Injection molding problems and Troubleshooting

一. Short shot 產品未完成

Short shot occurs when insufficient material is enjected and can't completely fill cavities.
Short shot can be caused by machine, mold or material.
當射出的材料不足且無法完全填充型腔時，就會出現短射。短射可能是由機器、模具或材料造成的。

Possible Cause	Correction action
Melt temperature too slow 原料溫度過低	Adjust melt temperature upward 提高料管溫度
Insufficient feed 塑膠原料不足	Increase shot size 增加原料用量
Short injection time 射出時間過短	Increase injection time 增長射出時間
Injection speed too slow 射出速度太慢	increase speed 加快射出速度
Low mold temperature 模具溫度過低	Increase mold temperature 增高模具溫度
Non-uniform cooling 模具溫度不均	Check water throughtput and line layout 重整模具水管
Poor venting in the mold 模具排氣不良	Adjust proper vent position or add vents 調整合適位置或加適度之排氣孔
Clogged nozzle 射嘴阻塞	Clean the nozzles 清洗阻塞
Feed material unbalanced 進膠不均勻	Redesign correct gates position 重開模具溢口位置
Gate or spill hole too small 澆口或溢口太小	Increase gate diameter 加大澆口或溢口
Inadequate lubricant for plastic 原料內潤滑劑不夠	Increase lubricant in material 酌加潤滑劑
Poor melt plastic flow 塑膠原料流動性不良	Add lubricant or change plastic 酌加潤滑劑或更換使用原料
Damage on check ring 螺桿止逆環(過膠圈)磨損	Check and replace 拆除檢查修理
Non correct machine 機器效能不適合	Use proper machine 更換適合的機器

二. Shrinkage / 縮水

Shrinkage is the contraction of a plastic moulded component
as it begins to cool after the injection process
塑料成型部件開始冷卻後, 會發生收縮. 收縮比率應塑料而異, 設計模具時要考量之

Possible Cause	Correction action
Insufficient filling 塑膠原料不足	Increase filling vollume 增加原料用量
Injection pressure too low 射出壓力不足	Raise injection pressures 提高射出壓力

Short injection time 射出時間太短	Increase injection time 增長射出時間
Injection speed too fast 射出速度太快	Reduce speed 減少射出速度
Unbalanced feeding to gate 溢口不平衡	Adjust gate positions or gate diameter 調整模具入口大小或位置
Clogged nozzle 射嘴阻塞	Clean the nozzles 清洗阻塞
Melt Temperature too high 溫度過高	Reduce temperature 降低溫度
Insufficient cooling 冷卻溫度不夠	Add cooling time 酌延冷卻時間
Poor venting 排氣不良	Add vents on shrinking parts 在縮水處設排氣孔
Injection nozzle too small 料管太小	Replace larger diameter nozzle 更換較大規格料管
Insufficient pressusre holding time 保壓時間不足	Increase holding time 延長保壓時間
Damage on check ring 螺桿止逆環(過膠圈)磨損	Check and replace 拆除檢查修理
Material too soft 材料過軟	Change material 更換原料
Large shrinking rate 收縮率過大	Change material 更換原料

三. Parts Sticking 成品黏模

When the mold is opened, the product get stuck to the mold, it cannot fall off

開模時，產品粘在模具上，不能脫落.

Possible Cause	Correction action
Injection pressure too high 射出壓力太高	Reduce injection pressures 降低射出壓力
Over filling 射膠量過多(填料過飽)	Use release agent 使用脫模劑
Pressure holding too long 保壓時間太久	Reduce injection time 減少射出時間
Injection speed too fast 射出速度太快	Reduce injection speed 降低射出速度
Melt material temperature too high 料溫太高	Reduce material melt temperature 降低料溫
Unbalanced filling 進料不均使部分過飽	Adust gate hole diameter or posiiton 變更溢口大小或位置
Insufficient cooling time 冷卻時間不足	Increase cooling time 增加冷卻時間
Mold temperature too high or too low 模溫過高或過低	Adjust mold temperature 調整模溫
Undercuts in the mold 模具內脫模倒角	Eliminate undercuts 修模具除卻倒角
Mold walls not smooth 模具內壁不光滑	Polish mold walls 拋光模具

四. Sprue sticking
澆道 (水口) 黏模

During injection, sprue gets sticking, the sprue cannot fall off when the mold is opened.
注塑時，澆道粘連，開模時澆道不能脫落。

Possible Cause	Correction action
Injection pressure too high 射出壓力太高	Reduce injection pressures 降低射出壓力
Melt material temperature too high 料溫太高	Reduce material melt temperature 降低料溫
Sprue diameter too long 澆道過大	Modify the mold 修改模具
Not enough sprue cooling time 澆道冷卻時間不夠	1. Add cooling time 延長冷卻時間 2. Reduce barrel temperature 降低料管溫度
Draft angle too small 澆道脫模角不夠	Increase angle 修改模具增加角度
Sprue bushing and nozzle dismatched 澆道凹弧與射嘴之配合不正	Adjust to match 重新調整與配合
Sprue surface not smooth 澆道內表面不光滑	Maintain the mold 檢修模具
Undercuts 脫模倒角	Maintain the mold 檢修模具
Damage sprue 澆道外孔有損壞	Maintain the mold 檢修模具
No sprue puller pin 無澆道抓鎖	Add sprue puller pin 加設抓銷
Over filling 填料過飽	1. Reduce injection volume 降低射出量 2. Shorten injection time 減短射出時間 3. Slow injection speed 降低射出速度

五. Flash 毛頭(毛邊) 、披風

When melt plastic flows out of the cavity at the parting line or between inserts, excess are formed on surface of molded product
當熔融塑料在分型線或嵌件之間流出型腔時，成型品表面形成多餘的料邊

Possible Cause	Correction action
Melt plastic temperature too high 原料溫度太高	1. Reduce melt temperature 降低原料溫度 2. Reduce mold temperature 降低模具溫度
Injection pressure too high 射出壓力太高	Reduce injection pressure 降低射出壓力
Filling too much 填料過飽	1. Reduce injection volume 降低射出量 2. Shorten injection time 減短射出時間 3. Slow injection speed 降低射出速度
Poor melding line or bad sealing surface 合模線或靠密面不良	Maintain the mold 檢修模具
Low clamping pressure 鎖模壓力不夠	Raise clamping pressure 增加鎖模壓力

Hold pressure too long 保壓時間過長	Reduce hold pressure time 減少保壓時間
Injection speed too fast 射出速度過快	Slow injection speed 降低射出速度
Melt plastic flow too fast 塑膠原料流動性太好	Change plastic 更換原料

六. Cracking or Crazing
開模時或頂出時成品破裂

A breakage occurs when the mold is opened or product is ejected
開模或頂出產品時發生破損

Possible Cause	Correction action
Overfeed material 填料過於飽和	1. Reduce shot volume 降低射出量 2. Reduce injection time 減短射出時間 3. Reduce injection time 降低射出速度 4. Reduce enjection pressure 降低射出壓力
Hold pressure too long 保壓時間太長	減少保壓時間
Mold temperature too high 模溫太低	升高模溫
Insufficient draft allowance 脫模斜度不足	Check the mold 檢修模具
Gate diameter too small 澆口太小	Check the mold 檢修模具
Ejector pin too small or wrong position 頂針不夠或位置不當	Check the mold 檢修模具
Mold Undercuts 模具有倒鉤	Check the mold 檢修模具
Excessive Moisture in material 材料吸濕性良好	1. Dry material 進行烘料 2. Change plastic material 更換塑膠原料
Vacuum occurs when mold is drafted 脫模時模具產生真空現象	1. Slow to open mold or reduce eject speed 降低開啟模具或頂出的速度 2. Use intake equipment 使用進氣設備

七. Weld or Meld line
結合線

A weekness or visible flaw created when two or more flow paths meet during the filling process.
在填充過程中兩條或更多流路相遇時, 因流動尖端的溫度稍微下降, 而互不熔合產生的條痕及缺陷。

Possible Cause	Correction action
Material flow too slow 原料流動性不良	1. Increase melt temperature 提高原料溫度 2. Increase back pressure to raise screw shaft speed 提高背壓加快螺桿轉速 3. Change material 更換原料
Mold temperature too low 模具溫度過低	Increase mold temperature 提高模具溫度

Injection speed too slow 射出速度太慢	Increase injection speed 增加射出速度
Injection pressure too low 射出壓力過低	Increase injection pressure 提高射出壓力
Contaminated material or mix with foreign material 原料不潔或滲有它料	Check material 檢查原料
Insufficient lubricant 潤滑劑不足	Less use Release agent 脫模油盡量少用或不用
Sprue and gate diameter too small 澆道及澆口太小	Increase lubricant 酌增潤滑劑
Air trapped in the cavity 模穴內空氣排除不及	Add vents or check if vents get clogged 增開排氣孔或檢查是否有阻塞
No overflow well 缺乏溢流井	Add overflow wells at the under end of welding line 在結合線處之下端增設溢流井

八. Flow lines 流紋

A flow pattern as linear grooves or circular ripples on the surface of the molded product which is the track of melt plastic flow within the cavity of the mold
成型品表面的直線凹槽或圓形波紋，是熔融塑料流痕在成品表面的現象

Possible Cause	Correction action
Improper material melt 原料熔融不佳	1. Increase melt temperature 提高原料溫度 2. Increase back pressure to raise screw shaft speed 提高背壓加快螺桿轉速
Mold temperature too low 模具溫度過低	Increase mold temperature 提高模具溫度
Injection speed too slow 射出速度太慢	Increase injection speed 增加射出速度
Injection pressure too low or too high 射出壓力過低或過高	Adjust injection pressure 調整適度壓力
Contaminated material or mix with foreign material 原料不潔或滲有它料	Check material 檢查原料
Gate hole diameter too small 溢口過小	Increase dimension 加大溢口
Not even thickness of product 成品斷面厚薄相差太多	Redesign product or spill hole location 變更成品設計或溢口位置

九. Silver streaks 銀紋(銀線)、氣瘡(起倉)

A shining line shaped patterns appear on the surface of the molded product.
在成形品表面，順著塑膠流動的方向出現銀白色條紋的現象

Possible Cause	Correction action
Moisture in material 原料含有水份	1. Dry material 原料徹底烘乾 2. Increase back pressure 提高背壓

Material temperature too high or overheat mold to produce air 原料溫度過高或模具過熱導致氣體產生	Reduce material temperature and mold temperature 降低原料溫度及模溫
Material contains additives like lubricant, dying pigment and so on 原料中含其他添加物如潤滑劑、染料等揮發物	Reduce useage or replace high temperature plastic 減少其使用量或更換耐溫較高之替代品
Poor mix among additives 原料中其他添加物混合不均	fully mix additives 徹底混合均勻
Enjection speed too fast 射出速度太快	Slow enjection speed 減慢射出速度
Mold temperature too low 模具溫度太低	Increase mold temperature 提高模具溫度
Material pellets not uniform 原料顆粒粗細不均	Use uniform pellets 使用粒狀均勻之原料
Poor flow path in the mold 原料在模內流程不當	1. Adjust gate dimension and position 調整溢口之大小及位置 2. Keep mold temperature stable 模具溫度保持平均

十. Dull surface finish
成品表面不光澤

Product surface become clouded, it has low gross or Less reflection on molded surface

產品表面變渾濁，表面有低粗反射或較少反射

Possible Cause	Correction action
Mold temperature too low 模具溫度過低	Increase mold temperature 提高模具溫度
Low filling material 原料用量不夠	1. Increase enjection volume 增加射出量 2. Extend enjection time 延長射出時間 3. Increase enjection pressure 增加射出壓力
Excessive release agent in the cavity 模內過多脫模油	Clean and dry it 擦拭乾淨
Moisture in the mold 模內壁有水	Wape and check if water leak out 擦拭並檢查是否有漏水
Rough wall of the mold 模內壁不光滑	Polish the mold 模具拋光

十一. Deformation
成品變形

Deformation could be reference to bending, cracking and warping of the item.

產品的彎曲、開裂和翹曲都是變形

不良原因	解決對策
No cooling at product ejection 成品頂出時尚未冷卻	1. Reduce mold temperature and material temperature 降低模具溫度及原料溫度 2. Extend cooling time 延長冷卻時間
Geometry misaligned or thickness uneven 成品形狀及厚薄不對稱	1. Use a frame to fix shape after molded product is ejected 脫模後以定型架固定 2. Redesign product 變更成品設計
	1. Reduce ejection volume 降低射出量

Improper enjection pressure and speed 射出壓力或速度不適當	2. Reduce ejection time 減短射出時間 3. Reduce injection speed 降低射出速度 4. Reduce enejction pressure 降低射出壓力
Unbalanced filling 進料不均勻	Change gate location 更改溢口
Unbalance ejected system 頂出系統不平衡	Improve ejected system 改善頂出系統
Material temperature too high 原料溫度過高	Reduce material temperature 降低原料溫度
Insufficient or excessive melt material from gate 進溢口部份之原料太鬆太緊	Reduce or increase mold closing time 增加或減少閉模時間
Cavity temperature and core temperature not even 公母模溫度不均	Balance cavity temperature and core temperature 均衡公母模溫度
Unbalanced enjection pressure 射出壓力不當	Balance mold pressure 平衡模具壓力

十二. Voids/Bubbles
成品內有氣孔

Defined as gaps or pockets with no material inside molded product
為產品內部的空隙, 沒有填料的區域.

Possible Cause	Correction action
Product cross section , rib or pole too thick 成品斷面，肋或柱過厚	Change product design or relocate gate positions 變更成品設計或溢口位置
Enjection pressure too low 射出壓力過低	Increase enjection pressure 提高射出壓力
Insufficient enjection time 射出時間不足	Increase enjection time 增加射出時間
Runner or Gate opening too small 澆道或澆口太小	Increase runner and gate hole dimentiosn 加大澆道及澆口
Enjection speed too fast 射出速度太快	Decrease enjection speed 調慢射出速度
Material contains moisstusre 原料含有水份	Fully dry material 原料徹底乾燥
Difficult to melt high temperature material 原料溫度過高以致分解	Reduce material temperataure 降低原料溫度
Uneven mold temerature 模具溫度不均	Adjust mold temperature 調整模具溫度
Cooling time too long 冷卻時間太長	Reduce mold cooling time , use watherbath 減少模內冷卻時間，使用冷卻水浴
Insufficient back pressure 背壓不夠	Increase back pressure 提高背壓
Improper barrel temperature 料管溫度不當	Decrease temperature for nozzles and at front half section ; increase temperature at half final section 降低射嘴及前段溫度，提高後段溫度

十三. Black Specks 黑紋

Black streaks or spots appear on the surface or within molded parts

成型產品表面或內部出現黑色條紋或斑點

Possible Cause	Correction action
Melt temperature too high 原料溫度過高	Reduce temperature 降低料管溫度
Screw speed too fast 螺桿轉速太快	Reduce screw speed 降低螺桿轉速
CONTAMINATED RAW MATERIAL 原料中混有雜質	Check raw material or clean enjection system 檢查原料或清洗射出系統
Small nozzle tip 射嘴孔過小導致噴射不順	Enlarge diameter 重新調整孔徑
Burned substances in the barrel 料管中有燒焦材料	Clean the barrel 清理料管

十四. Brittlenes 脆化

The tendency of a molded product to break or crack, at times shatter

模製成品容易破裂或破裂，有時會破碎的現象

Possible Cause	Correction action
Temperature too high 溫度太高	Reduce barrel temperature 降低料管溫度
Back pressure too high 背壓過高	Reduce back pressure 降低背壓
Screw shaft rotary speed too fast 螺杆迴轉速度太快 (樹脂分子鏈斷裂)	Reduce screw speed 降低螺桿迴轉速度
Not complete plastic drying 塑料乾燥不完全	Dry material completely 依原料種類進行適當乾燥
High volume of recyled resin 回收二次料加太多	Reduce volume of recycled material 減少回收二次料用量
Recycled material contain foreign substances 回收料中含有其他種材料	Add same resin for recycled and virgin material 添加回收料時使用與被添加樹脂同一材質