TROUBLESHOOTING

Troubleshooting Guide

STEPS TO RESOLVE

1. Get a clear understanding of what the issue is
2. Document the issue
3. Get a photo
4. Get a sample
5. Shipment information- Carton labels, PO#, Frequency of problem, etc.
6. Use your resources
   - Reach out to the vendor
   - RAC
6. Communicate updates regularly with client

BLACK SPOTS/SPECS

Black spots and brown streaks appear as dark spots or streaks in the molded part and are usually caused by thermal damage to the melt.

Possible Solutions

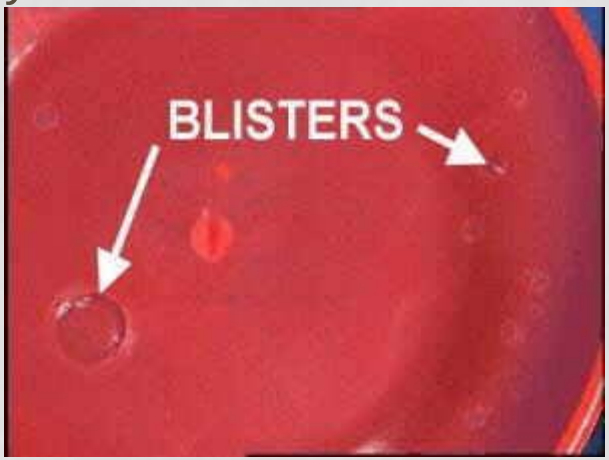
- Check the material for contamination.
- Decrease the melt temperature.
- Decrease the overall cycle time.
- Purge and/or clean the screw and barrel.
- Decrease the screw speed. High screw speeds may cause the material to degrade.
- Material may have too much regrind content.
- Material may be over dried. Decrease drying time/temperature. Refer to drying instructions provided by the material supplier.
- Material may be prone to thermal degradation. It may be necessary to use a more thermally stable material.
- Dead spots may be occurring, ensure that the alignment between the machine nozzle and mold sprue is correct.
- Residence time may be too long, or the shot size may be too small for the machine. It may be necessary to move the mold to a machine with less injection capacity.

BLISTERING/BUBBLES

- Blistering – These are bubbles or raised imperfections that are generally caused by too much heat and/or inadequate cooling

There are only two possibilities:

1. Trapped gas, which includes air, moisture vapor, volatiles from the resin, or decomposition gases from the polymer or additives.
2. Vacuum void.



Use a heat gun or something similar to gently warm the area of the part where the bubble is, the bubble should change form.

- If it is a gas bubble, the gas will warm up and expand, raising the surface, and often will pop as the part surface softens.
- If there is no air in the bubble and it's a vacuum void instead, the bubble will collapse due to the atmospheric pressure pushing on the softened surface of the part.

BRITTLENESS

Brittleness is a condition where the part cracks or breaks at a much lower stress level than would normally be expected based on the virgin material properties.

Possible Solutions

- Check for material contamination.
- Decrease amount of regrind use.
- Decrease back pressure.
- Decrease injection pressure.
- Decrease screw speed.
- Increase melt temperature.
- Dry material. Refer to the drying instructions provided by the material supplier.

COLD SPOTS

Sometimes, during the process of Extrusion blow molding, un melted material or cold spots appear on parisons that is also called Marbleizing. The appearance of these spots is a very serious problem that should be immediately resolved. In order to solve this, we have to know the possible causes that may lead to this problem.

- **Cause**: If the melt Temperature set is too low, it can cause cold spots in parison. Basically, due to the low temperature, melt gets cold, and resultantly Unmelt material appears in the form of clod spots.
- **Solution**: To prevent un melted material, increase the melt temperature.

- **Cause**: At times, inadequate “heat soak” time before the start-up can be a source of cold spots on parison.
- **Solution**: Simply increase the time of “heat soak”.

- **Cause**: If the heaters, thermocouples or controllers of Extrusion blow molding machine are defective, resin melt will not get heated properly and may lead to the emergence of cold spots on parison.
- **Solution**: Change or repair faulty heaters and controllers. Besides that, there may be some problem in the adjustment of these heaters, so calibrate them as needed.

- **Cause**: If you are using engineering resins, but your heater capacity is inadequate especially in areas between cylinder and head, or in head/die zones, it may lead to cold spots or unmelted material on parison.
- **Solution**: For the sufficient heating of engineering resin, upgrade the size of heater or provide thermal insulation.

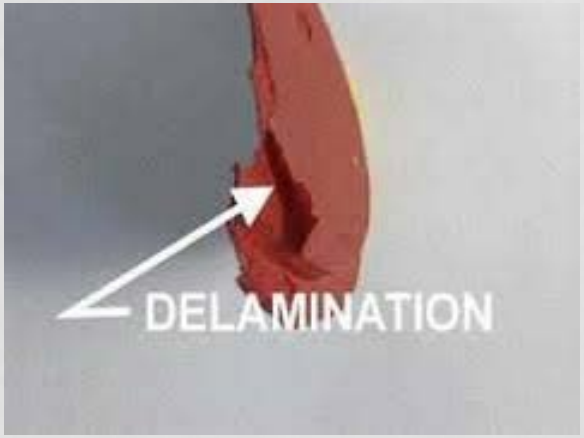
- **Cause**: Any air draught or leakage will lead to the External (or internal) cooling of head/die and create cold spots on parison.
- **Solution**: Take necessary actions to eradicate the source of air cooling.

DELAMINATION

Delamination occurs when single surface layers start flaking off the molded part.

Possible Solutions

- Adjust injection speed.
- Check for material contamination. Incompatible resins or colorants may have been accidently mixed causing this condition to be seen.
- Dry material.
- Increase melt temperature.
- Increase mold temperature.
- Insufficient blending. Check melt homogeneity and plasticizing performance.



DIM APPEARANCE OF BOTTLES

Oddly looking bottles in the whole lot can be easily identified. This happens due to the appearance of the bottle. People often get confused between marble tint and dim appearance. However, both are way different from each other. In the later one, the outer surface of the bottle becomes cloudy. This reduces the mechanical strength of the bottle too many folds. Also, makes it appear dim.

- **Cause of Defect** – There are several reasons that lead to the dim appearance of the bottle. A preform appears to be cloudy white tinted if the temperature range and cooling range goes beyond a certain level. PET container is heated up to the range of temperature that goes beyond the chilling point of 125°C. Another reason for the defect is the improper blowing of the bottle. If the bottle is kept for stabilization for a long-time due to slow speed, the tinted effect appears.

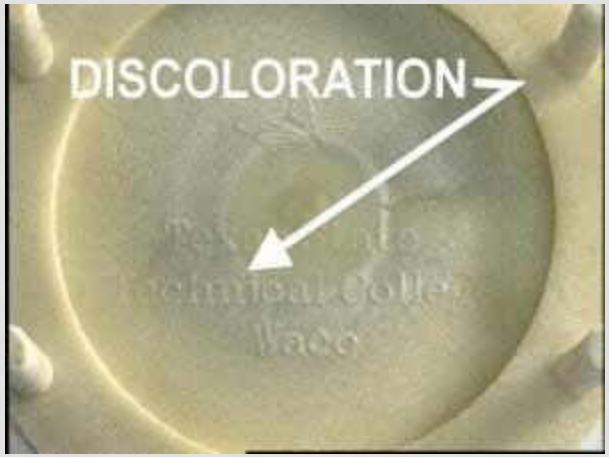
- **Remedy**– To rectify the defect, the heating temperature has to be decreased. This can be done by any of the following methods: Reduce the number of lamps or intensify the furnace ventilation. This lowers the percentage of heating area and adjusts the rate of heating to reduce the temperature of performing below the critical temperature.

DISCOLORATION

Discoloration – similar to burn marks or brown streaks but generally not as dark or severe. It may cause the part to be a darker shade than the virgin pellets and is often found nearest the gate area; however, it can also appear as
dark streaks throughout the part.

Possible Solutions

- Check hopper and feed zone for contamination.
- Decrease back pressure.
- Decrease melt temperature.
- Decrease nozzle temperature.
- Move mold to smaller shot-size press.
- Provide additional vents in mold.
- Purge heating cylinder.
- Shorten overall cycle.



EXCESSIVE FLASH

- Excessive flash is often seen near sealing faces, out of vent grooves, or down ejector pins. The extra plastic that breaks off on neck. It appears as thin or sometimes thick sections of plastic where it would not be on a normal part.

- Note: Flash can very quickly (within a few cycles) damage the parting line surfaces.

FISH EYES

Definition
Fish eyes are a surface defect that results from unmelted material being pushed with the melt stream into the cavity and appearing on the surface of a molded part.

Causes
Low melt temperature
If the melt temperature is too low to melt the material completely, the un melted pellets will merge with the melt stream, marring the surface of the part.

Too much regrind
The shape and size of regrind is irregular compared with original material, and can trap more air and cause the material to blend unevenly.
Incompatible materials blended together

Low screw rotation speed
If the screw rotation speed and the back pressure setting are set too low, there might not be enough frictional heating to melt the material completely in the barrel before the injection.

- Remedies
- Reduce regrind material
- Contact material suppliers to get the recommended levels of regrind to use.
- Optimize melt temperature
- Modify screw design
- Contact material suppliers to get the right screw design information to avoid improper melt mix or overheating that leads to material degradation.

Solving one problem can often introduce other problems to the injection molding process. Each option requires consideration of all relevant aspects of the design specification.

FLOW LINES

- Description: Flow lines are streaks, patterns, or lines - commonly off-toned in color - that show up on the prototype part as a consequence of the physical path and cooling profile of the molten plastic as it flows into the injection mold tooling cavity. Injection molded plastic begins its journey through the part tooling via an entry section called a “gate.” It then flows through the tool cavity and cools (eventually hardening into a solid).

- **Causes:** Flow line defects are caused by the varying speed at which the molten plastic flows as it changes direction through the contours and bends inside the mold tool. They also occur when the plastic flows through sections with varying wall thickness, or when the injection speed is too low causing the plastic to solidify at different speeds.

Remedies:

- Increase injection speeds and pressure to the optimal level, which will ensure the cavities are filled properly (while not allowing the molten plastic time to start cooling in the wrong spot). The temperature of the molten plastic or the mold itself can also be elevated to ensure the plastic does not cool down sufficiently to cause the defect.

- Round corners and locations where the wall thickness changes to avoid sudden changes in direction and flow rate.

- Locate the gate at a spot in the tool cavity with thin walls.

PANELING

- Paneling occurs when the pressure inside a plastic bottle becomes less than the ambient air pressure outside, causing the walls of the bottle to partially collapse inward.
- The result is an aesthetically unappealing product package with a distorted shape and badly puckered or wrinkled labels.
- Research has shown that a consumer's first reaction to a bottle affected by paneling is often, "there must be something wrong with it," potentially leading to a loss of business and damage to the brand.
- Fortunately, the causes of paneling are easy to identify and just as easy to avoid.

PANELING

Cause #1 - Temperature / Pressure Change

- When air is heated, it expands. Conversely, when air inside a closed container is cooled, it contracts, reducing the internal pressure and triggering the conditions that can cause paneling.
- This is most noticeable in the case of bottles that are hot filled and sealed before cooling. As the product cools, the contents and/or the air in the headspace contract, creating a negative pressure within the bottle. This negative pressure can cause the side panels to suck in to compensate for the loss of product volume.

FIVE HELPFUL SOLUTIONS

- Allow the hot filled bottle to cool before sealing it.
- Increase the structural integrity of the bottle walls through design mechanics means such as ribbing or vacuum panels - think Gatorade bottle.
- Reduce the headspace in the bottle by increasing the fill level.
- Ensure your bottle is of proper gram weight and consistent wall thickness to combat paneling.
- Design the bottles so that paneling will result in the entire label being intentionally “sucked” into a flat, aesthetically appealing surface.
- One of the most effective solutions to combat panel sink is to improve the structural integrity of the bottle by slightly changing the shape, from a cylinder to an oval for example. The design change adds structural integrity by replacing flat areas with a more rigid surface that can resist the pressure.

PANELING

Cause #2 - Oxygen Absorption / Reaction

- It is possible that oxygen in the headspace of the bottle is being absorbed in the bottle's contents, depending on the nature of the product. Oxygen may also be reacting with one or more of the product's ingredients. Since oxygen comprises about 16 percent of the volume of air, reducing its concentration can significantly reduce the bottles internal pressure, leading to bottle paneling.

THREE HELPFUL SOLUTIONS

- Reformulate the product to remove ingredients with which oxygen is reacting.
- Purge the headspace with liquid nitrogen in order to remove oxygen before the bottle is sealed.
- Increase the fill level of the product to reduce the size of the headspace and minimize the effect of oxygen absorption

PANELING

Cause #3 - Gas Permeation through Bottle Walls

- It is possible that the walls of the bottle may be permeable to the vapor of one or more of the ingredients in the product.
- LDPE, HDPE and Polypropylene (PP), for example, are excellent moisture barriers but poor gas barriers, although the higher the density, the better their barrier properties. PET, on the other hand, is an excellent oxygen barrier but not a great water vapor barrier. Any vapor permeation through the bottle walls can result in unequal pressure and paneling.

FOUR HELPFUL SOLUTIONS

- Reformulate the product to eliminate the problem ingredient(s).
- Change to a resin with barrier properties that match the requirements of the ingredient(s) causing the problem.
- Consider utilizing a barrier, such as fluorination or a multi-layer barrier resin in the design of the bottle. Either will reduce the gas/vapor permeation of the product through the walls of the bottle. Fluorination also reduces the chemical permeation, weight loss, odor emission, and flavor or fragrance loss.
- Explore the use of vented liners to equalize the inside/outside pressure on the container. The liners allow gas and moisture to pass in or out of the container.
- Remember, any change in resin would require elevated stability and compatibility studies in order to ensure there will be no adverse reactions between the product and container resin.

PANELING

Cause #4 - Fill Processing

- In some cases, machinery used to package the product may pinch or compress the bottle before capping, reducing the volume (and the pressure) inside the container.

- If the packaging machinery cannot be economically replaced, the solution may lie in redesigning the container itself. Effective design changes could include:

  - Fillers should review their manufacturing and production processes for any possible mechanical situations that may be causing panel compression (i.e. areas that rails may be constricting the container or exerting excessive force during packing).

PANELING

Cause #5 - Environmental Conditions

- To avoid paneling issues, environmental conditions such as altitude, temperature and relative humidity, also need to be taken into account.

- When a product is packaged at an elevation of over 5,000 feet, for example, its internal pressure will be equal to the external atmospheric pressure, which is considerably lower than that at sea level. If the product is later sold at sea level, the difference in internal and external pressures can cause paneling – and this just one example of how environmental conditions can affect this issue.

THREE HELPFUL SOLUTIONS

- Explore the use of the vented liners recommended above - allowing gas and moisture to pass in or out of the container will equalize the inside/outside pressure. This can be an excellent solution for containers that fill product at high altitude and then ship to distribution at lower geographic elevations.

- Pressurize the bottle slightly during filling to compensate for the drop in atmospheric pressure.

- Consider treating both the bottle and cap with a florination process that will serve as a barrier against the problem ingredient(s). (see Cause #3 above).

PIN HOLES

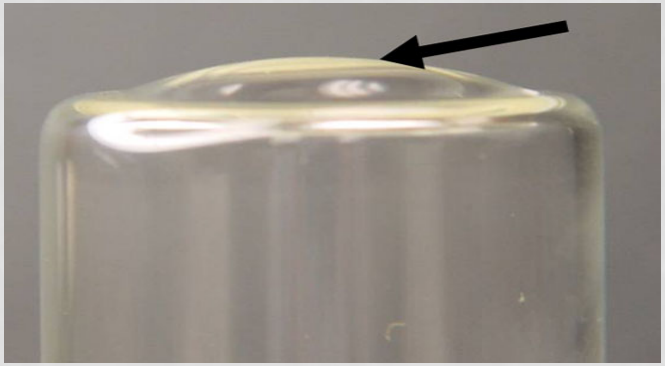
There are two reasons for your pin holes:

1. Residual air in fibers tows that were encapsulated by resin flowing through the weave openings. After resin plugs the vacuum port, no more air can be removed. Capillary action causes resin in the weave to wick into the tows, displacing the air into the weave openings. The solutions are: use a better vacuum (so there is no residual air to trap), or to slow down the infusion so that the wicking of resin into the tows closely follows the flow front through the weave (so the resin can push the residual air out the vacuum port).

2. You may be boiling the resin by applying too much vacuum after you close the resin inlet. Resins will always have some volatiles (including the resin itself). After closing the resin inlet and the vacuum line is full of resin, the vacuum should be reduced to 10 inHg. This provides 10 inHg of compaction pressure on the fibers while maintaining 20 inHg on the resin during cure.

ROCKER BOTTOM & OVAL NECKS

During the extrusion blow molding operations, sometimes the bottoms of finished containers get rocker bottoms and the necks acquire an oval shape. These two defects happen in blow molded objects as the result of Warpage. The reasons behind the faults of Rocker Bottoms and Oval Necks are the same that cause the warping of parts.



- **Cause:** Insufficient mold cooling is considered as the most major cause of part Warpage.
- **Solution**: Enhance the water flow and check if the Warpage is stopped or eliminated.

- **Cause**: If the melt or stock temperature is too high, it can cause Warpage of part.
- **Solution**: In order to normalize the melt temperature, decrease the heat of mold. But it is important to note to always drop the mold temperature by small amounts to check if this will resolve the problem without creating new defects such as cold spots etc.

ROCKER BOTTOM & OVAL NECKS

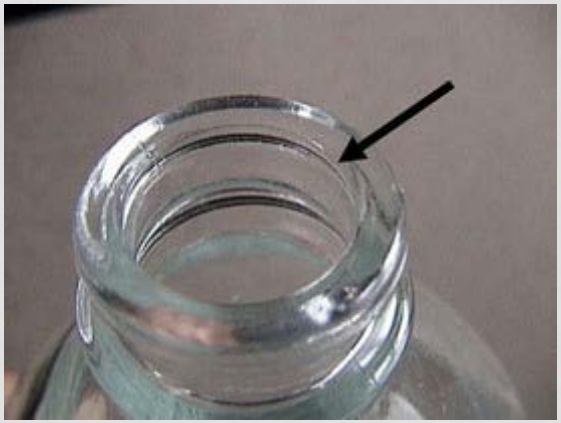
- **Cause**: Sometimes, the cooling channels of mold get blocked and become a source of Warpage.
- **Solution**: To determine the blockage of cooling channels, check the throughput. If it is considerably low than when the mold was new, it means a detailed cleaning of channels is required. A through clean-up will help to eliminate the obstructions of cooling channels. In addition, boost up the cycle time to provide longer cooling.

- **Cause**: Too short cycle time, is another possible reason of Warpage of part that leads to the defects of Rocker Bottoms and Oval Necks.
- **Solution**: So, if the cycle time is quixotically short, adjust it as needed.

- **Cause**: Inadequately designed cooling channels may also possibly cause Warpage of part.
- **Solution**: The design of the mold may have to be modified so as to increase the cooling capacity of the mold. Besides that, the design must facilitate the uniform cooling.

- **Cause**: If the part or container that is being blow molded is poorly planned and designed, it could cause Warpage in the part.
- **Solution**: Check the distribution of material in the part for unnecessarily thick or thin sections. And if you find any variation, increase the cycle time and let the part be thoroughly cooled. Besides that, it may be necessary to change the parison programming and redesign the part.

- **Cause**: Sometimes, when the Mold opens before being fully exhausted, it leads to the Warpage of parts and consequently Rocker Bottoms and Oval Necks.
- **Solution**: In order to sort out this problem, increase the exhaust time of mold.



SINK MARKS

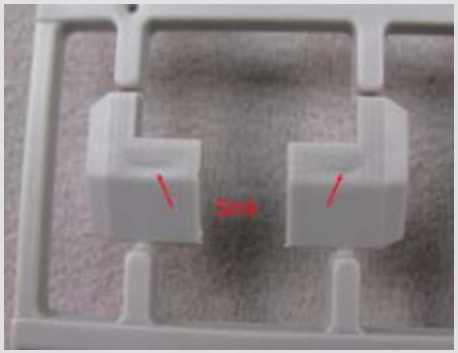
- Description: Sink marks are small craters or depressions that develop in thicker areas of the injection molded prototype when shrinkage occurs in the inner portions of the finished product. The effect is somewhat similar to sinkholes in topography, but caused by shrinkage rather than erosion.

- **Causes:** Sink marks are often caused when the cooling time or the cooling mechanism is insufficient for the plastic to fully cool and cure while in the mold. They can also be caused by inadequate pressure in the cavity, or by an excessive temperature at the gate. All else being equal, thick sections of the injection molded part take longer to cool than thin ones and so are more likely to be where sink marks are located.

Remedies:

- Mold temperatures should be lowered, holding pressure increased, and holding time prolonged to allow for more adequate cooling and curing.

- Reducing the thickness of the thickest wall sections will also ensure faster cooling and help reduce the likelihood of sink marks.



SPLAY MARKS, SILVER STREAKS

Splay Marks, Silver Streaks are usually caused by water vapor blisters at the flow front burst and freeze on the wall of the molding surface.

Possible Solutions

- Check for contamination.
- Decrease melt temperature.
- Decrease nozzle temperature.
- Dry resin pellets before use, as per the manufacturers recommendations.
- Incorrect storage of pellets. Moisture on the pellets could be transferred into the melt, especially if the resin is not normally pre-dried.
- Raise mold temperature. This will prevent condensation on the mold walls from being carried into the melt.
- Ensure the mold is not leaking water onto the cores or cavities. Again, this will prevent condensation on the mold walls from being carried into the melt.
- Relocate gates on or as near as possible to thick sections.
- Shorten overall cycle.

STRESS CRACKS

Stress cracking – or Crazing is caused by high internal stress during molding or by an external force imposed upon the part. They can also be caused by an incompatible external chemical being applied to the finished parts. The cracks often don't appear until days or weeks after the parts have been molded.

Possible Solutions

- Decrease injection pressure.
- Dry material.
- Increase cylinder temperature.
- Increase mold temperature.
- Increase nozzle temperature.
- Modify injection speed.
- If the material is partially crystalline, then it may help to reduce the mold and/or melt temperature.
- If the material is amorphous then it may help to increase the mold and/or melt temperature.

SURFACE FINISH (LOW GLOSS)

Surface Finish (Low Gloss). Gloss is the appearance of the surface of the molded part when light is reflected off of it. Molds that are textured or resins that are filled have an inherently reduced level of gloss when compared to highly polished mold surfaces.

Possible Solutions

- Clean mold surface.
- If the part design allows, increase the polish of the molding surface.
- Increase cylinder temperature. This applies to molds that have a polished surface.
- Increase injection pressure. This applies to molds that have a polished surface.
- Increase injection speed. This applies to molds that have a polished surface.
- Increase mold temperature. This applies to molds that have a polished surface.
- Decrease cylinder temperature. This applies to molds that have a textured surface.
- Decrease injection pressure. This applies to molds that have a textured surface.
- Decrease injection speed. This applies to molds that have a textured surface.
- Decrease mold temperature. This applies to molds that have a textured surface.
- Increase melt temperature.
- Make sure venting is adequate.

SURFACE FINISH (SCARS, WRINKLES)

Surface Finish (Scars, Wrinkles). Is the appearance of the ripples or wrinkles on the surface of the molded part.

Possible Solutions

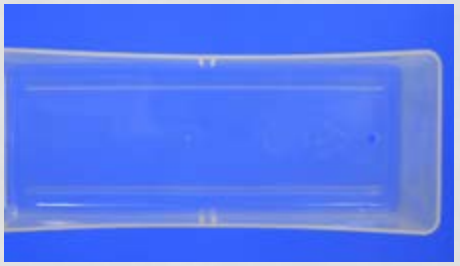
- Decrease back pressure.
- Decrease nozzle temperature.
- Increase booster time.
- Increase the melt temperature.
- Increase injection pressure.
- Increase injection speed.
- Increase overall cycle time.
- Increase shot size.
- Inspect mold for surface defects.

WARPING

- Description: Warping (or warpage) is the deformation that occurs when there is uneven shrinkage in the different parts of the molded component. The result is a twisted, uneven, or bent shape where one was not intended.

- **Causes**: Warping is usually caused by non-uniform cooling of the mold material. Different cooling rates in different parts of the mold cause the plastic to cool differently and thus create internal stresses. These stresses, when released, lead to warping.

Remedies:



- Ensure that the cooling time is sufficiently long and that it is slow enough to avoid the development of residual stresses being locked into the part.

- Design the mold with uniform wall thickness and so that the plastic flows in a single direction.

- Select plastic materials that are less likely to shrink and deform. Semi-crystalline materials are generally more prone to warping.

SCUFFING

Scuffing or Scratching

- Can occur when labeled products are shipped long distances or frequently come in contact with each other.

COMMON DECO ISSUES

COPY ERRORS

Grammatical Errors

- While this is a common human error in many fields, keeping your label free of spelling and grammatical errors is important.

- Proofreading before submitting artwork is crucial.

COLOR OFF

Inconsistent Color

- Inferior label printers have difficulty matching color from one label run to the next and sometimes even within a single order.

DIE CUTS

Cause

- Process of using a die to cut or trim material out of something.
- The “die” usually made of sharpened steel, stamps out irregular shapes out of a sheet.
- Dies can get dull over time which leads to bad cuts or nicks.

Solution

- Check grain direction. Cuts against the grain have a tendency to rip.
- The board against which the die presses needs to be a strong material.
- Using a “kiss” technique, which is when you first press into the material cutting just the top layer, leaving the back layer to b cut net instead of cutting it all at once.

WIND DIRECTIONS

The unwind direction is important when applying labels. So if you are working with a co-packer, communicate their unwind needs before the label makes it to press. Regardless of if it is a product of flexographic label printing or digital label printing, this is an important step in the process.

TIMEFRAME

	Expectations of EmpireEMCO	Number of days for response
1	Number of days to have custom item sourced:	5
2	Number of days to have custom item quoted:	7
3	Number of days to have stock item sourced:	3
4	Number of days to have stock item quoted:	3
5	Number of days to expect sample receipt of stock item:	5
6	Number of days to enter sales order and/or purchase order upon receipt of customer PO:	2
7	Number of days to acknowledge customer purchase order:	2
8	Number of days to update open order due to resin increase/decrease:	3
9	Number of days for quality issue resolution based upon customer supplying information in a timely manor - Initial report:	4
10	Number of days for quality issue resolution - Final report:	13
11	Number of days for quality issue resolution - Credit issued:	16