Kruse Training Cause-and-Effect Seminar

July 16-18, Salt Lake Community College

$1,750 / Person

3-Day Seminar Outline

Welcome to Kruse Training's Circle of Knowledge Seminar – a program focused on the "cause and effect" of injection molding. We aim to help develop synergistic teams of cross-trained professionals who can successfully design and mold plastic components.

DAY ONE

Lesson 1: Molding Fundamentals: Inside the Mold

A foundational understanding of what happens inside a mold during the injection molding process is key to a successful plastic industry career. From part designers to mold designers to process engineers, having a good grasp of these molding fundamentals is essential to designing and producing high-quality parts.

- Learn the fundamentals of what goes on inside a mold during the injection molding.
- Get a basic understanding of how a molding machine functions and its various components.
- Analyze real-life molded parts and learn the cause-and-effect behaviors of design and process.

Topics include:

- The Molding Machine
- Temperature & Fill Speed
- Wall Thickness & Injection Pressure
- Switch-Over from Filling to Packing
- Packing Behavior
- Cooling Behavior
- Warpage Due to Differential Shrinkage
- Warpage Due to Differential Cooling
- Warpage Due to Fiber Orientation
- Cooling Behavior

Lesson 2: Polymer Materials: Material Behavior

Part and mold designers need to understand not only design principles but also the fundamentals of plastics. The choice of a plastic or polymer strongly impacts the outcome of a molded part. A general understanding of the basic properties and structures of polymers will help part and mold designers and processors make choices and decisions that will optimize the final part.

- Learn the fundamentals of polymer material characteristics and basic polymer categorizations.
- Get a basic understanding of how different material behaviors will impact the molding process and the final molded part.
- Analyze real-life molded parts and learn the cause-and-effect behaviors of design and process.

Topics include:

- Polymer Materials
- Viscosity
- PVT
- Thermal Properties
- Mechanical Properties
- Deformation Behavior

Kruse Training Molding Defects Seminar

2-Day Seminar Outline

Welcome to Kruse Training's Molding Defects Seminar. Molding defects in injection-molded parts range from minor surface defects to major structural issues impacting overall quality. This seminar covers common molding defects, how and why they occur, and ways to minimize or prevent them.

DAY TWO

Lesson 1: Short Shots (Processing-Related)

Injection molding is a complex scientific manufacturing process. By following a systematic set-up process, molders aim to minimize or eliminate common molding defects that affect part quality.

Generally, molding defects are either part design, mold design, or process related. This lesson focuses on processing-related short shots, where portions of a part remain unfilled. Generally, molding defects are either part design, mold design, or process related. This lesson focuses on processing-related short shots, where portions of a part remain unfilled. The next lesson will cover part and mold design-related short shots.

Topics include:

- Potential reasons for short shots.
- How to Run various sensitivity analyses to evaluate process parameters affecting short-shot conditions.
- How melt and mold temperature, cavity fill time, injection velocity profiling, switch-over set-point from filling to packing, and the initial packing pressure affect short shots.

Lesson 2: Short Shots (Part Design-Related)

This lesson focuses on part design-related short shots, where portions of a part remain unfilled. The previous lesson focused on process-related short shots.

Topics include:

- Part design factors can lead to short-shot conditions.
- How to adjust rib size to eliminate race tracking, air entrapment, and other short-shot risks.
- How gate size and placement can affect injection pressures and short-shot risks.
- Why overall nominal wall thickness distribution of part features should be optimized.
- How adjusting process parameters can reduce the risk of short-shot conditions.

Lesson 3: Entrapments

This lesson focuses on gas and air entrapments, a common molding defect that can create numerous issues, including burn marks, short shots, and blemishes in molded parts. Optimizing part geometry and mold design is essential for avoiding entrapments.

Topics include:

- The causes and possible solutions for entrapment during the filling process.
- How race tracking and hesitations can cause entrapments.
- How internal venting may be a solution for entrapment conditions.
- How wall thickness variations can cause entrapments.

Lesson 4: Burn Marks

The injection molding process is a complex 3-dimensional polymer flow and transient heat exchange manufacturing process. Part design, mold design, and the molding process are all intrinsic to a successful outcome. This lesson focuses on the causes of burn marks and effective methods for avoiding this common molding defect.

Topics include:

- The causes of burn marks and preventative measures to avoid them.
- How part design, mold design, and processing changes can impact entrapments and burn marks.
- How air zone pressure and temperature evaluations can show where air entrapments may result in burn marks.
- The placement and sizing of vents are vital to eliminating short shots, entrapments, and burn marks.

Lesson 5: Flash

As injection molded parts become more complex, so do mold designs. Intricate mold designs may require multiple inserts, side cams, or pins with various parting lines. These intersecting mold components increase the risk of parting line flash, the focus of this molding defects lesson.

Topics include:

- How flash occurs in injection molded parts.
- The factors that can influence flash development.
- How to analyze flash development using different overflow gaps of different sizes on a sample part.
- How process parameters can impact parting line flash.

Lesson 6: Meld and Weld Lines (Processing-Related)

When plastic flows into an injection mold, the internal flow pattern depends on the part design, mold design, and processing factors. Part design features may cause a material flow to split and rejoin as it flows around design features in a mold. In this lesson, we will focus on the meld and weld lines that result from polymer material flow rejoining within a mold and how different process set-ups impact meld and weld line formation.

Topics include:

- How meld and weld lines occur.
- The differences between meld and weld lines.
- What different process set-ups impact meld and weld line formation?
- How to use a bulk temperature evaluation to understand the predominant flow path of a hot polymer material.

Lesson 7: Meld and Weld Lines (Part Design-Related)

When plastic flows into an injection mold, the internal flow pattern depends on the part design, mold design, and processing factors. Part design features may cause a material flow to split and rejoin as it flows around design features in a mold. This lesson will focus on the meld and weld line formation relative to part wall thickness and gating.

Topics include:

- How different wall thicknesses in a part design can affect meld and weld line development.
- How single or dual hot runners compare in meld and weld line development.
- How polymer temperature at a weld line location plays a significant role in its appearance and strength.

Lesson 8: Sink Mark and Voids, Part 1

In the injection molding business, producing high-quality parts without molding defects is essential. In this lesson, we will review two common and related defects, sink marks and voids, resulting from excessive polymer material shrinkage.

Topics include:

- What sink marks and voids are, and why do they occur?
- Molding process-related challenges that can cause and prevent sinks and voids.
- The process sets up parameters that can create sinks and voids.
- What uniform wall thickness and gate locations affect sink marks and voids?

Lesson 8: Sink Mark and Voids, Part 2

We will further evaluate design and process-related sink marks and voids in part two.

Topic includes:

- The various part design-related challenges impact the fill-to-shrinkage behavior of a molded part.
- The importance of evaluating and eliminating sink marks and voids before finalizing a part design.
- That injection pressure requirements may be dictated by sprue, runner, and gate sizes but not necessarily by the wall thicknesses inside the molded part.
- The way that polymer material cools inside the mold cavity may result in shrinkage and possible sink marks and void development.

Lesson 10: Gate Blush and Halo

Polymer material flows from a molding machine through a runner system and gates into a mold cavity. If the gates are too small for the project, a molding defect can develop near the gate on the final part. This lesson will focus on the gate blush molding defect, which sometimes appears as a dull spot or a halo on the molded part.

Topics include:

- The three primary reasons for gate blush.
- How gate types, sizes, and locations determine blush and halo effects.
- How cavity-filling velocity affects flow-induced shear stress and flow-induced residual shear stress.
- How to compare residual shear stresses during the filling and packing phases.

Lesson 11: Jetting

Injection molding polymer through small gates creates complex three-dimensional fluid flow behaviors. Generally, molten material is injected quickly into a mold cavity to avoid fill-related molding challenges. Jetting is a molding defect that can develop because of polymer viscosity, viscoelastic behaviors, and part and gate design.

Topics include:

- The causes and preventative measures for jetting development.
- How could gate location be the cause or the solution for jetting and air entrapment?
- How injection flow velocity impacts jetting behavior.

Lesson 12: Core Deflection

Producing high-quality parts without molding defects becomes more complicated as part and mold designs become more complex. Three-dimensional flow and pressure patterns inside a mold cavity can impact the deflection behavior of small steel features in a mold. This lesson will teach us about core deflection, a common molding defect that can cause many other defects if not appropriately managed.

Topics include:

- How core deflection occurs during injection molding.
- When the risk of core deflection is highest.
- How gating and injection pressures impact core deflection.
- How core deflection impacts wall thickness differentials.

DAY THREE

Lesson 13: Dimensional Challenges, Part 1

Producing parts without molding defects is essential in the injection molding industry. Molded parts are designed to specific dimensions, which must be achieved and maintained throughout molding. In the following lessons, we will evaluate several part designs, gate options, materials, and process conditions to evaluate dimensional challenges in molding.

Topics include:

- The critical aspects for molding a dimensionally stable part.
- How gate type, location, and sizing impact flow front temperatures, injection pressure requirements, gate seal time, and volumetric shrinkages.
- The impact of different polymer materials on injection pressure, volumetric shrinkages, and flatness.

Lesson 14: Dimensional Challenges, Part 2

This lesson focuses on the molding process and polymer-specific influences on dimensions.

Topics include:

- The molding process parameters can influence a molded part's dimensional stability.
- The importance of uniform shrinkage behavior to ensure consistent dimensional accuracy.
- How polymer material characteristics affect dimensional flatness.

Lesson 15: Dimensional Challenges, Part 3

This lesson focuses on achieving dimensional requirements when designing and molding round parts and round features.

Topics include:

- The critical factors for achieving dimensional roundness of a molded part.
- How wall thickness and gating can affect volumetric shrinkages, resulting in roundness dimensional differences.
- The importance of polymers and gating on fiber orientations and resulting dimensions.

Lesson 16: Warpage, Part 1: Wall Thickness and Fill-Related Processing Factors

Shrinkage in molded parts is a complex behavior that involves polymer material, part design, tool design, and the molding process setup. In these lessons, we will focus on the factors that can help reduce or eliminate warpages so that the molded parts meet all visual and dimensional requirements.

Topics include:

- The impact of wall thickness on part warpage.
- The necessary fill-related process conditions that can impact part warpage.
- Optimizing part design can minimize shrinkage and warpage.

Lesson 17: Warpage, Part 2: Evaluating Individual Parts within an Assembly

In Part 2 of the warpage lessons, we will focus on the factors that can help reduce or eliminate warpage so that the molded parts meet all visual and dimensional requirements.

Topics include:

- The importance of evaluating warpage behavior for an individual part within an assembly.
- How filling patterns, pressure distribution, and clamp tonnages impact warpage.
- That warpage may be acceptable when individual parts within an assembly all warp in the same direction and to the same degree.

Lesson 18: Warpage, Part 3: Structural Ribs and Fiber Percentage

In Part 3 of the warpage lessons, we will focus on how structural ribs, gate locations, and glass fiber percentages can affect part warpage.

Topics include:

- How rib height affects warpage behavior.
- How the number of gates and gate locations affects warpage behavior.
- How fiber orientation is directly related to part flatness.
- How the percentage of glass fiber can affect warpage.

Lesson 19: Warpage, Part 4: Cycle Time, Wall Thickness and Packing Time

In Part 4 of the warpage lessons, we will focus on how cycle time, wall thickness, and packing time can be adjusted to reduce shrinkage and warpage in molded parts.

Topics include:

- How cycle time can affect warpage behavior.
- The steps to determine a molding machine's cost per hour to produce a part.
- How to adjust wall thickness to meet a molding machine's capacity.
- How extending packing time may reduce warpage.

Lesson 20: Warpage, Part 5: Gate Location and Processing

In Part 5 of the Warpage lessons, we will focus on gate location and the processing effect on Warpage.

Topics include:

- How to make process adjustments within a set cycle time to reduce warpage.
- Why a polymer material's reaction to process changes is an essential indicator of part warpage.
- How pack pressure profiles affect part warpage.

Lesson 21: Warpage, Part 6: Filled and Unfilled Polymer Materials

In Part 6 of the warpage lessons, we will focus on how filled and unfilled polymer materials affect shrinkage and warpage behaviors.

Topics include:

- How to use a simple polymer flow front temperature test to compare material heat energy behaviors inside a cavity.
- What different injection pressure requirements will result in different packing pressure profile set-ups and affect shrinkage and warpage behaviors?
- The temperature ranges and distributions inside cavities differ for different materials and will influence shrinkage and warpage behaviors.
- How unfilled and glass-filled polymers impact shrinkage and part warpage.

Lesson 22: Warpage, Part 7: Dimensional Challenges

In Part 7 of the warpage lessons, we will focus on a part design scenario using a high-precision wheel as our sample part and show various gate options.

Topics include:

- How to evaluate a selected fill time to determine whether it is correct.
- How gating options affect fiber orientation, height direction shrinkages, and roundness of the molded part.
- Why a mold design team may evaluate an unconventional gate in a CAE simulation environment.

Lesson 23: Warpage, Part 8: Gate Location, Process Parameters and Polymers

In Part 8 of the warpage lessons, we will focus on gate location, process parameters, and polymer materials.

Topics include:

- How volumetric shrinkages correlate to linear shrinkages and part warpage behavior.
- The impact of gate location on fiber alignment and resulting shrinkage and warpage behaviors.
- How a polymer material's melt viscosity affects injection pressure.

Lesson 24: Warpage, Part 9: Fiber Percentage and Processing

In Part 9 of the Warpage lessons, we will focus on how the amount of glass fiber added to a material and the molding process can affect Warpage behavior.

Topics include:

- What varying amounts of fiber content in a polymer can affect warpage?
- How part cooling temperature when a mold opens can be used to evaluate cooling and shrinkage behavior.
- Why mold temperature can have a considerable impact on polymer cooling behavior.
- How to control part warpage with the molding process.

Lesson 25: Warpage, Part 10: Assembly: Base Part

In Part 10 of the Warpage lessons, we will focus on how part design influences overall Warpage behavior.

Topics include:

- How to evaluate part design modifications to reduce warpage behavior.
- Gating and process changes may not solve part warpage challenges.
- How to modify part designs to meet warpage tolerance requirements.

Lesson 26: Warpage, Part 11: Assembly: Frame and Dome Parts

In Part 11 of the warpage lessons, we will focus on gating and material selection.

Topics include:

- How different gating strategies can affect molten core, volumetric shrinkages, and warpage.
- How to apply cause-and-effect analysis using gating and material selection.
- If a shrinkage gradient will cause a part to warp.

The "Circle of Knowledge" program is a 3-day in-person seminar based on the Kruse Training online training solution. The seminar is a comprehensive cross-training initiative for part designers, mold designers, and process engineers.

Upon completion of the seminar, participants receive a 3-month single-user account with unlimited access to https://krusetraining.com. During this trial period, users who complete Level 1 lessons with passing quiz scores will receive certification from Kruse Training. After the trial period, participants may sign up for continued access to the online program for more advanced training as it becomes available.

For additional information, seminar pricing, or other questions, contact Torsten Kruse at 239-351-7428 or torsten_kruse@krusetraining.com.