ASCEND®
PERFORMANCE MATERIALS

Problem:

Short Shots

A short shot is a plastic part that is underfilled or contains sink marks. In most cases the part is not usable and must be scrapped. Short shots occur most frequently during startup. They can also occur during normal production due to incorrect machine settings, equipment failure or material issues.

Short shots

TECH TIPS

Your Vydyne® technical services team offers step-by-step troubleshooting tips to help locate and solve problems that may occur during injection molding in order for you to get production back up and running.

Problem:

Short Shots

Root causes

Several factors can cause short shots

- Process setup
- Melt temperature
- Venting
- Non-return valve
- Mold temperature
- Material issues
- Barrel/press size
- Part and tool design
- Machine issues

How to troubleshoot

Evaluate the molding process

Many factors in the molding process can cause short shots. If the process is not set up properly, it may not be able to accommodate normal variations in material moisture, ambient conditions, etc.

- *Proper shot size:* When initially setting up a mold it is recommended to run an apparent viscosity vs shear rate curve to define the ideal injection speed. Start by filling the part between 95% and 98% on first stage. The remaining 2% to 5% will be filled using pack and hold. The shot size and transfer position are then set to provide the final process conditions. Using this approach will produce a part that is consistent in size, properties and function.
- *Injection speed:* One typical cause of short shots is an injection speed set too low. Injection speed settings are sometimes set to an arbitrary value without running an apparent viscosity versus shear rate curve to define ideal fill speed. This approach ensures the apparent viscosity curve is at its flattest point to overcome viscosity variations that may occur. Small variation in moisture, RV, ambient conditions in the plant will influence melt viscosity.
- *Melt temperature:* If the melt temperature is too low, the melt viscosity is too high and makes it more difficult to fill the cavity. The process should be set so the barrel temperatures produce a melt temperature in the 518°F to 572°F (270°C to 300°C) range. Also, check for broken thermocouples and heating components on the barrel and in the hot manifold. The temperature PID controller needs to be checked and calibrated to reflect appropriate temperature reading.
- *Barrel residence time:* A longer residence time puts more heat energy into the material with the optimal residence time for PA66 at three to five minutes. If the residence time is too short, the melt temperature may not be at the desired set point. It may be required to increase the barrel temperature settings 15°F to 30°F (8°C to 17°C) across the barrel and run the process until you are at steady state and then take a melt temperature.
- *Mold temperature:* Consistent temperatures across the mold are vital in producing quality parts. For Vydyne materials, we recommend a mold surface temperature range between 100°F and 190°F (38°C and 82°C). If mold surface temperatures fall below this range, the Vydyne material will not flow as far and could cause short shots. The material also can begin to set before the part is completely filled, causing flow hesitation and weak knit-lines.

Evaluate the mold and machine design

- *Non-return valve:* If the press is not holding a consistent melt cushion from shot to shot, inspect for a worn, broken or blocked non-return valve. Sometimes a piece of metal or foreign debris can lodge in the non-return valve, blocking it from sealing off properly. This will cause material to slip past the non-return valve during injection and produce a short shot. Most times, a worn valve is the main cause of inconsistent melt cushion or this will keep it form sealing off properly.
- *Mold venting:* If the mold is not vented properly, the hot gases at the flow front will be trapped at end of fill or in rib walls, etc. The gas will push back on the polymer's flow front, causing burning, flow hesitations and short shots. Vents should be cut into the mold every 2 in. (50 mm) along the parting line from the gate to the end of fill. The runner systems also should be vented.
- *Poor cavity balance:* Parts with more than one gate or multi-cavity tools that are not flow balanced correctly can cause short-shots. Mold-filling simulation software is an excellent tool to help part designers balance multi-gated/multi-cavity parts. Sharp cooling rate differences between cavities could cause fill unbalances. Check cavity steel temperatures if unbalanced fill is identified via a short shot study.
- *Runner/gate and nozzle design:* When runners, gates and nozzles are too restrictive, a large pressure drop in these areas can result. The majority of available injection pressure is depleted before the material completes filling the cavity. Some tips to consider:
  - A full, round runner is the best design for a cold runner. It promotes the lowest pressure loss.
  - The recommended cold runner diameter range for nylon is 0.16 in. to 0.35 in. (4 mm to 9 mm), depending on part thickness. The runner never should have a diameter less than the thickest section of the plastic part to be molded.
  - Below are recommended gate dimensions for Vydyne resins.

Recommended Dimensions for Edge and Drop Gate Designs for Nylon Materials

Gate Type	Nominal Wall Thickness mm (in.)	Gate Thickness mm (in.)	Gate Width mm (in.)	Land Length mm (in.)
Edge	0.762 (<0.030)	to 0.762 (0.030)	to 1.14 (0.045)	0.762–1.27 (0.030–0.050)
Edge	0.762–3.18 (0.030–0.125)	1.27–2.03 (0.050–0.080)	1.27–2.03 (0.050–0.080)	0.762–1.27 (0.030–0.050)
Edge	3.18–6.35 (0.125–0.250)	1.52–3.56 (0.060–0.140)	1.52–3.56 (0.060–0.140)	0.762–1.27 (0.030–0.050)

Recommended Dimensions for Edge and Drop Gate Designs for Nylon Materials

Gate Type	Nominal Wall Thickness mm (in.)	Gate Diameter mm (in.)	Land Length mm (in.)
Pin	3.18 (0.125)	0.762–1.27 (0.030–0.050)	0.762–1.27 (0.030–0.050)
Pin	3.18–6.35 (0.125–0.250)	1.02–3.05 (0.040–0.120)	0.762–1.27 (0.030–0.050)

Source: multiple industry sources

- *Part design:* The part should be designed so material flows from the thickest wall section to the thinnest. This will ensure proper filling and avoid short shots due to frozen material in the thin-walled sections.

Problem:

Short Shots

Evaluate the material

- *Moisture:* Nylon material, both wet and dry, can cause a molded part to short shot. When the material is wet, screw slippage can occur. The melt also becomes compressible. When the material is over-dried, the relative viscosity can increase, resulting in higher first-stage pressures to fill. These higher pressures may become restrictive resulting in a short shot.
- *Material nucleation:* A product with the ability to nucleate more will flow a shorter distance and set up quicker in the mold. If you are experiencing short shots with a grade of Vydyne that is highly nucleated, consider switching to a similar grade with lower nucleation capabilities.

For more information, contact our expert applications specialists or visit **ascendmaterials.com.**

Rev. 04/2020 TT008

inspiring everyday