COMPROMISE FOR DESIGN—Achieving the optimum part design

The goal in designing a new part/product is to make the best compromise from all the factors available. You need to weigh all the facts and all the needs: The designer determines the QUALITY, the PRICING and the CONSISTENCY.

The toolmakers, and molders are stuck with the design they are handed. They can only try to optimize the situation. This design process is iterative. As you go through each section, and make a tentative decision, you need to go back to all of the prior sections to see how they have been affected.

1 PART DESIGN AND FUNCTION....Most critical is—Can it do what is needed? Did you Talk to the Customers (VOICE OF THE CUSTOMER)?, Marketing?, Sales?, Finance?, Production? They all have different needs and wants. And they are all important.

2 MATERIAL SELECTION...Plastic (thermoset or thermoplastic? Amorphous or Crystalline? Rigid or elastomeric?) Metal? Wood? Ceramic? Cloth? Where is it used----- Indoors, Outdoors? Chemical exposures? FDA requirements? UL requirements? ISO requirements? nsf requirements—and on and on----- Lots of choices, and more than one material will likely work. The process you pick will also affect the material—and vice versa. See the attached checklist to make sure you have considered all of the factors


4 TOOLING DESIGN AND FUNCTION.. Most processes require tooling. And the part design needs to be adjusted for the tooling requirements. Also, the desired life of the tool can affect the design requirements of the tool and the part design.

5 SECONDARY OPERATIONS, AUTOMATION, and ASSEMBLY (Ultrasonic, heat, dielectric, and mechanical) FINISHING (pad print?, silk screen?, hot stamp?, painting?, plating?) Everything you sequentially do with this part, will come back to affect the part design. They all need to be considered BEFORE you start any tooling.

6 QUALITY CONTROL and CONSISTENCY. The part needs to be readily inspected and measured. We need to go through steps of Validation and verification. The easier you make this process, the more likely it will be checked on a regular basis

7 END OF LIFE and ENVIRONMENTAL concerns. This is becoming more and more of a critical concept. And it does affect the part design, and the materials chosen to make the part from. The parts can affect the environment, just as the environment can affect the part.

8 Be Careful where you get help and information. If you go to an Architect with a problem, you will likely be given a building as a solution. You will get different "help" from Processing engineers, Assembly managers, Toolmakers, Material suppliers—et al.
COMMON ISSUES IN INJECTION MOLDING

There are some very common issues that reoccur in many parts...So we need to be concerned about the following:

INTERNAL RADII. All plastics are "notch sensitive". Any internal corner can create a stress riser—Even if we have almost no internal stresses, this is a highly likely cause for an impact failure...

Solution: Add radiused corners at all internal corners. No problem—EXCEPT ---- larger radii will effectively increase the wall section...And we can get a sink issue. There are several guidelines about the optimum radius at the inside corners—and they hold true—up to the point of causing sink.

GATING The two weakest places on an injection molded part are the gate locations and the weld line.

The gate is the place of highest fill pressure, and hence the highest internal stresses are there. The gate location and type can also influence flatness and warp, weld lines, needed secondary operations after molding, molecular orientation of the polymer( and more)... 

WELD LINE The plastic flow will separate when flowing around a core in the mold ( hole, window, etc.). When it gets to the far side of the hole, the material has to meld (weld) back together... There are many reasons this area is weaker: The plastic is cooling as it flows, so the fronts of material may not fully melt back together. There can be trapped air at the weld lines—and this prevents the material from melting together (You can actually get "dieselering", or burning, when you compress trapped air).... Also, any of the fillers, or reinforcing agents ( glass fiber, for instance) do not cross the weld line. They only come up to the weld line. At best, your weld line is only about 95% of the strength of the UNFILLED material.

VENTING. We all learned that only one type of matter can occupy a space, at one time. IF we want to fill a cavity with plastic, we need to get the air removed. This can be done by properly placed vent locations —which will ensure we are getting air removed at the last places to fill, as well as all along the fill path. We can even make the venting positive (vacuum venting). Improper venting severely limits the processing window, and can affect the overall strength of the part.

RIBS. When ribs intersect a wall, they give us a locally thicker area. This can be a cause of sinks. Rules of thumb are to make sure the ribs (including radii on internal corners) are no more than 50% of the wall it intersects.

BOSSES. Bosses have center core pins surrounded by plastic. The pins cannot always carry the heat away as fast as the surrounding steel—so the pin heats up. This can create sink marks. There are lots of little tricks to help alleviate the potential sink marks

SINK MARKS and SHRINKAGE. All plastic materials get smaller in volume when the solidify ( transition from liquid to solid). The amount of shrinkage depends upon:

The material—Amorphous ( shrink less) or Crystalline, Reinforcements ( glass fibers, minerals used in the plastic

- --Flow or Cross flow directions. Flow direction usually has lower shrinkage than cross flow
- --Wall sections. Thicker parts have more shrink.
- --Mold Temperatures.
When you start to get sinks in a part, the plastic no longer contacts the mold surface... You actually create a vacuum between the plastic part and the mold cooling surfaces. The rule of thumb is to run the mold as warm as possible to still get the heat transfer needed, and achieve the desired cycle. This allows for better packing of the part.

Plastics are very good heat insulators. As parts get thicker, the cooling time goes up logarithmically. The longer a plastic stays hot, the more shrinkage it will exhibit. If the design rules are "stretched", the processor usually needs to increase packing and holding pressures, increase speeds, and cool down the material stock temperature and mold temperature (creating a compromise for surface uniformity and weld lines).

All of this creates HIGHER INTERNAL STRESSES.

HIGH INTERNAL STRESSES

Ok—You have gotten past the initial set of compromises. You have a pretty good idea of part design and have been through the first iterations of compromise... Main remaining concern is INTERNAL STRESS LEVELS.

First—What is the definition of a "good" plastic part?: It needs to look good, meet tolerances, and have LOW INTERNAL STRESS.

High internal stress gives you the following:

- Lower impact strengths
- Lower heat distortion resistance
- Lower chemical resistance
- Low dimensional stability, and warp
- Lower environmental stress cracking resistance
- Worse low temperature properties---

Not a one of these makes a better part... So what causes high internal stresses, and how do we prevent them?

In general, the harder we have to pack out an injection molded part (higher pressures, and longer holding times), the more internal stresses we build into the part... Why do we build these high internal stresses into the part? ... Most of the time, we do so to make up for bad part design....And the most common culprit is sink...

Because of the importance, I will repeat myself----. Plastics get smaller in volume when they cool from liquid to solid. This is referred to as shrinkage. Every material has a shrinkage range. The longer the material stays hot in the mold, the more shrinkage occurs... When you have details, such as ribs, bosses, improper thickness transitions (try to keep a transition distance of at least 5 times the change in thickness), they create a localized thicker area. This area takes longer to cool, so you get more shrinkage in the thicker area. This will give you a "sink". This sink can be measureable (often at the intersection of a rib to a wall), or just slight enough to cause a surface gloss difference (often at the parting line where you have a thinner wall section for the interlocks).

Customers don't like sinks or the gloss changes (even though they have caused them by their design). Molders try to accommodate by increasing injection and holding pressures, and adjusting mold temperatures. They can minimize the sinks with these tricks—but at the expense of increasing the internal stress levels.
Please note the attached Checklist. This lists many of the factors you need to consider and balance, in order to make a GOOD PLASTIC PART.

This paper was intended to be a simplified overview of the many factors required in designing a good plastic part. There are a lot more details available from the many design guides from the raw material suppliers. Many of the articles/ brochures that I have found useful over the years, have been put on our web site—
PACKAGING CONSIDERATIONS
BARRIER PROPERTIES
H2O (WALL THICKNESS)
CO2
NITROGEN
ETHYLENE OXIDE
GAS
OTHER
BIO-DEGRADATION
BACTERIAL/Fungi RESISTANCE
OUTGASSING (Electronics+Painting Foams)
SPECIFIC GRAVITY (BASIC MATERIAL)
SPECIFIC GRAVITY (FOAM)
RADIATION OPACITY
SURFACE FEEL (Tacky, Dry, Slippery, Smooth, Rough)
SOUND DAMPENING
OPTICS
CLEAR, OPAQUE, TRANSLUCENT (THICKNESS)
BIREFRINGENCE
REFRACTIVE INDEX (ND)
HAZE (AT WHAT THICKNESS)
TRANSMISSION (AT WHAT THICKNESS/WAVE LENGTH)
CRITICAL ANGLE (IE)
REFLECTIVITY
FINISH (SCRATCH / DIG)
APPEARANCE CHARACTERISTICS
COLOR
(PIGMENT/DYE/PEARLSCENT/FLUORSCENT/PHOSPHORESCENT)
FINISH (SPE-SI NUMBER/OPTICAL/TEXTURED/MOLD TECH/EDM BLASTED/RMS)
SURFACE UNIFORMITY
SECONDARY OPERATIONS
DECORATING
MOLD SPRAY ALLOWED
SILK SCREEN
HOT STAMP
LABELS
PAINT
PLATE (Vacuum, Electro, Electroless)
FLAME SPRAY
ELECTROSTATIC COATING
FLUIDIZE BED COAT
BLAST/HONE
MECHANICAL
DRILLING, TAPPING, CUTTING
ULTRASONICS, VIBRATORY+SPIN WELDING, DIELECTRIC
HEAT STAKING
ADHESIVES (HOT MELTS, PRESSURE SENSITIVE, SOLVENT)
SCREWS (SELF TAP, CAPSERTS)
INSERTS (EXPANSION, PRESS IN, ULTRASONIC)
SNAP FITS
INTERFERENCE FITS
STRESS RELIEF / HEAT OR RADIATION CROSS LINKING / POST MOLD ORIENTING)
MOLDING CONSIDERATIONS
ENCAPSULATION
MATERIAL SHRINKAGE+SHRINKAGE RANGE
TOLERANCES
SINK. VOIDS
POST MOLD SHRINKAGE & CRYSTALLIZATION
ORIENTATION
MOLDED IN STRESSES (FOAM)
SHEAR SENSITIVITY
VISCOSITY AND FLOW LENGTH
HEAT STABILITY
MOLD SPRAY
REGRIND ALLOWED (PROPERTY CHANGES DUE TO CROSSLINKING OR VIS BREAKING)
MATERIAL CHECKLIST
EXPECTED PART LIFE
USE OF PART
LIABILITY IF FAILURE
ELECTRICALS
SURFACE + VOLUME RESISTANCE
DIELECTRIC MODULUS
DIELECTRIC CONSTANT (Hertz)
DISSIPATION FACTOR
ARC RESISTANCE
ANTI-STATIC PROPERTIES
SHIELDING
MECHANICALS
COMPRESSIVE MODULUS
HOURS UNDER LOAD
TENSILE STRENGTH
TENSILE/FLEXURAL MODULUS
Elongation
CREEP RESISTANCE
ABRASION/SCRATCH RESISTANCE
TEAR STRENGTH
FATIGUE RESISTANCE
VIBRATION TESTS
HINGE CAPABILITY (MOLDED IN/COINED)
SHEAR STRENGTH
COEFFICIENT OF FRICTION (STATIC/DYNAMIC)
HOOP STRENGTH (BURST PRESSURE)
IMPACT (IZOD/DROP)
TEMPERATURE
HIGHEST USE TEMPERATURE (LOADED/STATIC)
LOWEST USE TEMPERATURE (LOADED/STATIC)
HOURS AT HIGHEST TEMPERATURE--LTHA
COEFFICIENT OF EXPANSION
THERMAL CONDUCTIVITY
MOISTURE
ENVIRONMENTAL HUMIDITY
WATER ABSORPTION
WATER ADSORPTION
HYDROLYSIS RESISTANCE
FLAMABILITY
UL 94 /746 CLASSIFICATION
UL HEAT DISTORTION
OUTDOOR EXPOSURE
U.V. (LIFE)
CHEMICAL
MOISTURE
CHEMICALS
EXPOSURE TIME/TEMPERATURE
LOAD WHEN EXPOSED
STRESS CRACKING RESISTANCE
ACIDS (STRONG/WEAK)
ORGANIC SOLVENTS
ALKALAI (STRONG/WEAK)
GREASES / OILS / LUBRICANTS
PERSPIRATION, BODY FLUIDS
OTHER PLASTICS
FOODS (FDA)
MEDICINES (ALCOHOLS/BLOOD)
COSMETICS
PAINTS/COATINGS/ADHESIVES/ CLEANING
DE/IONIZED H2O
POTABLE WATER (NSF)
GREY WATER (SEWAGE)
Consider both the effect of the environment on the plastic and vice versa
STERILIZATION
GAS (ETO) / RADIATION / CHEMICAL
STEAM (TEMPERATURE / PRESSURE / TIME)
RoHS, REACH, NSF, FDA, EPA
EN (ISO), EPA, ASTM, WEEE Specifications
ENVIRONMENTAL STRESS CRACKING
PART DESIGN CONSIDERATIONS
Control Plastics Inc. 377 Oyster Point Blvd #10 SSF CA 94080
800-600-2010 www.controlplastics.com sales@controlplastics.com

SPRU (CONVENTIONAL/HOT) SIZE
SPRU PULLER (Z OR UNDERCUT)
COLD SLUG WELL - (SIZE)
RUNNER BAR (MATERIAL/HARDNESS)
BACK-UP PLATES (1 7/8" MIN) OR INSERTS (MATERIAL HARDNESS)
(BLIND-THRU) POCKETS
Material hardness finish surface-treatment special Mfg.
CAVITY INSERTS
CORE INSERTS
(H-12, S-7, P-20, ALUMINUM, BE/CU, O-1, A-10, M-2, P-4)
All hardened inserts are to be stress relieved (double drawn). When
Feasible, stress relieve inserts after EDMing.
GATES – (Round Edge, Trapezoidal Edge, Rectangular Edge, Half Round
Edge, Tapered – ½ Edge, Tapered-Full Edge, Sub Gate, Thru Ej, Pin, Pin
Point, Diaphragm, Flash, Fan, Ring, Pool, Tab.
SIZE LAND
EJECTION – (PINS- DME – std., Pins-High Speed, Sleeves-Std/Special,
Blades, Plate, Double Action, Hot-side, Cammed Ejectors, Kicking Ejectors,
Spring Loaded, Multiple (via Springs & Jifty Locks), Guided Ej. Plate
(Required with All Sleeves & Pins under .062")
PARTING LINE (Regular/Streped)
SIDE ACTIONS (With Positive Return & Full Locking) – (A-Side/B-Side,
Angle Pin, Hydraulic, Cam, Cammed Ejectors, Kicking Ejectors,
Interlocking, Automatic Unscrewing, Collapsing Cores, Stilson Core, Hand
Loaded – Core/Slide, Molded In Inserts (All HL or
Molded In Inserts must fit in only one side of the mold, in only one
Direction, and in only one location.
SAFETIES
BERMER RETURN
DME RETURN
MICROSWITCHES
SPRING LOADED EJECTION
OTHER
COOLING
BASE ONLY
(ALL PLATES)
INTO EACH CAVITY & CORE
(INDIVIDUAL CONTROL)
FOUNTAINS
BAFFLED INNER CORES
HEAT SINKS
HEAT PIPES
RUNNERS - TYPE
SIZE __________(2 RD) __________(3 RD)
STREAMLINE/CONVENTIONAL/LAYOUT
VENTING
VACUUM POPPETS
VENT WELLS
P/L EDGE VENTS
THRU CORE/EJECTORS
VACUUM EVACUATION
SPARE PARTS

WALL THICKNESS/UNIFORMITY
SINK/VOID POTENTIAL (CORING)
RADII ON CORNERS
DRAFT
REINFORCING RIBS
GATING (SIZE, LOCATION)
WELD LINES (STRENGTH)
PARTING LINE
EJECTION (PINS, BLADES, SLEEVES, PLATES)
SIDE ACTIONS (A-SIDE, MECH, HYDRAULIC, EJECTORS)

HAND LOADED INSERTS
STRESS RELIEVING/POST MOLD CROSSLINKING OR ORIENTING
HOW IS PART USED?
WHAT HAPPENS IF IT FAILS?
EXPECTED FAILURE OR REJECT MODE?
WILL PROPERTY CHANGES, DUE TO AGING, CAUSE PROBLEMS?

FIXTURES NEEDED
COOLING
TRIM
Q.C.

Q.C. & INSPECTION REQUIREMENTS
DIMENSIONS
LINEAR
HOLE DIAMETER
CONCENTRICITY OR T.I.R.
FLATNESS (BOW/WARP)
PERPENDICULARITY
CRITICAL/CONTROL DIMENSION & TOLERANCE FIT AND FUNCTION
REQUIREMENTS
(INITIAL DIMENSIONS VS. PART TO PART UNIFORMITY)

APPEARANCE
GATE TRIM
SINKS
VOIDS/BUBBLES
FLASH
PARTING LINE MATCH
CLARITY
SCRATCHES
UNIFORMITY OF SURFACE
FINGER PRINTS
MOLD SPRAY
FLOW LINES (PIGMENT ORIENTATION)
WELD LINES
BLACK SPECKS
URNS
SPLAY/BLUSH
JECTOR MARKS
AQL LEVEL
SPECIAL TESTS
PACKAGING
LAYERED (DIVIDERS)
BULK
TIE WRAP
SEPARATE P.E. BAG
FOAM PACK
STACKED
Egg CRATE
SPECIAL TRAYS
PARTS PER BOX
PARTS PER BAG

TOOLING CHECKLIST
NUMBER OF CAVITIES
MATERIAL (PLASTIC)
SHRINKAGE (FLOW, CROSS, FLOW)
TOOL LIFE
PRESS SIZE
SHOT SIZE
ESTIMATED CYCLE (PARTS/HOUR)
MOLD TYPE (STANDARD, HOT RUNNER, 4 PLATE, INSULATED RUNNER
MOLD SIZE (MINIMUM OF 2" AROUND CAVITIES)
BASE MATERIAL (DME #1,2,3, ALUMINUM, H-13)