Sharing The Knowledge
Module 10

Design and Performance

M–PLA–0117–STK–MOD 10–12/89–00–1

GE Plastics

Module 10

Design and Performance

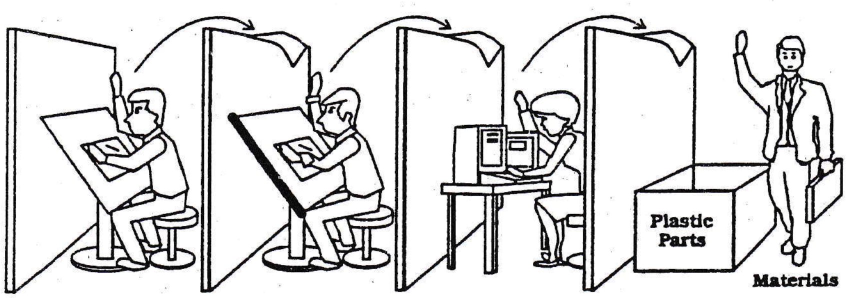
- Development Process
- Design Considerations
  - Moldability
  - Part Geometry
  - Tooling Considerations

10

Participant's Notes:

GE Plastics

Traditional Development Process



Isolating each area of the product development cycle results in more development time, increased costs, and compromised part performance.

10

Participant's Notes:

Introduction

Design & Performance

Product development is a complex process requiring careful consideration of numerous issues such as functionality, form, material, manufacturability, and cost. Plastics in general, and injection molding specifically, have their own unique design requirements. It is important for the design engineer to understand these material- and process- specific requirements at the outset of the product development cycle. Although each of the important areas of product development are interdependent, they often are considered separately.

STK 1001

Objectives:

At the end of this module, participant should be able to:

- Describe the advantages of simultaneous engineering compared to the traditional development process.
- Describe the relationships between moldability, tooling and part geometry that dictate high performance part design.
- Describe strategies to minimize molded-in stress.
- Describe strategies to optimize part geometry.
- Describe the tooling considerations for ensuring high performance part design.

Development Process

The traditional compartmentalized approach to applications is inefficient.

Traditional Development Process

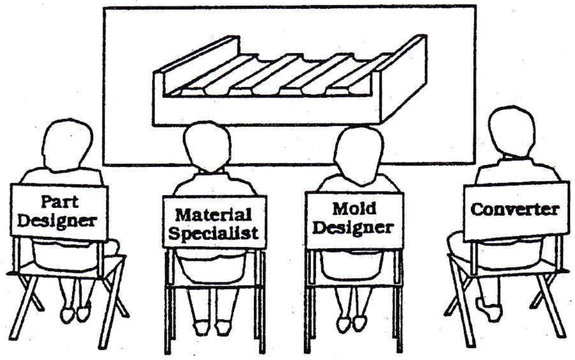
It is all too typical for the design engineer to design parts for functional performance then, in essence, throw the drawings over the wall to the mold maker. After solving any tooling problems, the mold maker then throws the drawings over the wall to the converter. It's then up to the converter to figure out how to manufacture the thing, and to select a material that meets <u>all</u> its necessary requirements: performance... moldability.... cost! It is an inefficient approach that usually gives way to compromised product quality and increased product cost.

STK 1002

STK 1003

GE Plastics

Simultaneous Engineering



Simultaneous engineering calls for the integration of each aspect of a part's development at its inception.

10

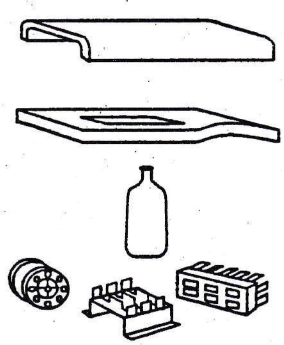
Participant's Notes:

STK 1004

GE Plastics

Conversion Processes

- Extrusion
- Sheet Fabrication
- Blow Molding
- Injection Molding



A key decision facing the part designer is the choice of process.

10

Participant's Notes:

Simultaneous Engineering

Integration of design functions is more effective.

Simultaneous engineering calls for the integration of each area of the product development cycle by uniting the design engineer, the mold maker, the converter, and the material supplier at the product's inception. Important interdependent issues can then be cross-examined and any potential problems addressed before they become major obstacles. Ideally, the result is a higher quality part at a lower cost.

STK 1003

Conversion Processes

An important decision the designer must make is the choice of a conversion process. The choice of process will, of course, impact the tool design, but it may also call for a new material selection or an alteration in the original part design. If, for example, it has been decided that the part be extruded, then a material grade with good melt strength should be considered.

A key decision facing the part designer is the choice of process.

Each process is chosen for its own advantages and hindered by its own limitations, and each calls for its own set of design and material considerations. Extrusion is a continuous process capable of molding long, simple shapes without incurring high tool and die costs but is limited in terms of part complexity. The sheet fabrication process, compression molding, is capable of producing parts that have large variations in wall thickness with little or no warpage, distortion, or voids. Still compression molding tends to be labor intensive.

Pressure and vacuum forming, both sheet fabrication processes, are low pressure processes that are easily automated and relatively inexpensive in terms of tooling. Yet they are very limited in terms of part complexity. Blow molding is the best way to produce hollow parts, and it also molds parts with good stiffness-to-weight ratios while incurring relatively low tooling costs. Like the others, it can be limiting as it requires longer cycle times and expensive start-up costs.

STK 1004

STK 1005

GE Plastics

Injection Molding Thermoplastics

Design Freedom

- Consolidated Parts
- Snap-Fits
- Diverse Shapes
- Close Tolerance Parts
- Broad Material Selection

Productivity

- Fast Cycle Times
- Efficient Material Usage
- Eliminated Secondary Operations
- Reduced Labor
- Automation
- Reduced Secondary Operations

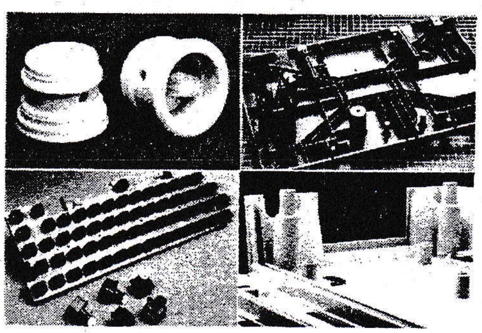
10

Participant's Notes:

STK 1006

GE Plastics

Complex Thermoplastic Parts



High performance thermoplastic parts can be produced by understanding the relationships that dictate good design.

10

Participant's Notes:

Injection Molding Thermoplastics

Injection molding offers the manufacturer design freedom and productivity advantages.

Injection molding is often the most productive processing method for long runs. It is an automated process that allows for faster cycle times, efficient material usage, and the elimination of certain secondary operations. It is a sophisticated process that requires good tooling and part design for optimal productivity. Injection molding offers the designer increased freedom by allowing for the consolidation of parts and the incorporation of snap-fits. This module concentrates on crucial aspects of the injection molding process.

STK 1005

Complex Thermoplastic Parts

High performance thermoplastic parts can be produced by understanding the relationships that dictate good design.

Injection molding is the only process that can deliver high tolerance repeatability with the elimination of scrap and numerous secondary operations. As a process it offers greater design flexibility, broader material selection, and increased manufacturing productivity. And by understanding the relationship of design, materials, and processing and by adhering to the rules and limitations imposed by each, the designer can successfully create complex, high performance thermoplastic parts at lower cost.

STK 1006

STK 1007

GE Plastics

High Performance Part Design

Moldability

Optimize

Tooling Considerations

Part Geometry

10

Participant's Notes:

STK 1008

GE Plastics

Stress

- Stress is an applied load per unit area.
- Strength is the maximum stress a material can withstand.

10

Participant's Notes:

High Performance Part Design: Moldability

Optimize moldability.

We've broken down the important design considerations for injection molding into three categories: moldability, part geometry, and tooling considerations. Moldability refers to the ease with which a material is processed and includes filling, packing, cooling and ejection of the part. Ultimate part quality is often a direct outcome of the material's flow behavior in the mold. **STK 1007**

Stress

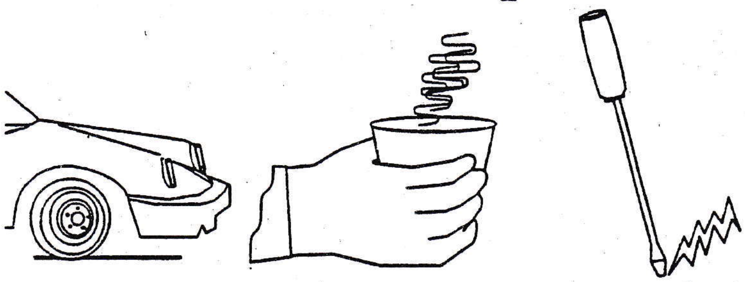
Stress and strength are key terms.

Minimizing stress is a critical consideration for the designer. Stress refers to any applied load to the material such as pulling, squeezing, bending, impact, heat, or electricity. Strength is the maximum stress a material can withstand. To a designer, strength is the maximum stress a material can withstand and still recover. Tensile strength, for example, measures the maximum pulling load a material can support before yielding or breaking. **STK 1008**

STK 1009

GE Plastics

Part Performance Requirements



Strength | Heat Insulation | Electrical Insulation

Each part calls for its own specific set of performance requirements.

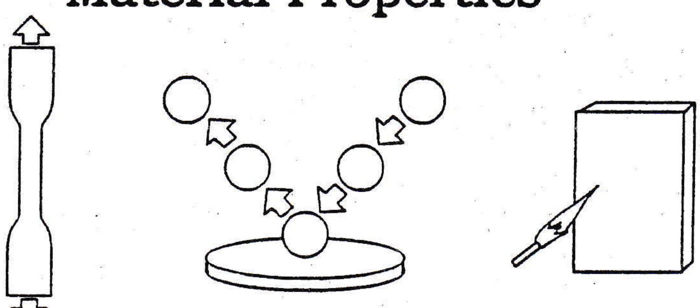
10

Participant's Notes:

STK 1010

GE Plastics

Material Properties



Stiffness & Strength | Impact | Flammability

The property values reported on product data sheets indicate relative performance of materials.

10

Participant's Notes:

Part Performance Requirements

A part is designed to be functional in terms of both structure and material. Material can be chosen for aesthetic purposes, but is usually selected for functional reasons.

Each part calls for its own specific set of performance requirements.

Each part calls for its own specific set of performance requirements. A car bumper, for example, must be strong and able to withstand high impact. A coffee cup must provide substantial heat insulation to protect the user from hot liquids. And it is essential that the handle on a screw driver provide electrical insulation from electrical shock. Performance requirements of the part should dictate material selection. **STK 1009**

Material Properties

Numerous material properties are used to describe material performance. For our discussion, the designer has chosen to use an engineering thermoplastic to manufacture their product design. We are therefore interested in the properties that are used to compare and describe thermoplastic materials such as tensile strength, impact performance, and flame resistance.

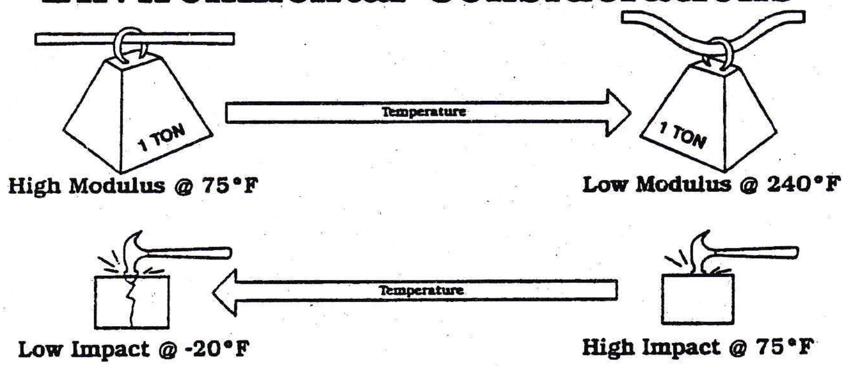
The property value on product data sheets indicates the relative performance of materials.

Each resin is subjected to an assortment of standard tests that simulate actual application requirements. Engineers then report the test results on the resin's product data sheet for material comparison and selection. The product data sheet can only provide a "snap-shot" of how a material will respond in application. The test results can only measure how well a material will perform when tested under one distinct set of conditions and variables. **STK 1010**

STK 1011

GE Plastics

Environmental Considerations



A material should be tested under anticipated environmental conditions to determine its suitability.

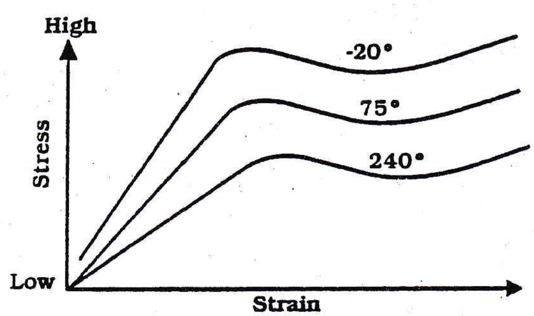
10

Participant's Notes:

STK 1012

GE Plastics

Engineering Design Database (EDD)



- Tensile Stress - Strain
- Tensile Creep
- Tensile Fatigue
- Rheology

GE's Engineering Design Database contains graphical data that describes the material through its useful performance ranges so the designer can predict material responses under specific conditions.

10

Participant's Notes:

A material should be tested under anticipated environmental conditions.

Environmental Considerations

GE's Engineering Design Database allows the material designer to check the suitability of a material selection under anticipated environmental conditions. Environmental variables such as temperature can greatly affect a material's properties. Modulus, for example, tends to increase as the temperature goes down. Impact resistance, however, tends to decrease as the temperature drops. For example, if an application calls for a material to maintain a high modulus at a temperature @ 240°F, a material must be chosen that meets the modulus requirement at that temperature. Or if the application calls for a material to maintain the same impact resistance as the temperature is lowered from room temperature to –20°F, then a material must be chosen that meets the impact requirement at that temperature. **STK 1011**

EDD contains graphical data that describes a material through its useful performance ranges.

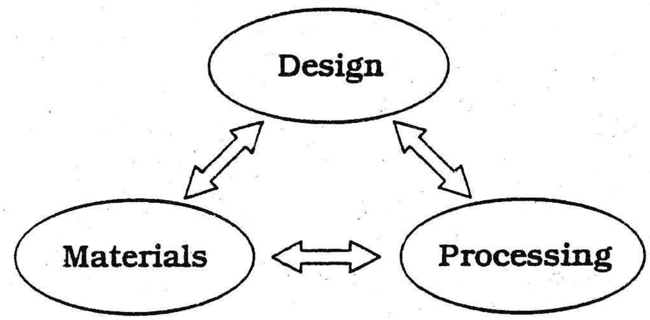
Engineering Design Database (EDD)

GE's Engineering Design Database (EDD), contains graphical data that describes a material through its useful performance ranges. By offering material performance information over a range of stresses, strain rates, temperatures, and times, the EDD can supply application-specific data that more accurately predicts actual part performance than standard data sheets. **STK 1012**

STK 1013

GE Plastics

High Performance Parts



Plastic part performance is maximized & cost minimized by integrating the three areas of plastics technology.

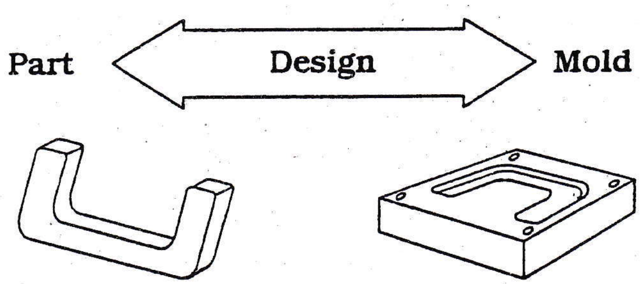
10

Participant's Notes:

STK 1014

GE Plastics

Design



When discussing design, both the part and the mold must be considered.

10

Participant's Notes:

Integration of design, materials, and processing results in efficient part production.

High Performance Parts

Design, processing, and materials are the three major areas of plastics technology. Only by integrating information from these three areas of expertise can a product designer meet the design challenge to maximize part performance and minimize part cost. **STK 1013**

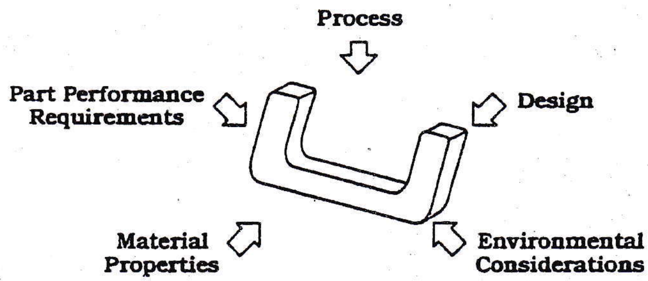
When discussing design, both the part and the mold must be considered.

Design

Once the part is designed it must be molded. Moldability must be considered as part of the design process. STK 1014

GE Plastics

Application Development



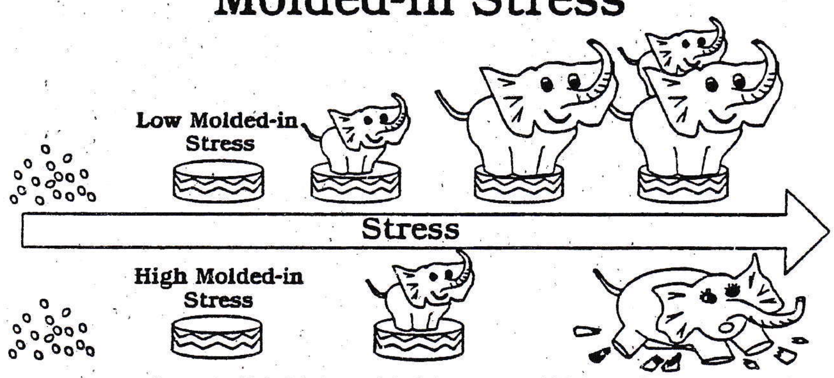
The designer's challenge is to select a process and a material, create a design which meets application requirements and survives the environment at the lowest cost.

10

Participant's Notes:____

GE Plastics

Molded-in Stress



A part with high molded-in stress will have less available strength & therefore fail at a lower in-use load.

10

Participant's Notes:____

Many factors need to be considered when designing parts.

Application Development

Ultimately, it is the responsibility of the designer to design a part that is functional. But it is the challenge of the designer to select a process and material that meet the part's performance and environmental requirements at the lowest possible cost. **STK 1015**

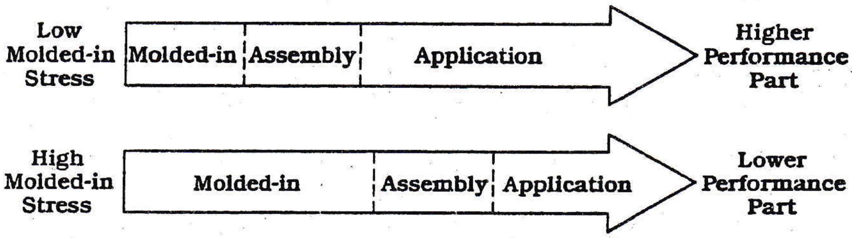
A part with high mold-in stress will have less available strength & therefore fail at a lower in-use load.

Molded-in Stress

A part designer determines the maximum amount of stress the part may need to withstand in application, then chooses a material able to withstand the application requirement. But the degree of stress any material can withstand is finite. Molded-in stress refers to stress that has been "built into" the part during processing. Design, tooling, and molding steps can be taken to reduce molded-in stress, though some stress is always inevitable. Still a part with high molded-in stress will have less available strength in application and therefore may fail at a lower load. **STK 1016**

GE Plastics

Application Strength



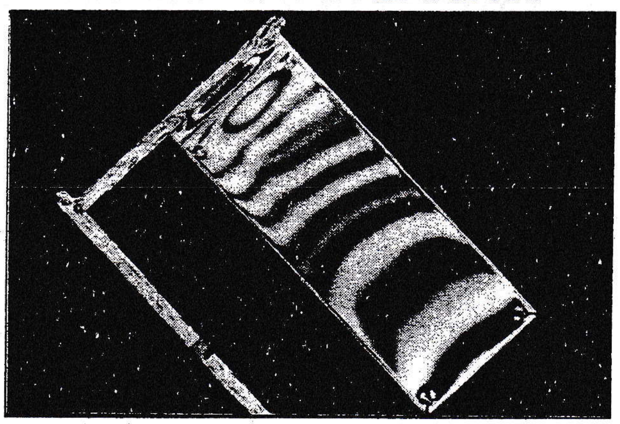
Minimizing molded-in stress will maximize the available strength for application.

10

Participant's Notes:

GE Plastics

Actual Part with Stress



10

Participant's Notes:

Application Strength

Minimizing molded-in stress will maximize the available strength for application.

The designer must consider the total required application strength. These include molded-in, assembly, and application stresses. Minimizing assembly stress and molded-in stress allows for maximum available strength in the application. High molded-in stress may contribute to premature part failure. **STK 1017**

Actual Part with Stress

Light patterns indicate areas of stress.

The concept of stress is often difficult to understand because it is impossible to see with the naked eye. Here is an actual photograph of stress taken with polarized light. Light hits the object at many angles then scatters in all directions. A polarizer can filter out the angles, allowing just one to get through. This one-directional beam of light aimed at a transparent part containing no stress will pass through without bending and no pattern would arise. Uneven stress in a part causes the beam to bend and a colorful pattern results. The greater the stress the more patterns will be visible. **STK 1018**

GE Plastics

Molded-in Stress

- Over-packing
- Uneven Shrinkage

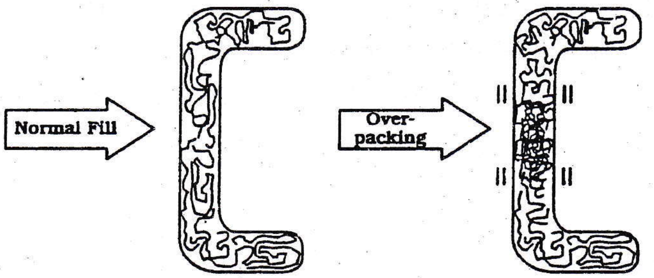
10

Participant's Notes:

GE Plastics

Over-packing

Over-packing forces more material into the cavity than is required.



The polymer molecules are frozen into this compressed state upon cooling causing molded-in stress.

10

Participant's Notes:

Over-packing.

Molded-in Stress: Over-packing

Molded-in stress is caused by either over-packing, uneven shrinkage or a combination of the two. **STK 1019**

Over-packing causes molded-in stress.

Over-packing

When too much material is forced into the cavity, the result is over-packing. When a mold is over-packed, uneven densities result. These uneven densities lead to uneven shrinkage which can cause molded-in stress. Instead of resting in a relaxed configuration, the molecules are "cramped" and therefore under a higher amount of stress. Cooling the part merely freezes the molecules in this compressed state causing molded-in stress. **STK 1020**

GE Plastics

Molded-in Stress

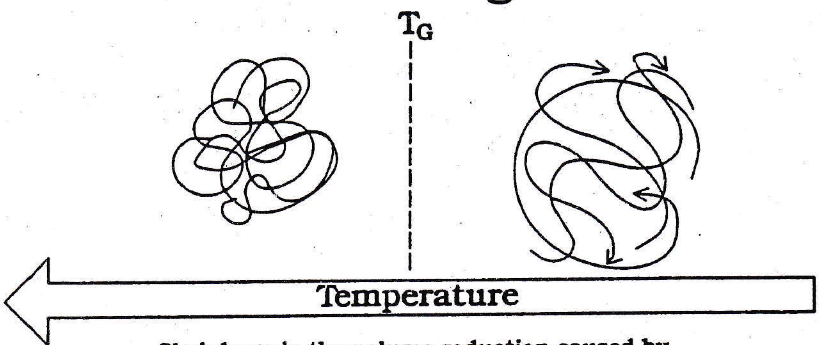
- Over-packing
- Uneven Shrinkage

10

Participant's Notes:

GE Plastics

Shrinkage



Shrinkage is the volume reduction caused by the decrease in the amount of space between the polymer molecules upon cooling.

10

Participant's Notes:

Molded-in Stress: Uneven Shrinkage

Uneven shrinkage.

Uneven shrinkage is a prevalent cause of molded-in stress. It may be caused by design constraints and may result in structural defects.
STK 1021

Shrinkage

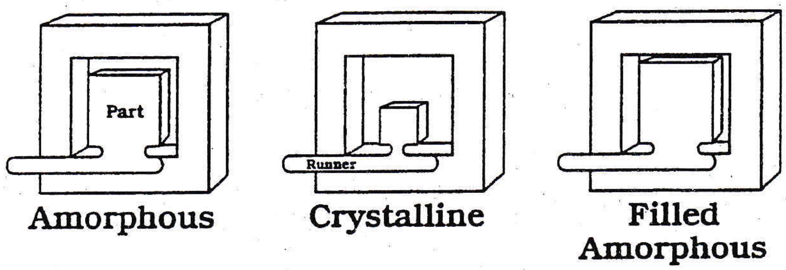
Shrinkage is the volume reduction caused by the decrease in the amount of space between the polymer molecules upon cooling.

Shrinkage is the material volume reduction that occurs upon cooling. During processing, heat is added to a polymer, creating enough space between the molecular chains to allow them to slide past each other and flow. This space causes an increase in the volume of material in the melt state. Upon cooling, the molecular chains slow down and move back together again, causing a reduction in the volume of the material. The degree to which the material shrinks depends on the material and is an important design, tooling, and processing consideration. **STK 1022**

STK 1023

GE Plastics

Material Consideration



Materials shrink differently than others and therefore require different part & tool design considerations.

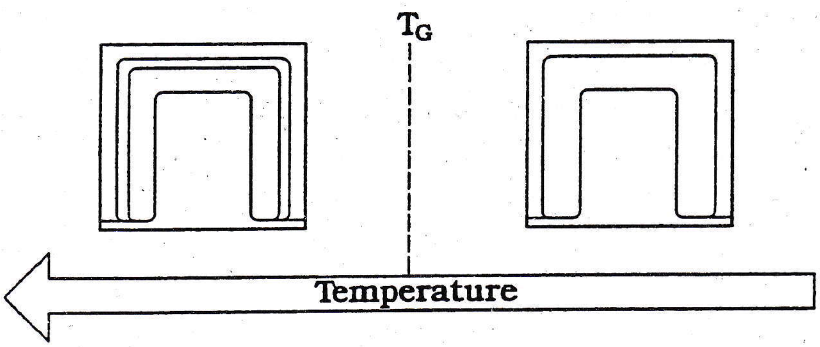
10

Participant's Notes:

STK 1024

GE Plastics

Geometric Consideration



A part with uniform wall thickness will fill and cool evenly.

10

Participant's Notes:

Material Consideration

Materials shrink differently & require different part and tool design considerations.

Every material has its own specific shrink characteristics which are recorded on the product data sheet. Amorphous materials tend to shrink less than crystalline materials. As a crystalline polymer cools, it builds areas of crystallinity. The molecules in these areas of crystallinity lie more closely together than the molecules in an amorphous polymer. A crystalline material tends to shrink more than an amorphous material. Although amorphous and crystalline materials exhibit nearly isotropic shrinkage, a glass fiber reinforced material exhibits anisotropic shrinkage. Glass filled polymers will shrink more in the cross flow and less in the flow direction. It is important that the tool accommodate the selected material's shrink behavior. **STK 1023**

Geometric Consideration

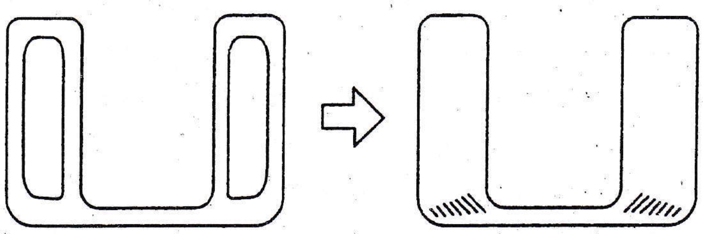
Parts with even wall thickness will fill and cool evenly.

A part with uneven wall thickness does not cool evenly. This may result in uneven shrinkage which causes stress. STK 1024

STK 1025

GE Plastics

Stress Due to Uneven Shrinkage



A part with non-uniform wall thickness will cool unevenly, resulting in high molded-in stress.

10

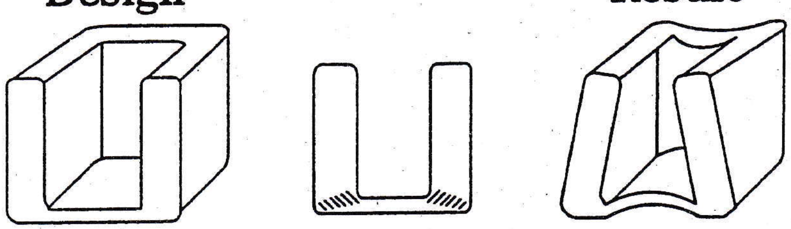
Participant's Notes:

STK 1026

GE Plastics

Warpage Due to Uneven Shrinkage

Design **Result**



Concentrated stress at the junction of high & low shrinkage areas may cause a part to warp.

10

Participant's Notes:

Molded-in stress is increased by uneven shrinkage due to non-uniform walls.

Stress Due to Uneven Shrinkage

When a part contains varying wall thickness, thinner sections will cool more quickly than thicker sections. Once the thinner section has cooled and shrunk, it solidifies thus hindering the adjacent thicker section from shrinking freely. The result is a high degree of molded-in stress where the thin and thick sections meet. **STK 1025**

Stress may cause warpage.

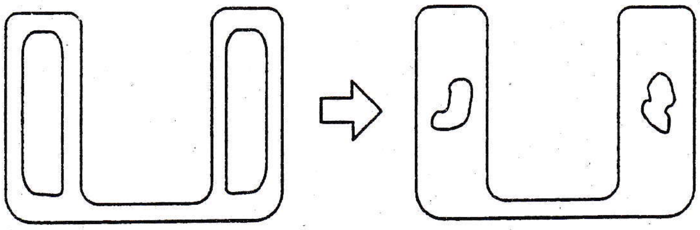
Warpage Due to Uneven Shrinkage

Stress at the juncture of high and low shrinkage areas may cause a part to warp. **STK 1026**

STK 1027

GE Plastics

Voids Due to Uneven Shrinkage



The already cooled section will not yield to the shrinking action of the cooling interior mass causing voids in the thick portion of the part.

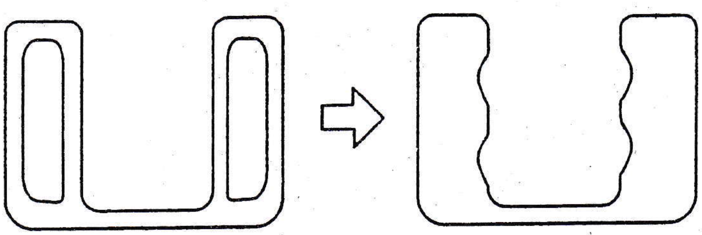
10

Participant's Notes: ______

STK 1028

GE Plastics

Sink Marks Due to Uneven Shrinkage



Sink marks result from a wall yielding to the still shrinking interior mass.

10

Participant's Notes: ______

Voids can result from non-uniform wall thickness.

Voids Due to Uneven Shrinkage

Uneven shrinkage may result in a void in the thick section. When the already cooled section will not yield to the still cooling thicker section, the shrinking action in the thick section may create a void. **STK 1027**

Sink marks result from an outside wall yielding to the still shrinking interior mass.

Sink Marks Due to Uneven Shrinkage

The shrinking action in the still cooling thick section may compensate by creating a sink mark on the surface of the part. A sink mark is nothing more than a void on the surface of the part. Both are the result of continued shrinkage in one section of a part being hindered by the solidification of another already cooled section. **STK 1028**

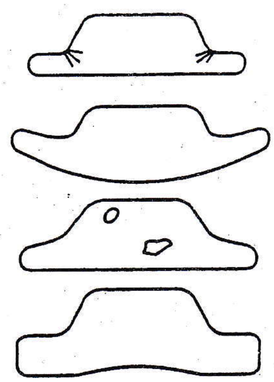
STK 1029

GE Plastics

Uniform Wall Thickness

Minimizes:

- Molded-in Stress
- Warpage
- Voids
- Sink Marks



10

Participant's Notes: ____________________

STK 1030

GE Plastics

High Performance Part Design

Moldability

Optimize

Tooling Considerations ⟺ Part Geometry

10

Participant's Notes: ____________________

Uniform wall thickness minimizes molded-in stress, warpage, voids, and sink marks.

Uniform Wall Thickness

Uniform wall thickness promotes even cooling and therefore even shrinkage thus reducing such potential defects as molded-in stress, warpage, voids, or sink marks. **STK 1029**

Optimize part geometry.

High Performance Part Design: Part Geometry

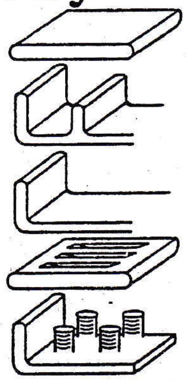
For optimum moldability it is important to strive for uniform wall thickness. This is perhaps the most fundamental rule when designing the geometry of a part. Design features and tooling considerations create the need for variations in thickness. **STK 1030**

STK 1031

GE Plastics

Part Geometry

- Walls
- Reinforced Structures
- Corners
- Holes
- Hollow Bosses



High Performance parts incorporate uniform wall thickness, gradual transitions, and corner radii.

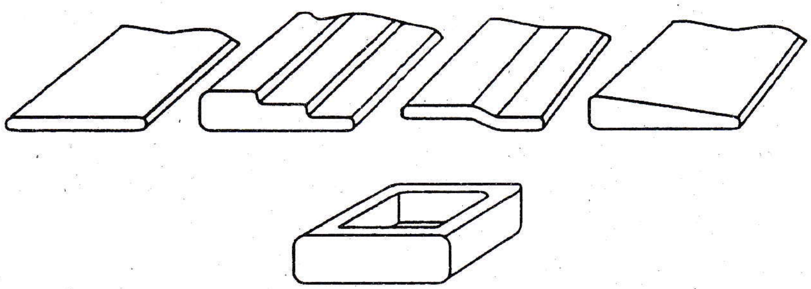
10

Participant's Notes:

STK 1032

GE Plastics

Walls



The wall refers to the thickness of the shell which provides the basic shape of the part.

10

Participant's Notes:

Part Geometry

Part geometry addresses several structural considerations.

Part geometry addresses such structural considerations as walls, reinforcement structures, corners, holes, and hollow bosses. We will discuss each individually, and explain how the designer can create complex, functional parts and still strive for uniform wall thickness using gradual transitions and rounded corners. **STK 1031**

Walls

The wall refers to the shell of the part.

The wall refers to the shell which provides the basic shape of the part. All other structures can be examined in relation to the part wall. Reinforcement structures, holes, and hollow bosses are just projections and depressions of the original wall, while corners are actually extensions of the part wall. **STK 1032**

STK 1033

GE Plastics

Efficient Wall Design

The Designer Should Strive for Minimum Wall Thickness to Increase Productivity:

- **Reduce Material Consumption**
- **Reduce Cycle Time**

While Still Maintaining Part Criteria:

- **Appropriate Flow Length**
- **Sufficient Structural Stiffness & Strength**
- **Required Thickness for Flammability Rating**
- **Uniformity**

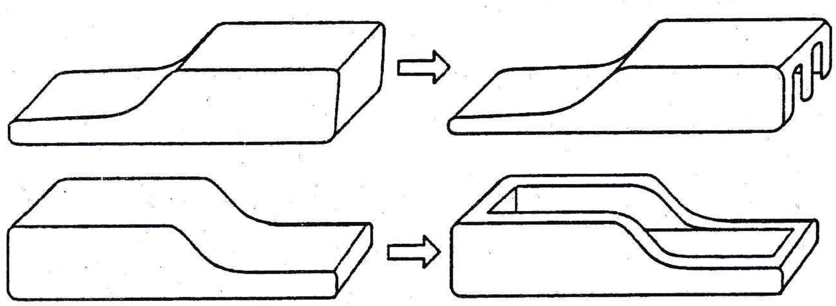
10

Participant's Notes:

STK 1034

GE Plastics

Wall Design



When non-uniform walls are present, the part should be designed with coring and ribbing to create uniform wall thickness while maintaining strength & stiffness.

10

Participant's Notes:

Use minimum walls while maintaining part performance.

Efficient Wall Design

Wall thickness is a primary design consideration, yet it is entirely dependent on the material selection. Minimum wall thickness tends to increase productivity. Efficient wall design calls for the thinnest possible wall that still meets the performance requirements. Thin walls reduce cycle time and material usage. Wall thickness is limited by the flow length of the selected resin. The wall thickness must also meet the structural and flammability requirements. And, of course, the designer must strive for uniform thickness throughout.

STK 1033

Use coring to achieve uniform walls in non-uniform parts.

Wall Design

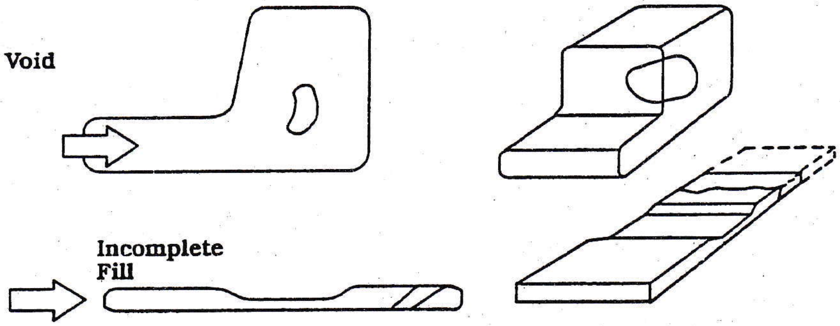
Functionality may call for the wall to be thicker in certain sections. Thick walls increase production costs and are a source of potential part defects, such as voids and sink marks, when used in conjunction with thinner sections. Therefore even the slightest necessary variation in wall thickness should be handled gradually. The designer should strive for gradual transitions and the tool designer should allow the material to flow from the thicker section to the thinner section to minimize the chance of voids or incomplete fills. But when the part calls for thickness or additional strength and stiffness in one section, it should be designed with coring or ribbing to create structural reinforcement, and still maintain uniform wall thickness.

STK 1034

STK 1035

GE Plastics

Voids & Incomplete Fills



The melt should flow from thick to thin sections to minimize voids and incomplete fills due to excessive drops in pressure.

10

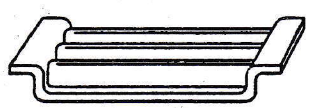
Participant's Notes:

STK 1036

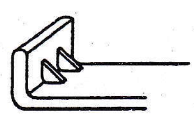
GE Plastics

Reinforcement Structures

- Ribbing



- Gusset



Structural requirements can be met with thin uniform walls by use of ribbing and gussets for reinforcement.

10

Participant's Notes:

Voids & Incomplete Fills

The melt should flow from thick to thin sections.

Thermoplastic material is injected into a cavity under pressure. While over-packing is the result of too much pressure, voids and incomplete fills can result from insufficient pressure. Though an incomplete fill can result from insufficient injection pressure, it more often occurs when material is forced to flow from a thin section to a thick section. Typically there is a drop in pressure as the flow moves from a thin section to a thicker section. As the pressure drops, the material sort of runs out of steam. Insufficient material is injected into a section of the part and a void develops upon cooling. And if the drop is excessive enough, the material may not even fill the entire cavity resulting in an incomplete fill. **STK 1035**

Reinforcement Structures

Ribs and gussets may be used in lieu of increasing wall thickness.

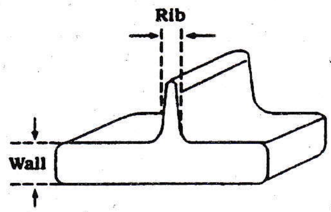
Reinforcing features increase structural strength and rigidity with minimal increases in material usage and cycle time. Properly designed and located ribs increase the stiffness and load carrying capacity of the part. They can also help to control melt flow through the cavity. Gussets help to reinforce the part wall and structural supports such as bosses without increasing wall thickness. Still there are some general rules that govern the design of ribs and gussets.

STK 1036

STK 1037

GE Plastics

Reinforcement Thickness



Rib thickness should be less than the wall thickness.

10

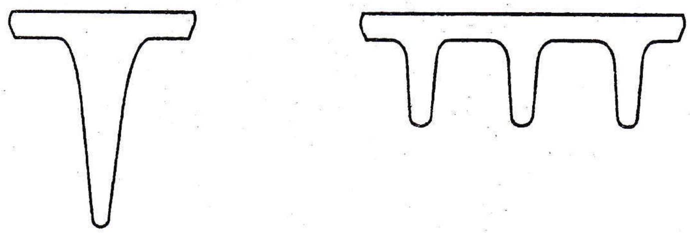
Participant's Notes:

STK 1038

GE Plastics

Ribbing Guidelines

To increase stiffness, increase the number of ribs, or gusset plates.



For a given stiffness, it is better to increase the number of ribs, not the height.

10

Participant's Notes:

Rib thickness should be thinner than the wall.

Reinforcement Thickness

Rib thickness should be less than the wall thickness. The actual ratio is dependent on the material and the requirements of the design. Rib thickness recommendations can vary from 40% to 80%. Excessive rib thickness may cause sink marks and voids. The height of the rib should be limited to 2 1/2 times the wall thickness and include draft to allow for better ease of ejection. **STK 1037**

To increase stiffness, increase the number of ribs, or gusset plates.

Ribbing Guidelines

To increase the stiffness of the part, do not increase the thickness of the rib, as it may cause sink marks or voids. And do not increase the height of the rib for added stiffness, as it will become very difficult to fill. For additional stiffness, increase the number of ribs. It is important to allow enough distance, at least 2 times the wall thickness, between each rib. Also, it helps to match ribs on each side of the part to avoid warpage. **STK 1038**

STK 1039

GE Plastics

Optimum Rib Design

For Maximum Stiffness & Strength with Limited Cosmetic Requirements:

- **Thickness** - 40-80% of Wall Thickness
- **Height** - No More than 3 Times Wall Thickness
- **Location** - At Least 2x Wall Thickness Between Reinforcements

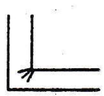
10

Participant's Notes:

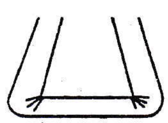
STK 1040

GE Plastics

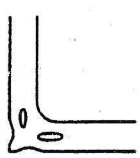
Corner Design



Sharp Corners Concentrate Stresses in a Localized Area - Notch Sensitivity



A Rounded Outside Corner / Sharp Inside Corner Creates Uneven Shrinkage - Warpage



A Rounded Inside Corner / Sharp Outside Corner Creates Uneven Shrinkage - Sink Marks & Voids

10

Participant's Notes:

Optimum Rib Design

For optimum rib design, thickness, height, and location must be considered.

To reiterate, the optimum reinforcement design calls for: thickness that is less than the wall thickness; height that is no more than 3 times the wall thickness; and space of at least 2 times the wall thickness between reinforcements. And as we'll see later, most part projections and depressions should be tapered for ease of ejection. **STK 1039**

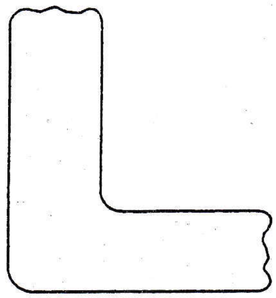
Corner Design

Corner design is critical in good parts.

Corner design is critical in good parts. Sharp corners tend to concentrate stresses in a localized area. It is not enough to round the corners on just the outside or just the inside. A corner that is rounded on the outside only has little effect on the notch sensitivity of the angle, and it can create uneven wall thickness, uneven shrinkage, and possibly warpage. A corner that is rounded on the inside only also creates uneven wall thickness, uneven shrinkage, sink marks and possibly voids. **STK 1040**

GE Plastics

Rounded Corners



- Minimum Stress Concentration
- Uniform Wall Thickness
- Smooth Melt Flow
- Improved Impact Performance

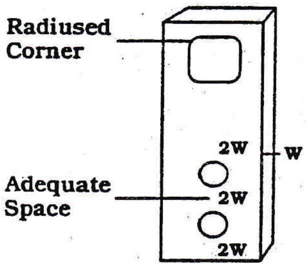
Corners should be radiused as much as possible while maintaining uniform wall thickness.

10

Participant's Notes: ______________________

GE Plastics

Hole Design



Radius inside corners of all holes to eliminate stress concentration.

Minimum space between holes or between holes and wall is two times the wall thickness or two times hole diameter.

10

Participant's Notes: ______________________

The more rounded the corner, the broader the distribution of stress and the higher the impact performance.

Rounded Corners

A corner's stress concentration decreases as the radius of the corner increases: the larger the radius, the lower the stress. The designer should try to create a corner that is as uniform with the wall thickness as possible, with rounded interior and exterior corners. Rounded corners improve melt flow through the cavity and ultimately provide better impact performance by minimizing corner stresses.

STK 1041

Rules for hole design.

Hole Design

Holes in a part should be thought of as nothing more than depressions in the part wall. Square holes containing sharp interior corners are subject to the same kind of stress concentration that affects corner design. A rounded hole distributes the stresses throughout and strengthens the hole. Like the rules of reinforcement design, the space between holes should be at least 2 times the wall thickness or 2 times the diameter of the hole. And since holes are created by a core in the tool, the core should be tapered or drafted for better ease of ejection.

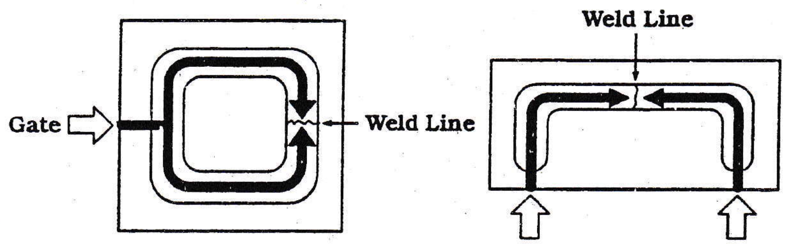
STK 1042

STK 1043

GE Plastics

Weld Line

A weld line results when two flow fronts meet and "knit."



The weld line is an area of weakness and should be located in a low stress portion of the part.

10

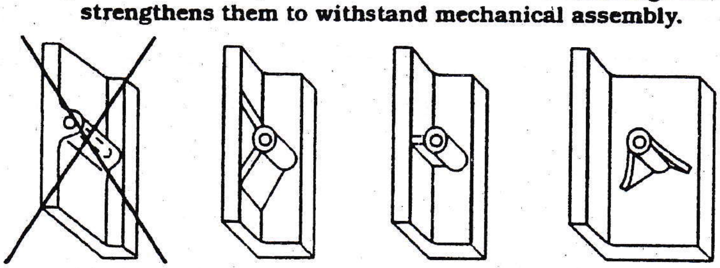
Participant's Notes:________________________________

STK 1044

GE Plastics

Hollow Boss Design

Coring bosses and supporting them with gussets minimizes high molded-in stress or defect due to uneven shrinkage and strengthens them to withstand mechanical assembly.



Fastening bosses should never adjoin vertical walls but should be connected to the walls to eliminate thick sections and sink marking.

10

Participant's Notes:________________________________

Weld Line

The weld line is an area of weakness and should be located in a low stress portion of the part.

As the material flows through the cavity, a core or core pin will divide the flow into two fronts. The point at which the two flow fronts rejoin on the other side of the obstruction is called the weld line. The weld line is an area of weakness and should be located in a low stress area of the part. The location of the weld line depends on gate location.

STK 1043

Hollow Boss Design

Hollow boss design will differ depending on application requirements.

Hollow bosses are typically used to facilitate part assembly by locating mating parts and accepting fasteners. Since the hollow boss must be capable of withstanding mechanical assembly and use, it is important that it be well supported. For maximum strength, the boss's interior diameter should be 2 times the diameter of its hollow core. The boss's outside diameter should be two times the inside diameter.

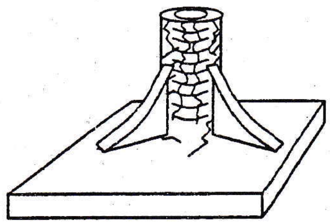
A hollow boss is often tied to the adjacent wall to give it additional strength. The design may result in locally thick sections that can cause sink marks and other defects resulting from uneven shrinkage. Moving the boss inboard and coring between the wall and the boss will avoid heavy sections and the associated problems. Freestanding bosses should be reinforced with gussets for better stability.

STK 1044

GE Plastics

Molded-in Inserts

Excessive
Hoopstress
Causes Cracking



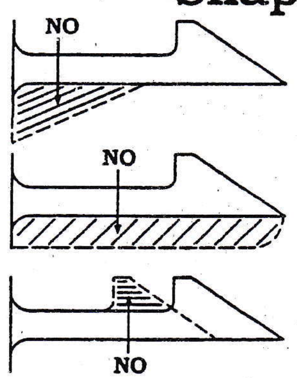
Molded-in metal inserts may crack the boss due to hoopstress caused by shrinkage of the plastic.

10

Participant's Notes:

GE Plastics

Snap-Fits



Keep It Flat

Keep It Thin

Longer Is Better

If a Snap-Fit or Flex Finger breaks under stress, make it flatter, thinner, or longer to improve its strain capability.

10

Participant's Notes:

Molded-in Inserts

Molded-in metal inserts can cause cracking.

Metal inserts are sometimes molded-in during processing, but the metal can create hoopstress that weakens the boss. During cooling, the metal insert retards the surrounding plastic boss from shrinking freely causing a hoopstress and a weakened part. Excessive hoopstress will eventually cause the boss to crack. It is suggested that metal inserts not be molded-in, but inserted later. **STK 1045**

Snap-fits

Snap-fit design rules: keep it flat, keep it thin, and longer is better.

Snap-fits are helpful in facilitating part assembly. When designing a snap-fit, it is important to keep it flat, keep it thin, and longer is better. Adding support to the bottom, or increasing the thickness of the snap-fit will increase the stress in the snap-fit. Lengthening rather than shortening the snap-fit will reduce the strain during assembly.

STK 1046

GE Plastics

High Performance Part Design

Moldability

Optimize

Tooling Considerations ⟺ Part Geometry

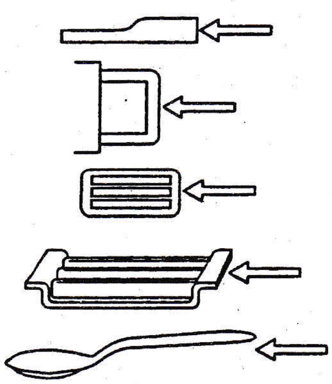
10

Participant's Notes:

GE Plastics

Gate Location

The Gate or Gates Should Direct Flow...



From Thick Sections to Thin

Against the Cavity Wall to Avoid Jetting

To Minimize the Effect of Weld Lines

In the Direction of Ribs & Other Reinforcements

Away from High Performance Areas

10

Participant's Notes:

Design Considerations
Tooling

Optimize tooling considerations.

High Performance Part Design: Tooling Considerations

Even while discussing moldability and part geometry, we've touched on tooling considerations. As we stated at the beginning, most important design issues are interdependent. Throughout the product development cycle, it is also important to optimize certain tooling considerations. **STK 1047**

Understand gate location.

Gate Location

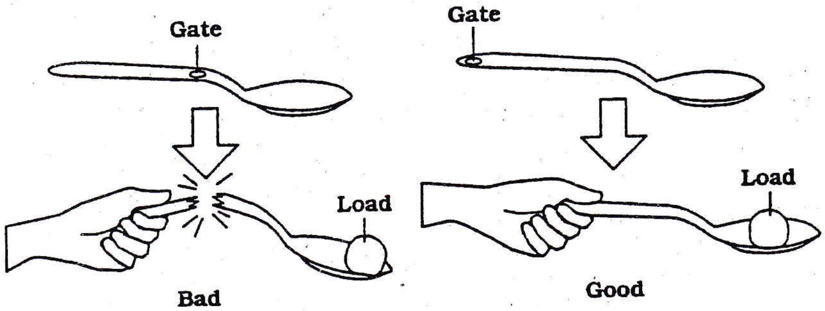
The gate location determines the flow of the material through the cavity. Gates should be located to direct the material flow from thick sections to thin sections for smoother flow. They should be located to allow impingement against a cavity wall or core to avoid jetting. Jetting occurs when the melt doesn't substantially touch the cavity surface to create a flow front but hurls forward undisciplined. Gates should be located to minimize the effects of weld lines. If possible, gates should direct the flow in the direction of any reinforcement ribs. The gate is an area of stress and should be located away from high performance areas.

Multiple gates are often designed to reduce the flow length and create better balanced loading on the core. Multiple gating introduces weld lines and weak areas due to incomplete fusion of the flow fronts. It is the challenge of the design team to weigh the various advantages and disadvantages of multiple gating when deciding the best way to design the part. **STK 1048**

STK 1049

GE Plastics

Gating & Performance



Since the gate is a stressed area it should be located in an area which avoids load, heat, pressure, UV, or chemical exposure.

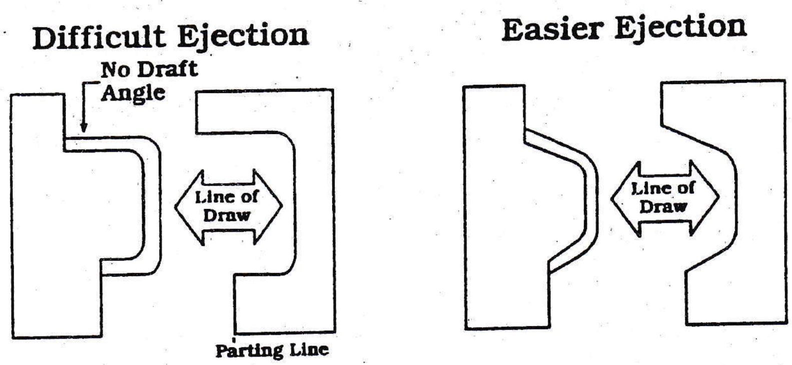
10

Participant's Notes:

STK 1050

GE Plastics

Ejecting the Part



Generous draft angles will facilitate the part ejection.

10

Participant's Notes:

Gating & Performance

The gate affects the surface appearance of a part, it is often located according to aesthetic preference. The gate is also an area of molded-in stress, and should be located in an area which avoids loads, heat, pressure, U.V., or chemical exposure.

The gate should be located in an area which avoids load, heat, pressure, UV, or chemical exposure.

Gate size is also an important detail. While a small gate is usually aesthetically preferable, too small a gate may cause such problems as gate blush, cold flow, jetting, or improper filling and packing due to premature freezing at the gate. Again it is up to the design team to ascertain the optimum gate size considering the part and tool design, as well as the material being used. **STK 1049**

Ejecting the Part

Throughout the section of reinforcement structure, we emphasized the need to angle ribs, holes, and bosses for ease of ejection. This angle is called the draft angle. An angle that is parallel to the line of draw is difficult to eject, as the plastic tends to shrink tightly around the core upon cooling. A slight angle allows for easier part ejection.

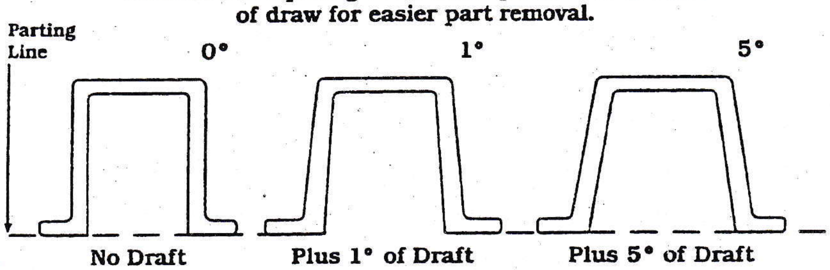
STK 1050

An easy part ejection is achieved with generous draft angles.

GE Plastics

Draft Angle

Draft is the tapering of surfaces parallel to the line of draw for easier part removal.



The larger the draft angle, the easier the ejection.

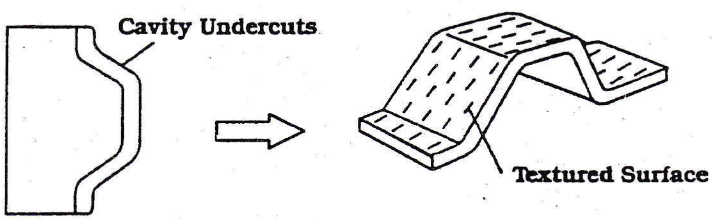
10

Participant's Notes:

GE Plastics

Textured Surfaces

Texturing the side walls of a cavity to enhance part appearance will produce undercuts:



Textured surfaces require a normal draft angle plus at least 1° per side for each .001 inch of texture depth.

10

Participant's Notes:

Draft Angle

Draft is the tapering of surfaces parallel to the line of draw.

Draft refers to the angling or tapering of part surfaces that are parallel to the line of draw for easier part removal. Generally, a draft angle should be included on any wall in the direction of draw; the larger the draft angle, the easier the ejection. The minimum amount of draft angle depends on the polymer being used.

Soft, ductile, self-lubricating materials such as polyethylene, nylon, and polypropylene will require less draft than hard, brittle, abrasive materials such as polystyrene, acrylic, or glass filled polysulphone. It is important for the designer to be specific when indicating draft angles. **STK 1051**

Textured Surfaces

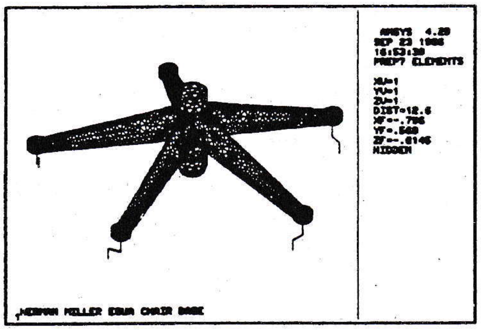
Textured surfaces require additional draft.

Thermoplastic parts are sometimes textured for aesthetic reasons. Texturing creates undercuts that may make ejection difficult. As a general rule, draft angles should be increased by 1° for every .001 inch of texture depth. However, the draft angle may have to be greater when molding stiffer materials. **STK 1052**

STK 1053

GE Plastics

Computer Aided Engineering



Computer aided engineering provides numerous analytical techniques to optimize materials & part & mold designs.

10

Participant's Notes:

STK 1054

GE Plastics

Understanding Design and Performance

10

Participant's Notes:

CAE facilitates part design.

Computer Aided Engineering

Computer Aided Engineering (CAE) allows the designer to examine the many design, material, and processing considerations up front at the product's inception. A computer program completes the necessary design iterations in less time than it would take manually. This allows the designer to arrive at the optimum design more quickly.

STK 1053

Summary and Performance Feedback

Understanding Design and Performance

Product development is a complex process requiring careful consideration of numerous issues such as functionality, form, material, manufacturability and cost. To integrate the design functions, simultaneous engineering offers an advantage over the traditional development process. Three important design considerations were discussed: moldability, part geometry, and tooling. Strategies to minimize molded-in stress and optimize part geometry were suggested.

STK 1054

STK 1055

GE Plastics

Describe the disadvantages of the traditional development process.

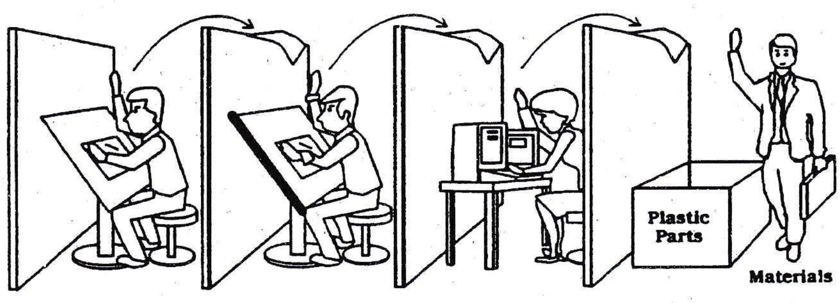
10

Participant's Notes:

STK 1056

GE Plastics

Traditional Development Process



Isolating each area of the product development cycle results in more development time, increased costs, and compromised part performance.

10

Participant's Notes:

STK 1057

GE Plastics

Identify two causes of molded-in stress.

10

Participant's Notes:

STK 1058

GE Plastics

Molded-in Stress

- Over-packing
- Uneven Shrinkage

10

Participant's Notes:

GE Plastics

List four advantages of uniform wall thickness.

10

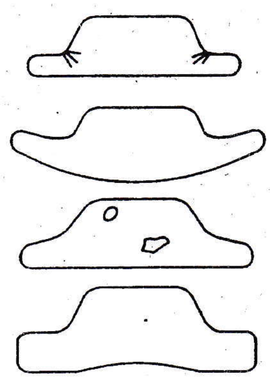
Participant's Notes:

GE Plastics

Uniform Wall Thickness

Minimizes:

- Molded-in Stress
- Warpage
- Voids
- Sink Marks



10

Participant's Notes:

STK 1061

GE Plastics

Describe the tooling considerations for ensuring high performance part design.

10

Participant's Notes:

STK 1062

GE Plastics

Tooling Considerations

- Gate Location
- Draft Angles
- Textured Surfaces

10

Participant's Notes:

Module 10

Performance Feedback

1. Describe the advantages of simultaneous engineering compared to the traditional development process.

2. Describe the relationships between moldability, tooling & part geometry that dictate high performance part design.

3. Describe strategies to minimize molded-in stress.

4. Describe strategies to optimize part geometry.

5. Describe the tooling considerations for ensuring high performance part design.