Plastic Part Design

A Practical Approach

Tuan Dao
WPTF 2011
SoCal-SPE

Design For Molding

Top Ten Design Tips

1. Comparison of Materials

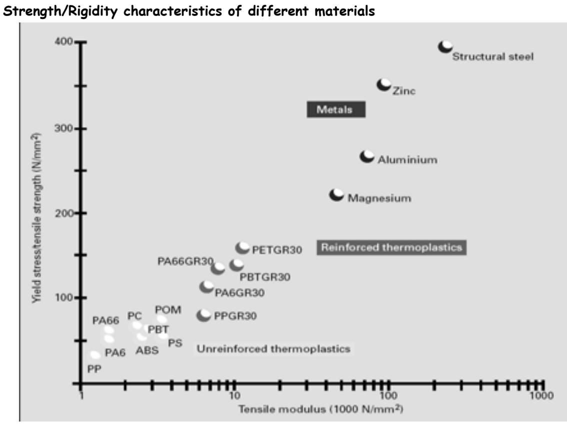
1. Comparison of Materials

- Many plastic designs still continue to be derived from “metal parts”.
- But Plastic is not metal.

Plastic vs Metal - A Direct Comparison

Plastics are greater:	Metals have higher:
Mechanical damping	Density
Thermal expansion	Max service temperature
Elongation at break	Rigidity/ strength
Toughness	Thermal/electrical conductivity

Strength/Rigidity characteristics of different materials



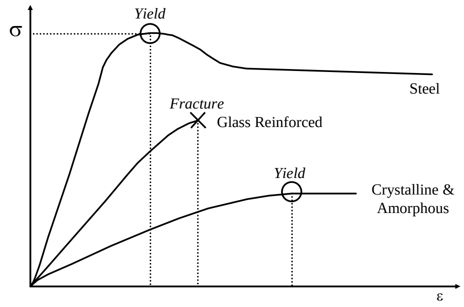
Different Material Behavior:

Plastics sometimes exhibit completely different behavior to that of metals under the same service conditions

- Plastics have Temperature and Time dependence of deformation characteristics at elevated temperatures.
- Factors influencing component properties:
  - Environment
  - Molding
  - Tooling

Mechanical Properties

Comparative Stress-Strain Curves



Tensile Modulus

For a linear elastic material

$$E = \frac{\sigma}{\varepsilon}$$

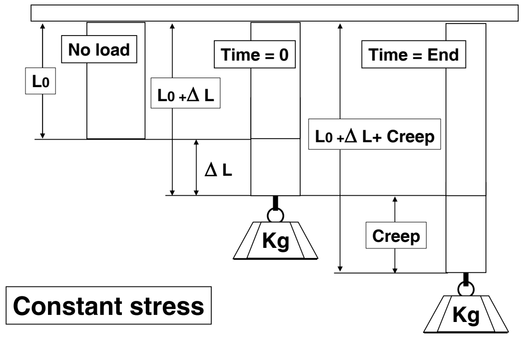
E	= Tensile Modulus	[MPa]
σ	= Stress	[MPa]
ε	= Strain	[%]

Long Term Loads

Creep

Plastic materials under load will undergo an initial deformation and will continue to deform at a slower rate with continue application of the load.

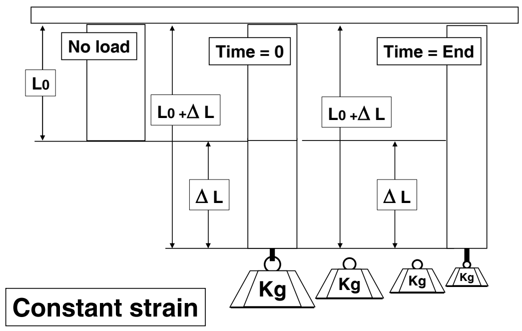
Creep principle



Stress Relaxation

Stress relaxation is defined as the decrease, over a given time period, of the stress required to maintain constant strain

Stress-relaxation principle



2. Material Selection

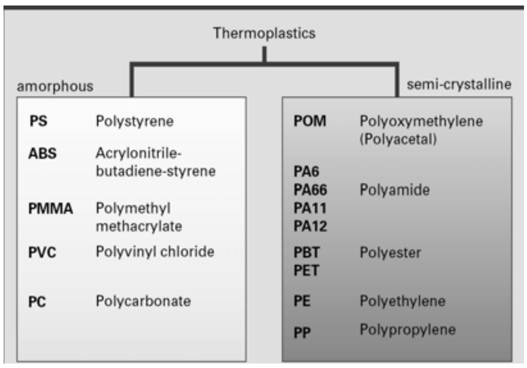
Material Selection

- There is no bad material – just the wrong material for a particular application.
- Designers need to know the material properties.

Thermoplastics

Amorphous vs. Crystalline

Engineering Thermoplastics



Property Comparison of Thermoplastics

	amorphous	semi-crystalline
Mechanical properties	O	+
Tendency to creep	+	O
Chemical resistance	–	+
Flexural fatigue strength	–	+
Critical strain	0,4%-0,8%	0,5%-8%
Notch sensitivity	–	O
Service temperature	O	+
Onset of melting	Softening range	Precise melting point
Shrinkage	0,3% – 0,8%	1,0%-3%

+ favourable O satisfactory – unsatisfactory

Effects of Additives

Additive	Max. content (% w/w)	Elastic modulus	Strain	Impact strength	Dimensional stability	Flame retardancy
Glass fibres	60	↑↑↑	↓↓	↓	↓	↑
Minerals	40	↑	↓	↓	↑↑	↑
Aramid fibres	20	↑	↓	↓	↓	↑
Elastomers	15	↓	↑↑	↑↑↑	↓	↓
UV stabilizers	1	↓	↓	↓	—	—
Flame retardants organic	20	↓	↓↓	↓↓	↑	↑↑↑
inorganic	40	↓	↓↓	↓↓↓	↑	↑↑↑↑
Antistatic agents	5	↓	↓↓	↓↓	—	—

Effects of Humidity

	moist	dry
Strength	↓	↑
Strain	↑	↓
Elastic modulus	↓	↑
Impact strength	↑	↓
Dimensions	↑	↓
Weight	↑	↓
Electrical properties	↓	↑

Choosing a resin

Select a resin for easy flow, fast cycling, low mold deposit, clean ejection.

Relevant properties for molding:

- Viscosity, melt flow, melt index
- Resin additives:
  - Nucleated, fast cycling, lubricated
  - Reinforcements or fillers

3. Wall Thickness

Wall Thickness

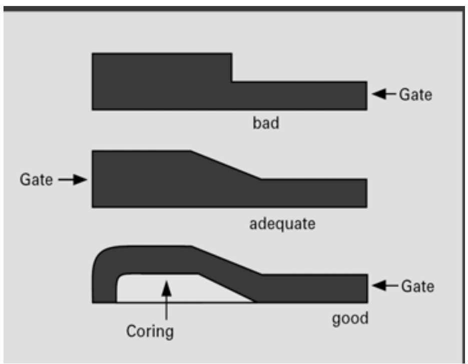
Maintain Uniform Wall thickness

Effect on Specific Component Criteria

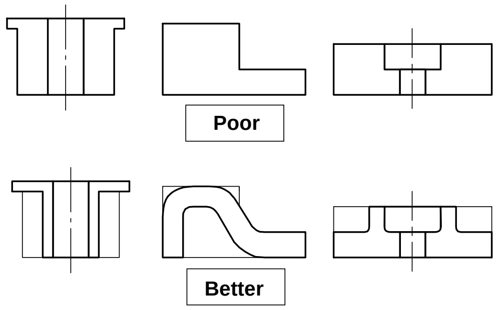
Changing the wall thickness of a part has a significant effect on the following key properties:

- Part weight
- Part rigidity
- Tolerances
- Surface finish
- warpage and voids
- Molding cycle

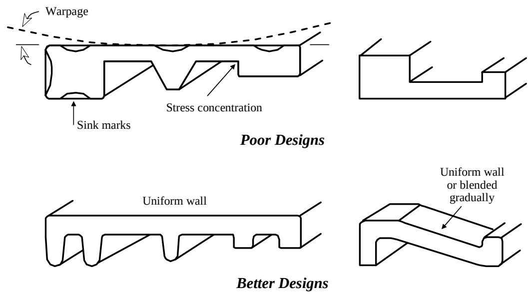
Transition Between Different Wall Thicknesses



Wall thickness design

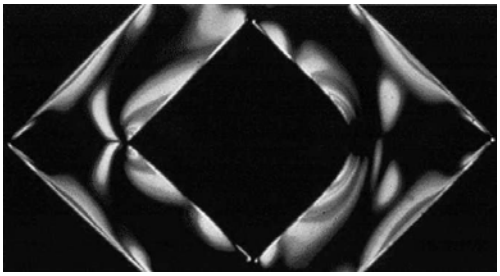


Non-Uniform Wall

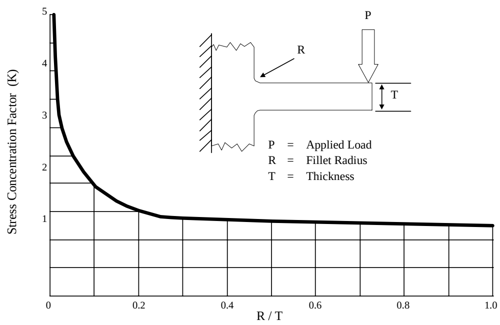


4. Avoid Sharp Corners

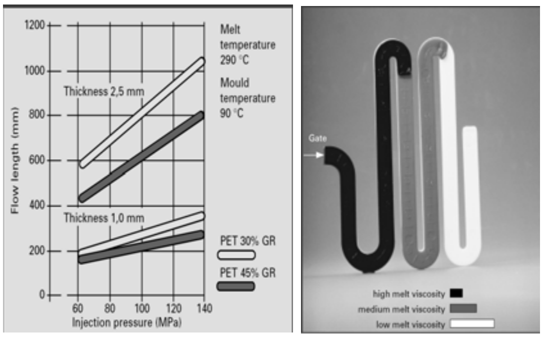
Avoid Sharp Corners



Effect of Fillet Radius on Stress Concentration Factor



Flow Length to Wall Thickness (L/T Ratio)



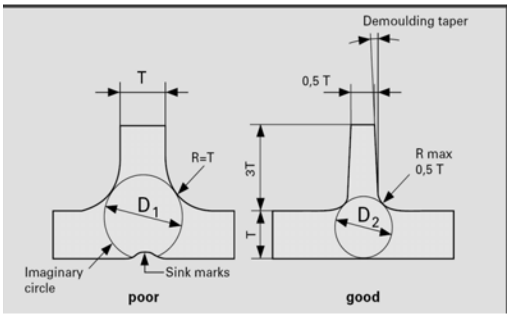
5. Optimum Rib Design

Optimum Rib Design

Ribs are an effective means of increasing rigidity while allowing wall thickness to be reduced. The rigidity of a part can be improved by:

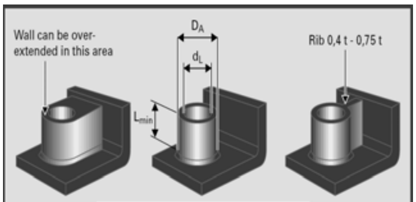
- increasing the wall thickness
- choosing material with higher elastic modulus
- incorporating ribs into the part design

Rib Design



Design recommendations

Correct dimensioning of the boss:



Material	d_L (mm)	D_A (mm)	L_{min} (mm)
POM	0,75 d	1,95 d	2,0 d
PET GR30	0,8 d	1,8 d	1,7 d
PBT GR30	0,8 d	1,8 d	1,7 d
PA66 GR30	0,82 d	2,0 d	1,8 d
PA66	0,75 d	1,85 d	1,7 d

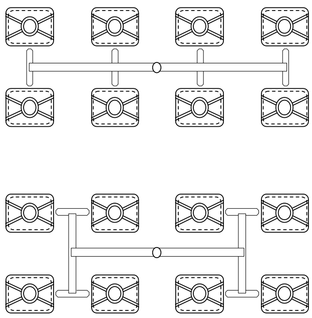
d = Nominal screw diameter

6. Runner Systems

Runners

- Melt “pipeline” from sprue to cavity
- Geometry is important
  - Full round – best
  - Trapezoid – ok
  - Half round – poor

Runner Systems



Unbalanced

Balanced

7. Gate Location

Gate Location

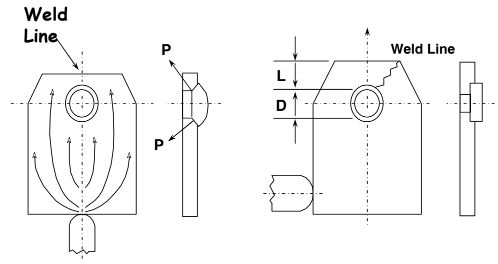
Gate location influences the following characteristics of a plastic part:

- Mold filling
- Final part dimensions
- Shrinkage
- Mechanical properties
- Surface finish

Gate Location

- Do not gate parts in highly stressed areas
- Avoid or minimized weld lines
- Avoid leaving weld lines in highly stressed areas
- With reinforced plastics, gate location determines warpage
- Avoid air entrapment by providing adequate vents in the mold

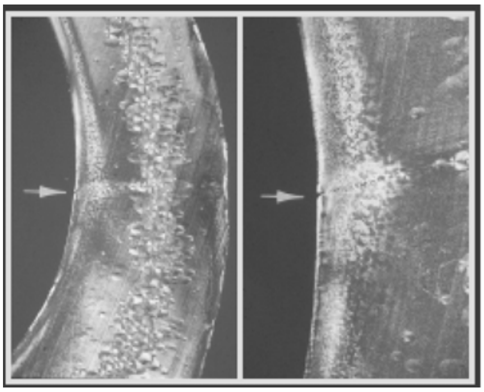
8. Weld Lines



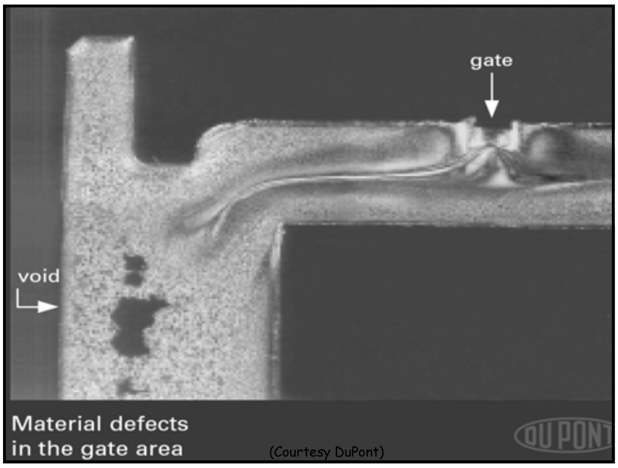
The weld line is at the weakest cross section.

Change gate location and keep L larger than D.

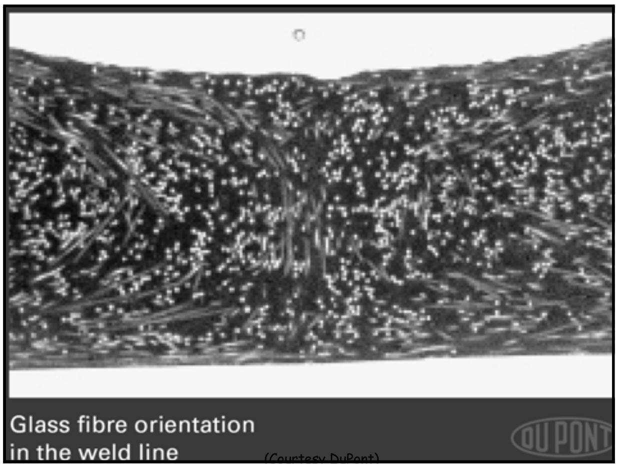
Weld Lines



Courtesy DuPont



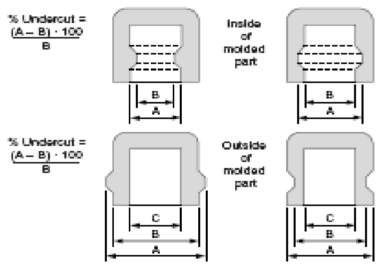
(Courtesy DuPont)



(Courtesy DuPont)

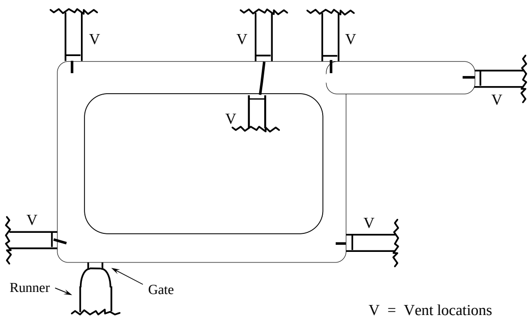
9. Undercut

Undercut

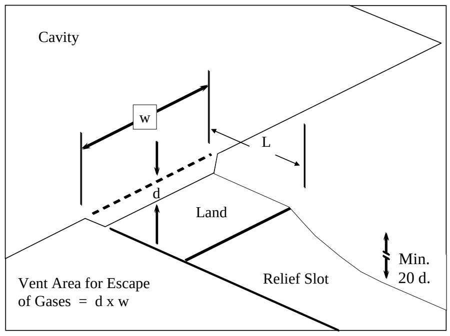


10. Venting

Vents at Weld Lines



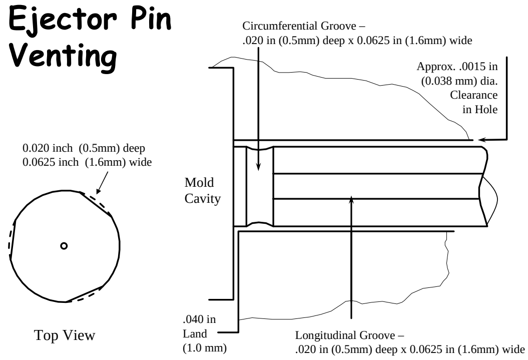
Parting Line Venting



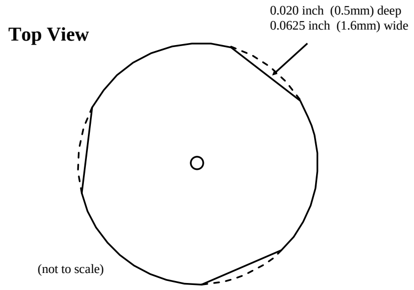
Venting Problems

- Consider:
  - Vent pins (must be self-cleaning)
  - Overflow tabs
  - Secondary parting lines (inserts)
  - Porous metal inserts (Porcerax® II)
  - Mold vacuum (MoldVac®)
- Anticipate
  - Difficult to fix once tool is built
  - Fixing can result in chasing the problem

Ejector Pin Venting



Ejector Pin Venting



9. Tolerances

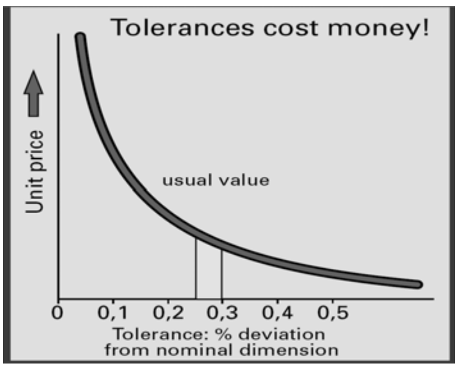
9. Tolerances:

Injection molded parts cannot be produced to the same tolerances as machined parts.

Factors influencing tolerances:

- Part design
- Material (amorphous, crystalline, regrinds…)
- Mold design
- Processing
- Molding equipments

Cost versus tolerances



10. Design Check List

10. Design Check List

- Aim for uniform wall thickness
- Design wall thickness as thin as possible and only as thick as necessary
- Use ribbing instead of greater wall thickness
- Provide radius, avoid sharp corners if possible
- Provide adequate draft for better mold ejection
- Avoid undercuts
- Do not design for greater precision than required
- Design multi-functional components
- Use economic assembly techniques
- Gate on the thickest section of the part

Design for Assembly

Design for Assembly

- Minimize the number of parts
- Minimize assembly surfaces
- Design for Z-axis assembly if possible
- Maximize part symmetry
- Use economic assembly techniques

Assembly Techniques

Technique Types

- Mechanical
- Welding
- Heading
- Adhesive

Mechanical Techniques

Primary Mechanical Techniques

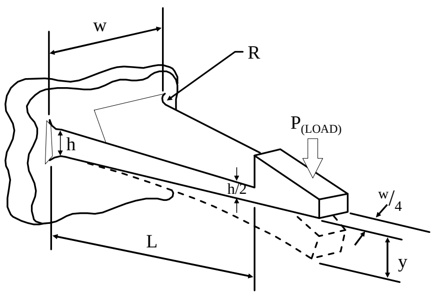
- Snap fits
- Press fits
- Threaded inserts
- Self-threading fasteners

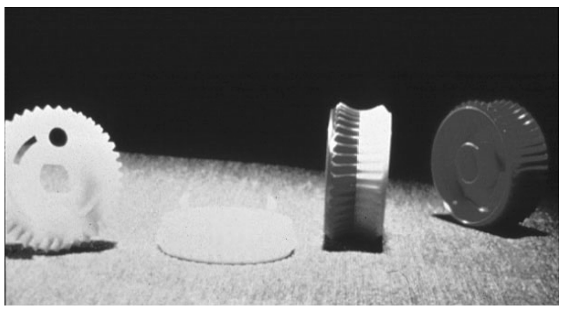
Basic Types

- Cantilever
  Beam – Flexure load
- Annular
  Shaft in hub – Multiaxial load
- Torsional
  Twisting bar – Shear load

Cantilever Snap Fits

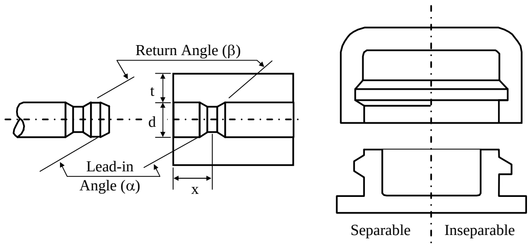
Cantilever Design Considerations





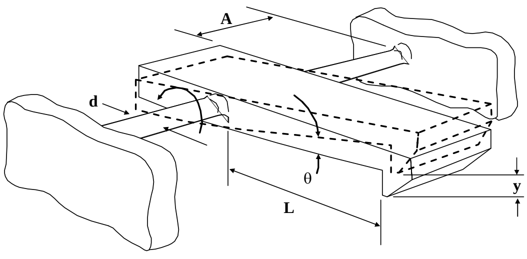
Annular
Snap Fits

Annular Snap Fit



Torsional
Snap Fits

Torsional Snap Fit



Torsional Design Considerations

- For maximum deflection
  - Maximize torsion bar length (A)
  - Maximize snap-lever length (L)
  - Use ductile plastics
- For maximum retention force
  - Maximize torsion bar diameter (d)
  - Minimize torsion bar length (A)
  - Use high modulus plastics

Advantages of Snap Fits

- Unlimited assembly and disassembly
- Can be made permanent
- Inexpensive
- Fast
- Strong

Disadvantages of Snap Fits

- May need a complicated mold
- Non-hermetic seal

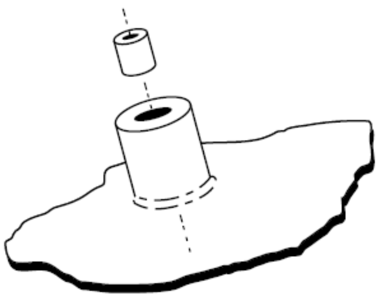
Press-Fitting

Press-Fitting

Press-fit assembly uses the interference between male and female features to create a permanent joint.

Press-fitting may be used to joint plastics to other plastics or plastics to metals. Press-fitting provides joints of high strength at minimum cost.

Press Fit Assembly



Press-fit Allowable Interference

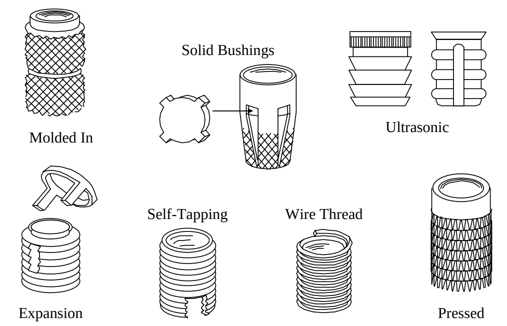
- Most applications utilizing press-fits are design for maximum pull-out forces.
- The allowable interference between cylindrical plastic materials and metals is generally limited by the tensile or compression strength of the part in the circumferential direction.
- The allowable interference varies with material properties, part geometry and environment conditions.
- Factor of safety must be incorporated in the press-fit design

Press-fit Allowable Interference

- Most applications utilizing press-fits are design for maximum pull-out forces.
- The allowable interference between cylindrical plastic materials and metals is generally limited by the tensile or compression strength of the part in the circumferential direction.
- The allowable interference varies with material properties, part geometry and environment conditions.
- Factor of safety must be incorporated in the press-fit design

Threaded Inserts

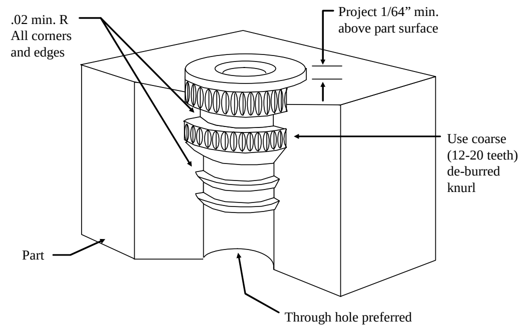
Inserts



Insert Uses

- Permit frequent part disassembly
- Increase joint strength
- Provide electrical conductivity
- Permit use of custom fasteners (i.e., 1/4 turn)
- Permit use of standard machine screws

Insert Design Considerations



Prototyping The Design

Benefits of Prototyping:

- Identify Design Problems
- Part Testing
- Assembly
- Cost Estimate

Prototype Modeling

Methods Used For Making Prototypes:

- CAD Generated Models
- Machined From Stock Shapes
- Molded From Die Cast Mold
- Molded From Prototype Mold
- Molded From Pre-production (Pilot) Mold

CAD Generated Models

- SLA/SLS Models
- 3D Printing Models

Machining from Stock Shapes

Usually from rod or slab stock. This method of prototyping is commonly used where the design is tentative and a small number of prototypes are required.

- Strength, toughness and elongation may be lower than that of the molded part.
- Effect of fiber orientation can be misleading.
- Surface characteristics (e.g., knockout pin marks, gate marks…) are not represented.
- Effect of weld/knit lines in molded parts cannot be studied.
- Dimensional stability may be misleading.

Prototyping From Stock Shapes

Advantages:

- Quick
- Low cost

Disadvantages:

- High cost per unit
- Limited selection of materials
- Property differences

Molding From Die Cast Mold

If a die casting tool exists, it can usually be modified for injection molding of prototypes. However this method may be of limited values. Because:

- The tool was designed for die cast metal, not for plastics.
- The wall thickness and ribbing will not be optimized.
- Gates are usually oversized and poorly located for plastics molding.
- Die cast tool is not equipped for cooling plastic parts.

Prototyping From Die Cast Tool

Advantages:

- Quick
- Large number of parts
- Convert tool back to die cast

Disadvantages:

- Impact toughness
- Molded in stress/warpage
- Dimensional tolerance
- Other mechanical differences

Prototype Mold

Prototype molds are usually made from inexpensive kirksite, aluminum, brass or copper beryllium. It is a better approach than molding from die cast mold. This type of tool will provide parts which are suitable for end-use testing. However, there are still some limitations:

- The mold can only withstand a limited amount of injection pressure.
- It may not be possible to optimize cycle time.
- Mold cooling may be limited or non existent.

Molding From Prototype Mold

Advantages

- Low cost
- Evaluate many materials
- Similar properties to commercial parts
- Evaluate molding problems

Disadvantages:

- Cannot duplicate production conditions or parts

Pre-production or Pilot Tool

The best approach for large design developments or precision parts is the construction of a steel pre-production mold (usually in P-20 steel).

- Commonly in single cavity mold, or a single cavity in a multi-cavity mold base.
- The cavity will have been machined finished but not hardened.
- It will have the same cooling as the production tool so that any problems related to warpage and shrinkage can be studied.
- Actual cycle times can be established.
- Most important, these parts can be tested in the actual or simulated end-use environment.

Molding From Pre-production Tool

Advantages:

- Duplicate production parts
- Precise cavity dimensions
- Define mold and molding parameters

Disadvantages:

- Higher cost
- Longer lead time

Testing The Design

Testing The Design

- Every design should be thoroughly tested while still in the prototype stage.
- Actual end-use testing is the best test of the prototype part. All performance requirements are encountered here, and a completed evaluation of the design can be made.
- Simulated service tests can be carried out. The value of such tests depend on how closely end-use conditions are duplicated.
- Field testing is indispensible. However long term field or end-use testing is sometimes impractical or uneconomical. Accelerated tests are used instead.

Writing Meaningful Specifications

Product Specifications

A specification is intended to satisfy functional, aesthethic and economic requirements by controlling variation in the final product.

Product Design Specifications

The designers' specifications should include:

- Material brand name and grade, and generic name (e.g., Zytel® 101, 66 nylon).
- Surface finish.
- Parting line location desired.
- Flash limitations
- Permissible gating and weld line areas (away from critical stress points)
- Location where voids are intolerable
- Allowable warpage
- Tolerances
- Color
- Decorating considerations
- Performance considerations