues

- Increase the size of the runner and the gate
(appropriate ratio between width and depth is needed)
- Set up the gate in the thick part
- Set up the gate in the B side (where surface quality is comparatively not important)
- If it is multi-gate system, check if there exist excessive load on only one gate
- Apply appropriate gate type according to product shape / resin characteristics



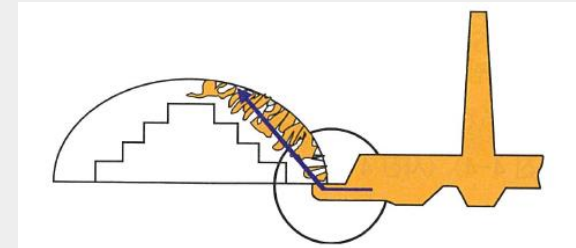
Jetting



➤ A phenomenon which the fusion of the pre-flow and the post-flow is entangled after passing through the gate, and is filled and solidified like the shape of a snake.

(1) Injection conditions

- Increase mold temp. and injection temp.
- Apply multi-stage injection speed system (especially lower the speed near the gate)



Source: <https://www.solidsolutions.co.uk/>

(2) Mold shape

- Change the position of the gate
- Widen the sectional area of the gate
- Install a pin on the gate to guide the flow to travel through the wall
- Change runner shape as S or C shape
- Change gate shape as tap-gate, pan-gate, or flash-gate etc.

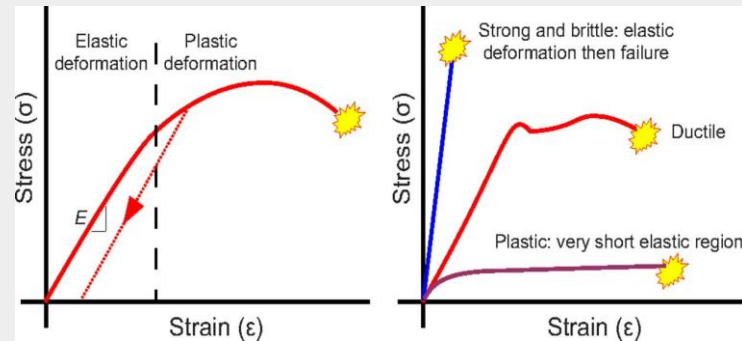
Crack

➤ A phenomenon in which the molded product is damaged or cracked when opening the mold or being ejected

- (1) When injection pressure is high or overcharged and the product is solidified with residual stress, cracks occur inside or outside the product when the shear stress exceeds the elastic limit.
- Thicken the part or change the weld-line position to increase the strength of the product
 - Increase injection temp. and mold temp., or injection speed to lower the injection pressure
 - Lower the dwelling pressure lengthen the cooling time or adjust V/P
 - Pre-heat if you use "Insert"
- (Heat expansion differs about 10 times, Plastic vs. Metal)

(2) Issues with mold

- Set the subtraction gradient that fits to the material and product.
- Adjust ejecting time/position
- Set up an undercut or remove to assure not to be shaken
- Set up a R value



Source: <https://www.apporo-cnc.com/>

Source: <http://feb.biologists.org/content/217/1/5>

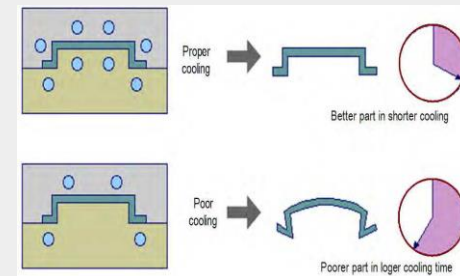
Warpage, Deformation



- Items bent or deformed from the original shape of the cavity after the product is ejected

(1) Warping due to uneven cooling

- Shrinkage deviation to the direction of the product thickness (concave to the hot side)
 - The high-temperature parts are gradually cooled resulting large shrinkage, and the low-temperature parts are quickly cooled resulting small shrinkage
- Box shaped products are deformed due to the slower cooling speed of the core compared to the cavity
- Apply the appropriate cooling system
- Make the mold inverse gradient or provide temperature difference to core cavity



Source: <http://www.macplas.it>

(2) Warping due to uneven shrinkage

- Thick part cools slowly, thin part cools quickly → shrinkage deviation (Thick part is concave)
- Minimize the thickness deviation between parts
- Set up the gate in the thick part, in order to deliver dwelling pressure toward the thin area

Troubleshooting Examples



Solution for Short Shot

Hybrid Connector, LUPOX GP1000K

Phenomenon

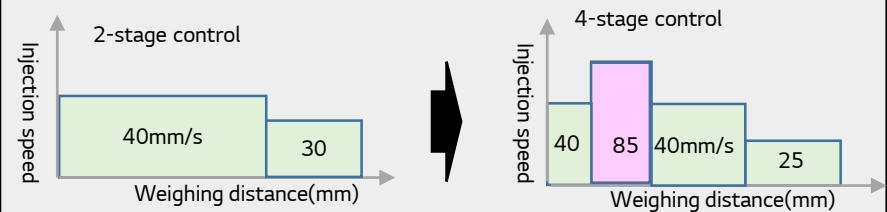
- Occurs at the thin area near the gate
(Main part thickness : 2T, Thin area : 0.8T)
- Occurs at the thin areas with
large thickness variations (3T→0.8T)

Cause

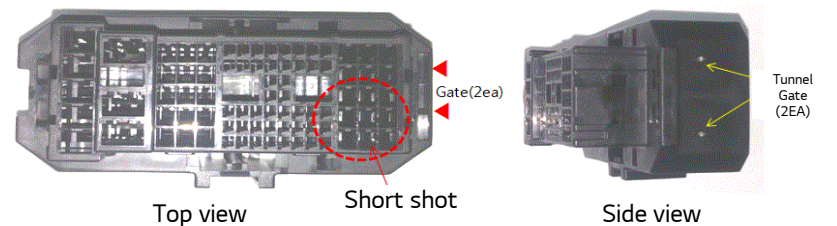
- Short shot occurs at thin areas with
large thickness variations :
The flow is stagnated due to the flow resistance of
the thin area. So, the thin part is re-filled after filling
the other thick main part which is easier to flow.

Solution guide

- Fundamental improvement plan :
Equalize the thickness of products
- Multi-stage control of injection speed
→ Increase speed at thin part



- Result : Short shot problem solved



Solution for Short Shot

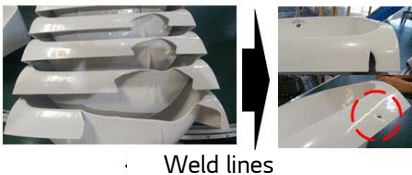
Bathroom vanity parts, ABS AF365FT

Phenomenon

- Short shot during injection process
- Adjusting injection condition did not help solving the problem

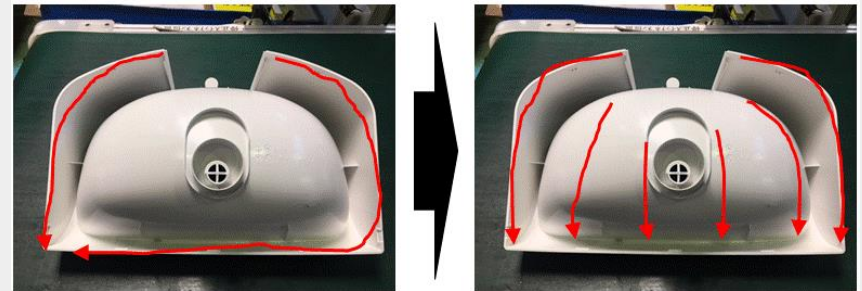
Cause

- Flame Retardant ABS has high flowability, but generates large amount of gas.
- Gas generated at the end of the flow is expected to result low stagnation.



Solution guide

- Add gas vents on the rib area, where short shot occurs
- Adjust flow balance by adding gas vents at the end, and by changing the thickness of product



Change flow pattern by changing thickness

- Result : Resolved short shot & weld-line problem
- If adjusting injecting condition does not work, it is necessary to consider mold modification to solve the problem.

Hesitation (Flow Congestion) case

Slim ODD, LUMILOY GN4356FH

Phenomenon

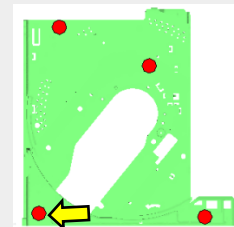
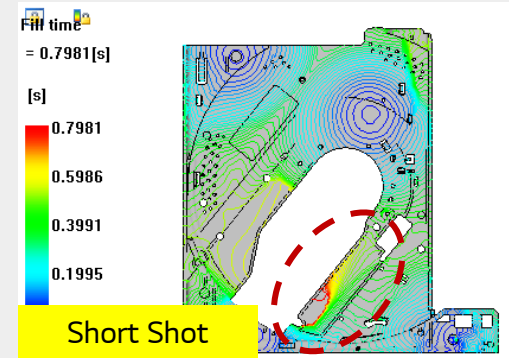
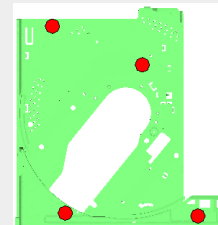
- Short shot at the thin area, near the gate
(Main part 1.2T, thin part 0.45T)

Cause

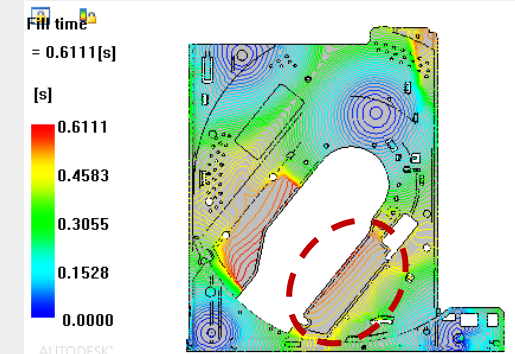
- Short shot occurs at thin areas with large thickness variations :
The flow is congested due to the flow resistance of the thin area, so the thin area is re-filled after filling the other main, thick part, that are easier to flow.

Solution guide

- Change gate position (From thin to thick part)
→ Short shot problem resolved



Relocated the Gate



Sink mark case

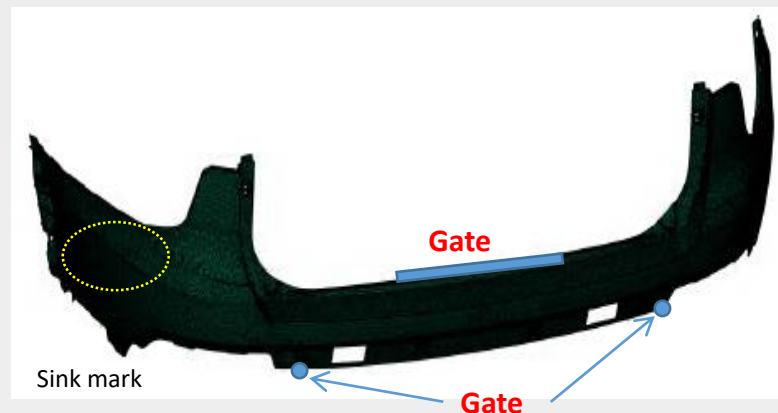
Auto Bumper, LUPOX TE5011

Phenomenon

- Sink mark occurs at the farthest area from the gate of large items (bumper C side)
- In-situ reaction :
If injection pressure & packing pressure are increased, the sink does not resolve but caused burr & peeling because of overpressure near the gate.

Cause

- The pressure cannot be delivered to the farthest area from the gate due to the long flow path
→ Increase flowability to improve pressure delivery



Solution guide

- Adjust injecting condition in order to deliver sufficient pressure to the farthest area of the product
 - Increase injection speed/ temp. if allowed
 - If necessary, set up the flow leader
 - Change the position of the gate or suggest to increase the gate size

Sink mark case

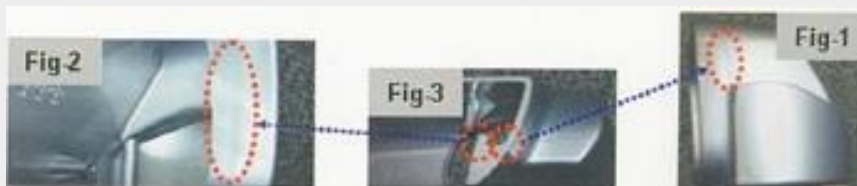
Side mirror housing, Heat Resistant ABS

Phenomenon

- Sink mark occurred at the thick part of "Side mirror HSG neck" (End part of filling)

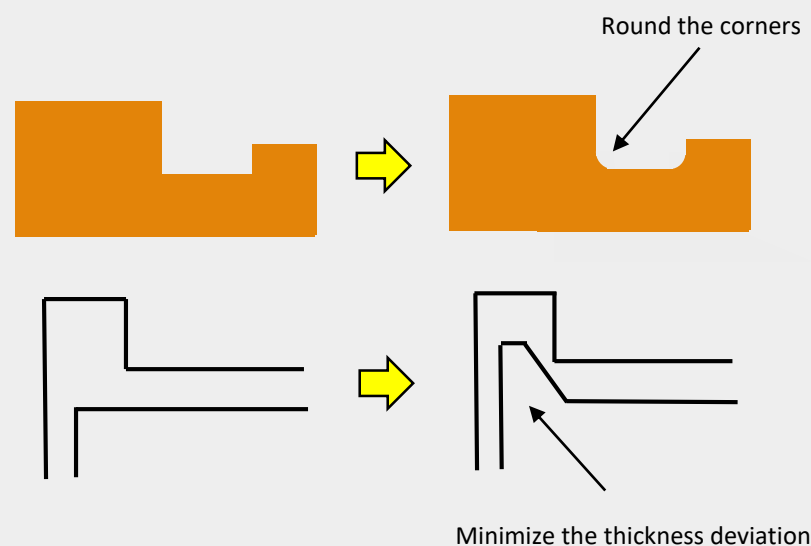
Cause

- Shrinking observed both inside and outside of the product
- As shown in the figure below, mold was designed thick, so pressure could not be delivered sufficiently.



Solution guide

- Enlarge cross-sectional area of the sprue/ runner/ gate to relieve pressure loss
- Minimize the thickness deviation
- Set up the R value

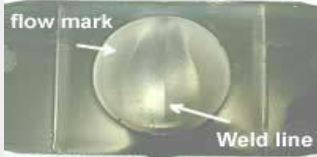
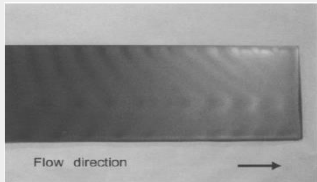


Flow mark case –Unstable flow of material

Wide Spiral, ASA L1941

Phenomenon

- Defect caused by uneven flow of material

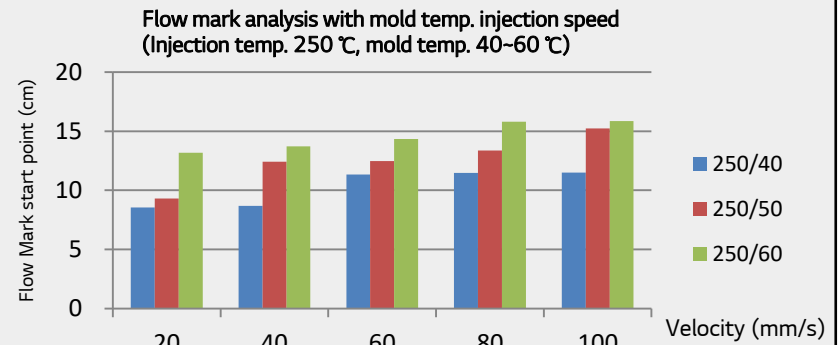
-  : Uneven thickness
-  : Staggered
-  : Record

Cause

- If viscosity is high and cooling rate is fast
- Orientation difference due to unstable flow

Solution guide

- Increasing injection speed and mold temp. would help to resolve the problem.



**If mold temp. increases, the start point of the flow mark is pushed back.

- Higher flowability might relieve the flow mark

(Orange line : flow mark starting point)



Standard



Fluidity Improved Material

Flow mark case –Thickness deviation

Airconditioner Front cover, ABS GT540

Phenomenon

- Surface Flow mark



~1000mm
Large-sized part

Cause

- This product has diamond pattern on its surface
- Flow mark occurred because of flow caught in the shape of the diamond

Solution guide

- To reduce the flow mark, apply multi-stage injection speed, and lower the overall speed at the same time : 30~40mm/s → 15mm/s
 - To prevent decrease in flowability, increase the injection temperature : 240 °C → 245°C
- Result : After continuous 600-shot test, 1% of defect rate was observed in mass production.

Flow mark case –Material's Shear deviation

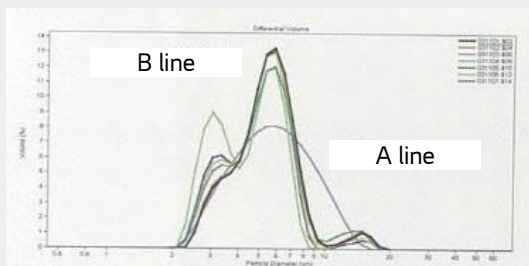
Copier cartridge, HIPS

Phenomenon

- Flow mark occurred on the exterior of copier cartridge, which used HIPS
- Observed severe flow mark and partial surface gloss defect in certain LOT
- No special issues with injection conditions

Cause

- After tracking base resin LOT, could figure out that there was rubber size deviation between each LOTs



Solution guide

- If the rubber size of the product is big, decomposition and flow degradation occurs when the shear is excessive, resulting poor surface quality.
- Lowering injection speed and increasing mold temp. help to decrease shear rate.



Silver streak case

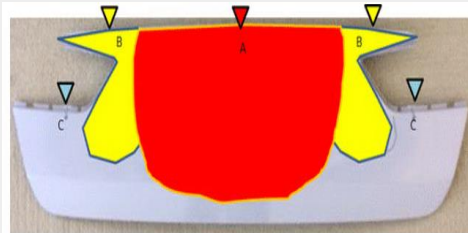
Back Panel, ABS XR404E

Phenomenon

- Silver streak occurred on gate C
(Valve opens in order of A → B → C)



Silver streak

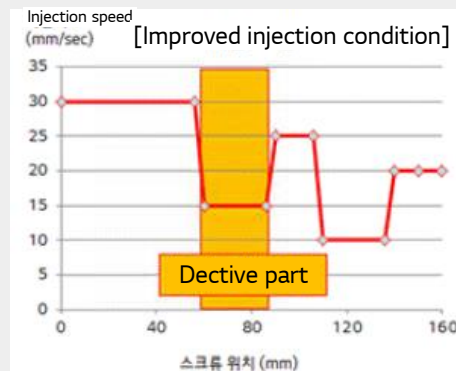


Cause

- Material filled from gate B solidified near the gate C
- If material is filled from gate C, solidified resin from gate B re-melts, which causes cold slugs
- When the resin of gate C is filled, the speed is fast

Solution guide

- Change the gate C opening time :
Set within the range where no solidified layer is created near gate C (Reduce valve open delay time)
- Apply multi-stage injection speed :
Reduce the injection speed from the point where the resin starts to be filled through gate C to the point where the defect occurs



Resolved



Void (Air trap) case

Inside door handle, LUPOX TE5000SP

Phenomenon

- Air trap occurred near the surface
- Visual defect observed on the upper matt part



Cause

- Air trap occurs due to flow pattern during injection due to product structure
- Lack of gas vent even the injection speed is high
- Degradation gas due to high resin temp.

Solution guide

- Apply multi-stage injection to reduce injection speed near the defective part, which allows filling without air trap to the end and gives enough time for gas venting
- Prevent thermal decomposition by reducing the cylinder residence time of resin through weighing delay
- Injection condition change
 - Terminal speed 30 mm/s → 7 mm/s
 - Cylinder residence time 10 sec → 1 sec
- Result : Air trap defect solved

Gate mark case

Body skirt, LUPOY NS5000CU

Phenomenon

- Surface is important as a non-painting product
- Gate mark appeared near the gate



Cause

- Processing window was narrow due to UV-stabilizer in the material
- If injection speed adjusted 3%↓, gate mark problem is solved but flow mark appeared
- Gate was set at the thin part (1T) and there was a step difference between gate and the product. And due to the narrow width of the gate, it gets high shear stress

Solution guide

- Gate design change
 - Remove step difference, widen the gate width
 - Suggest runner design change (I type → C type)
 - If gate structure is changed, Max. shear rate decreases from 11,000 1/s to 1,760 1/s
- After changing runner design & step difference: Gate mark and flow mark defect was resolved because processing window ↑

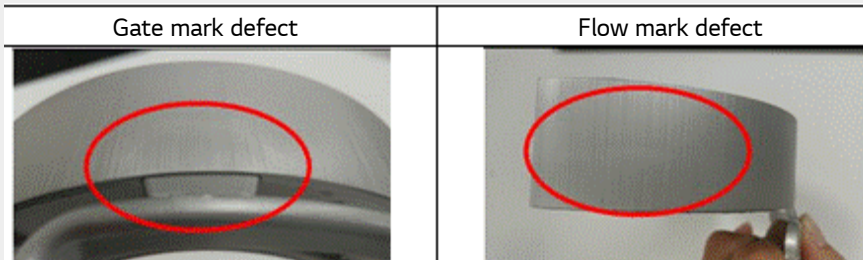


Gate Mark, Sink mark case

Water purifier machine Top cover, ASA LM915

Phenomenon

- Surface quality was important because it is non-painting top cover product
- Semi-circle shaped gate mark occurred the product



Cause

- Step difference between gate and the product.
- Both LGC's material and competitors' had same problem but customer didn't want to change the mold system.

Solution guide

- Adjust injection condition
 - 1st step (Gate) (24mm/s → Multi : 6~16mm/s)
 - 2nd~4th step (Main) : 11 mm/s
 - 5th step (End) : 16mm/s
 - Packing pressure : 30(1) → 80(5) Mpa (sec)
- Result : Sink mark, Gate mark defect solved



Jetting case

Auto Dust seal cap, High filler content material

Phenomenon

- Jetting is a defect that is caused by a material which passes the gate part so fast, that it reaches the end of the product and solidifies like a snake



Cause

- Normally caused when the gate size is small compared to the main part, and the injection speed is so high
- Related with material's elasticity

Solution guide

- Mold : Bigger gate size, apply fan/tab/overlap gate structure, or install the structure at the end of the runner or just in front of the product's gate (see below figures)
- Injection condition : lower the speed near the gate, higher mold temp., higher injection temp.



Set up a square pin



Set up S shape runner



Square pin at back part



Before
(Dust seal cap)



After
(Install structure at gate)

Crack case

Washing machine Top cover, Transparent ABS

Phenomenon

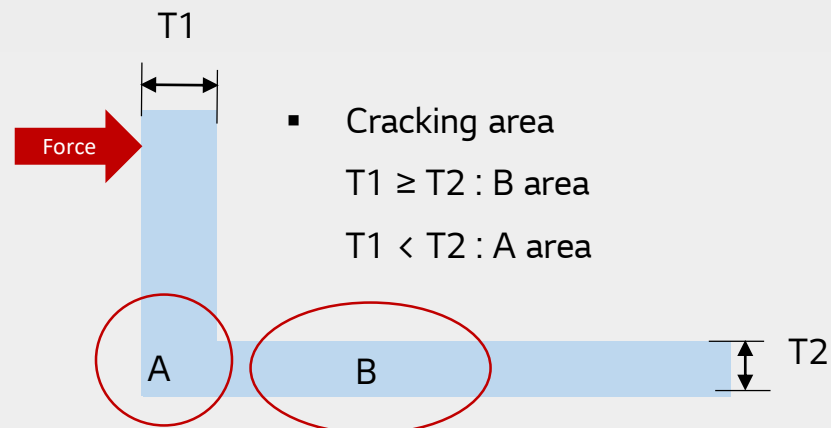
- Crack at washing machine top cover, which applied transparent ABS

Cause

- Crack occurs in transparent ABS after assembly with GP ABS
- Cracks due to residual stress because of thickness deviation



Solution guide



- Cracking area
 $T1 \geq T2$: B area
 $T1 < T2$: A area

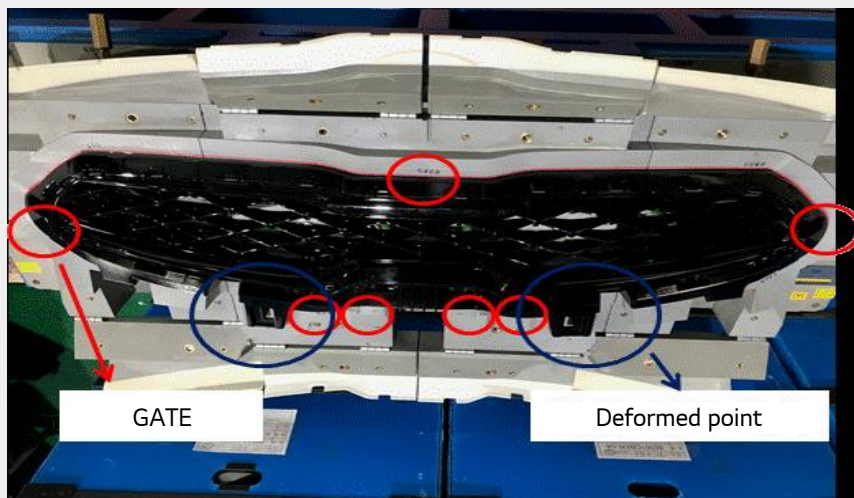
- This product : $T1 = 2.5\text{mm}$, $T2 = 2.3\text{mm}$
 - Crack occurred in B area
 - Changed T2 thickness to 2.5mm
 - Round the corner of A area

Deflection case

Radiator Grille, ABS XR404G

Phenomenon

- Lifting at the centre part of the product
- NG due to deformation of several specific areas shown below, while assembly



Cause

- Lifting occurs due to excessive deformation of a long product in a transverse direction
- Stress is concentrated due to high injection pressure as there are a large number of gates in the area where deformation occurred

Solution guide

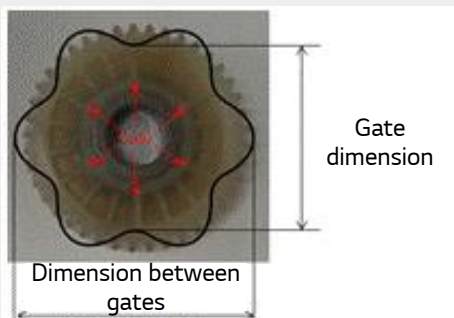
- By raising the rear mold temp., improved deformation direction : Fixed 50°C, Moving 40 °C → Fixed 50°C, moving 60°C
- Induce reduction in central part deformation by lowering injection pressure and packing pressure
- Result : Improved deformation

Deformation : Improving out of roundness

Idle Gear, LUXY KI25CR

Phenomenon

- Dimensional difference occurred between the gates



Cause

- Solidifying part before VP switching lead uneven pressure transmission
- Filling by packing pressure, not the injection pressure (VP switch starts at 80% filling)
- Uneven pressure at the end of the product

Solution guide

- Higher injection speed (10mm/s↑ for all sections) : Prevent solidifying before VP switch)
- Change VP switch position (20mm → 15mm) (VP switch after 98% filling)
- Reduce packing time (4s → 2s) : Applying the minimum time to compensate shrinkage
- Result :
 - Dimensional deviation OK (<0.1mm)
 - Weight OK

Solving size problem

Auto Wheel cover, LUMID HI5003A

Phenomenon

- W/cover weight, size was smaller than the spec (Higher shrinkage rate than the former material)
- In-situ reaction :
 - Burr occurs if increased packing pressure
 - Packing time didn't affect better sizing

Cause

- Additional shrinkage was found after painting (Post-processing)
- Increasing temperature causes burr because it reduces flowability

Solution guide

- Decrease injection temp. (290 °C → 270 °C), mold temp. (80 °C → 40 °C)
- Increased packing pressure to meet the weight spec
- Result : Size, weight OK



Color defect case

Window switch frame, Heat resistant ABS

Phenomenon

- Difference in color tone between main part and the assist part
- Defects occur after 10th cycle of production

Cause

- It is OK right after purging all the material, but defects occur again soon
 - Problem occurs in the barrel
- If there's a problem in the barrel temp. and weighing condition, material deformation causes color, surface defects
 - Need to check the actual temp.

Solution guide

- Heater 1 temp. was 500 °C, and this was figured out after checking the actual temp.
(Set temp. : 240°C)
- Repair Heater 1



Gas mark case

Clutch Pedal, LUMID GP2400AW

Phenomenon

- Material change from PA66 to PA6
- Pedal product : Ejecting pin marks
- Housing : Gas mark at the end of product



Cause

- High injection temp. (PA66's condition)
- Pedal : Ejecting before enough cooling
- Housing: No gas vents at the end of the product

Solution guide

- Pedal : Adjust injection temp.
 - Temp(°C) : 320 → 280
 - Cooling time(s) : 15 → 25
 - Ejecting speed(%) : 40 → 30
- Housing : Mold change
 - Set up a overflow
- Result : Ejecting pin mark, Gas mark Improved



Gloss, color improvement on embossed surface

Over Head Console, LUPOY EU5000GA

Phenomenon

- No gloss and color deviation on smooth side
- Gloss and color difference occurs with respect to embossed pattern

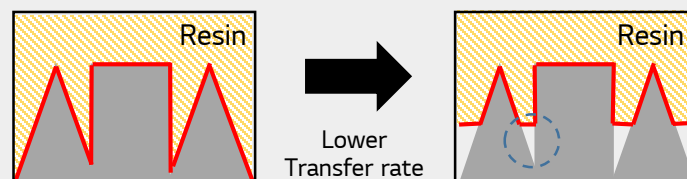


Cause

- Difference of diffuse reflection due to different embossed pattern lead gloss & color difference
- Different transfer degree leads different diffuse reflection

Solution guide

- Decrease mold temp. ($85\text{ }^{\circ}\text{C} \rightarrow 35\text{ }^{\circ}\text{C}$)
: Lower the transfer rate, higher the surface gloss
- Decrease packing pressure
: Improved surface gloss, color difference by lowering transfer rate

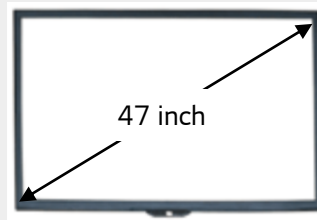


Mold-deposit case

TV Front cover, LUPOY GN5151FL

Phenomenon

- Mold-deposit occurred
- Surface polishing after 4hours of production
- 2hours of polishing time



Cause

- Large injection machine is used because it is a 47inch TV front cover, but the actual injection volume is so small and staying time long (Use 30mm out of 600mm barrel size)
- The size of hot runner and manifold is big, which makes material stay longer in a mold
- Plasticizing condition are not optimized yet.

Solution guide

- Partial temp. drop excluding the actual injection volume of the cylinder



<Actual injection volume ~30mm> <Temp. decrease around 20~30°C>

- Adjust injection condition
 - Back pressure 0 bar → 10~15 bar
 - RPM 50 % → 30 %
 - Set weighing delay time 0 sec → 22 sec
 - Decrease cushion 16mm → 3~5mm
- Result : No mold-deposit more than 24hours

Ejecting defect case

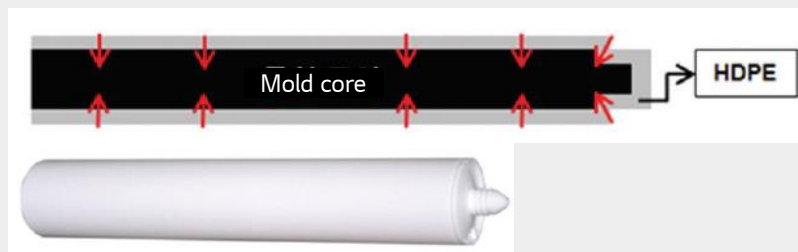
Catridge, HDPE SM800

Phenomenon

- Ejecting problems

Cause

- Suggested new products, which have higher fluidity
- As the flow improves, the resin is filled more under the same injection conditions, and the material gets trapped in the mold core.



Solution guide

- MI was same before and after (Catalyst change)
- To observe viscosity change in a high shear area, fluidity test was done in a same screw size condition with the customer's (90mm)
- Catalyst change lead fluidity shift for about 50°C, and the problem was solved after lowering injection temp.

Temperature [°C]	Holding pressure [bar]	SM800 Weight [g]	ME8000 Weight [g]
200	30	76.3	69.8
	80	89.2	83.5
	110	89.0	83.6
	140	87.6	83.9
230	30	87.0	76.3
	80	93.5	88.3
	110	92.6	88.1
250	140	93.0	88.0
	110	94.5	89.3
260	30	90.3	79.1
	80	94.6	90.0
	110	95.6	90.4
	140	95.1	89.9

Antimal tail case

Hot runner mold, LUPOX GP1000DS

Phenomenon

- Animal tail(string) defect near the HRS mold gate
- Affects the next shot due to contamination

Cause

- Constant heat transmission at the manifold of the hot runner system leads solidifying difference.
 - Continuous heat energy transmission
 - Slow solidification near the gate tip
 - Cutting is not appropriate
- Occurs easily in crystalline material than in amorphous materials
- Occurs easily on non-reinforced material

Solution guide

- Lower the hot runner temp. and breaks tip :
Lower 20~30°C (Should be adequate to prevent cold slug)
- Maintain tip melting by applying higher temp. :
+30°C (Only change the tip temp. to prevent fluidity changes)



Sprue Bush blocking case

Phenomenon

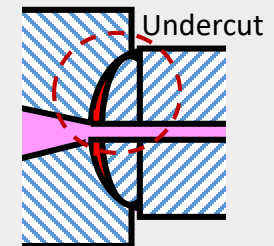
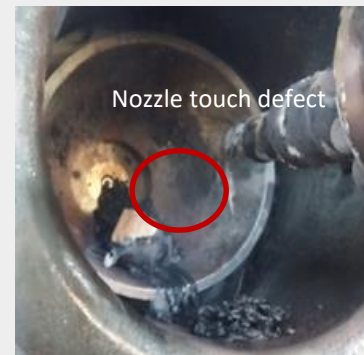
- Leakage in the nozzle touch part
- First shot is OK, but hard to inject from 2nd shot
- Leaked resin was found when the nozzle was separated

Cause

- Mold problem :
Sprue Bush roundness and orifice phi error
May vary depending on material properties
(Easily found in high viscosity, high filler content materials)
- Free space of the nozzle touch lead leakage :
Rapid cooling after leakage may cause defects
- Gets serious when drooling occurs

Solution guide

- Check sprue bush design and whether locate ring is damaged
- Adjust weighing condition (when not serious)
 - Prevent drooling (Set low back pressure, and excessive 2nd suckback)
 - Increase nozzle temp. (+20°C or more)
 - Use injection part retraction
(Need to remove leaked material)



Painting POP and Crack cases

Phenomenon

- Usually occurs near the outermost part of the ejected product, parting line
- POP and crack from certain period

Cause

- Typical chemical crack defect
- No raw material issues, and thinner analysis showed difference in ingredients and content of before and after the defect [T-725 (Company D)]

Solution guide

- Ingredient change of thinner affects injected plastic products.
- Especially the outermost part of the product could be damaged easily due to higher stress level.
- The lower the temperature of use, the lower the volatilization rate of contact chemicals, which intensifies the defects. So proper managing is a must

Ingre dient	EA	MEK	MIBK	TOL	BA	XYL	PMA	Volatilization spped	△δ
Dec. (OK)	5.4	3.6	7.7	16	8.7	47.6	11.0	1.25	0.21
Jan. (NG)	11.3	8.4	13.9	35	15.1	16.3	-	1.94	0.25

Troubleshooting Guide

Recommended	Defects												
	Surface void	Crack	Burr	Carbonization	Bigger size	Surface Defect	Weldline	Short shot	Silver Streak	Sink Mark	Small size	Big void	Deflection
Change gate position			8				9						9
Polish mold surface				5		8							
Clean up vents	5			2		9	8	7	6			10	
Check contamination	6	4		6					5			9	
Check uneven mold temp.													1
Check mold coordination			7										
Dry the material	1	6		7		6			1			1	
Higher weighing							7	3		5	5	7	
Higher back pressure						5			4			6	
Higher die clamping force			6										
Higher cooling time													7
Higher packing pressure							3	4		2	1		
Higher packing time							4			3	2	5	
Higher injection pressure						3	2	2		1	3	4	3
Higher injection speed						2	1	1			4		
Higher injecting time					6								
Higher mold temp.		7			4	1	6	5			7		4
Bigger the gate size								9	9	8	8	11	
Bigger the runner size / add them								10		9	9	12	
Bigger the sprue size								11		10	10	13	
Bigger the vent size				4			9	8	8				
Move gate to a thicker part										7			
Higher the injecting temp.					5	4	5	6			6		
Redesign ejecting part													8
Decrease recycling portion		5										8	
Lower the back pressure		2											
Lower the injection temp.	2	1	4	3	7				3			3	6
Lower the packing pressure			3		2								5
Lower the injection pressure			2		1								
Lower the injection speed	3		1	1	3	7			2	4		2	
Lower the mold temp.			5							6			2
Decrease cycle time					6								
Decrease the screw speed	4	3							7				

Confider

*** Priority can be changed according to various molded product structures/ defect types