HENRY PLASTIC MOLDING INC.
41703 Albrae Steet, Fremont, CA. 94538 Phone 510-490-7991 Fax 510-490-3548

A Well-Molded Plastic Product

There are many steps to producing quality Plastic Injection Molded parts. Skip any step, confuse the order of the steps, or fail to pay close attention to the details of each step, and it may mean possible failure of the product in its function, appearance, performance or in customer acceptance. Each step should be completed in its proper order before progressing to the next, because each step depends on the success of the previous step.

1. Market Demand and Concept Development
2. Initial Budget Projections and Part Quantity Analysis
3. Initial Cost Analysis
4. Customer or company ñBuy-inòand approval of project
5. Design and Engineering
   - Consult with molder / mold maker
   - Plastic material options: Determine the physical environment that the plastic components must survive:
     - Outdoor, Chemical, Temperature, Load Conditions, Impact Requirements, Flammability Requirements, Second Operation Requirements, EMI, RFI
       - Talk to the resin manufacturers for their advice and other input
       - (Ask your Henry Plastic Sales Engineer at 510-490-7993 our Materials Resources List)
6. Tool Production Capacity Analysis (Optional)
   - (Monthly Parts Req. / Tool Cavities) Cycle time in seconds = Machine Hours / Month
     - 3,600 seconds per hour
7. Budgetary Quote (Optional)
   - Mold price, part prices, time frame (Lead time for mold build to T-1 samples)
   - What is most important: Part price, mold price, delivery dates, part quality?
8. Function, Fit-up, Ergonomics, FCC, UL, NEBS, RoHS, UV and Assembly Considerations
   - Determine Material to be used, i.e. exact material type, grade, color, specs.
   - Involve your preferred mold maker and molder at this point
     - Number of Cavities & Cavity Configuration: Tool cost & configuration vs. part cost
     - Mold Type & Mold Class: Dedicated Base vs. M.U.D. Insert, Standard or 3 Plate, Hot Runner, Prototype or Production, Mold Class (See www.henryplastic.com/links.php)
     - Mold Material and Hardness: Aluminum, Steel, Pre-Hard (P-20, NAK80), Hard Steel (H-13)
     - Product life and total mold cycle requirements
     - Build time (Lead Time to T-1 Samples)
     - Production parts delivery dates (Not the same as T-1 Lead Time)
     - Gates, parting lines, ejection, side action, lifters, slides, proper plastic design
     - Runner system: Standard, size, lay-out, balancing, hot runner, 3 plate, multiple drops
     - Slides, lifters, unscrewing cores, collapsing cores and other mechanics
9. Marketing and Senior Management ñBuy-inò

Release the Project, Stage 1

10. Testing Requirements and Measurements
11. Set a Schedule and Know the Deadlines
12. Design, Engineering and CAD File Preparation 3D & 2D dimensioned drawings with tolerances) with Design for Manufacturability and Design for Testing in mind
13. Allow for adequate manufacturing time requirements (Often 4 to 12 weeks, depending on part complexity)
14. Decorating, Engraving, Color, Texturing, Decals, Labeling, Painting, Shielding & Other Cosmetic Decisions
15. Packaging
16. Second Operations Considerations, machining, welding, assembly
17. Bill of materials (BOM) for springs, metals, gaskets, labels, threaded inserts
18. Part Pricing and Quoting
19. Off-shore vs. On-shore, and shipping considerations

Release the Project, Stage 2

20. Prototyping and method
21. Tool Design, Cavities and Tool Build
22. First Article (T1) Qualification Requirements
23. Production parts

HENRY PLASTIC MOLDING INC.
41703 Albrae Steet, Fremont, CA. 94538 Phone 510-490-7991 Fax 510-490-3548

Injection Molding: Plastic Part Design Guide

In general, consider the following issues and the impact that they have on your part. This list should not be thought of as an all-inclusive and complete list of considerations for your part. Always consult multiple resin manufacturers and the molder/mold maker for assistance and design guidance early in the design process.

1. Moldability and design in general (See checklist below)
2. Desired plastic injection molding process
3. Part quantity requirements are sufficient to justify the cost of molds
4. Function of the part
5. Expected lifetime of the part
6. Agency approvals are required, i.e. FDA, UL, NSF, USDA, MIL spec, etc.
7. Electrical characteristics required
8. Temperature the part must tolerate
9. Chemical resistance needed
10. Moisture resistance
11. Weather and or UV exposure
12. Material shrinkage and shrink rates vs. tolerances
13. Testing process and worst case environment
14. Part cost reduction considerations
15. Press sizes required to run the parts
16. Designs that allow faster molding cycle times
17. Undercuts and other side action features.
18. Combining multiple parts
19. Assembly
20. Permanently assembled (one time) or temporary assembly (multiple times and may allow user access)
21. Adhesives
22. Fasteners
23. Insert molding and resulting continuous stresses
24. Snap fit features
25. Molded-in threads
26. Surface appearance and cosmetics
27. Color
28. Painted, Primer, Curing, Baking
29. Decorating/artwork and the application process
30. Labeling requirements
31. Required tolerances
32. Impact
33. Part loading requirements and resulting induced stress
34. Loaded continuously or intermittently
35. Permanent deformation or creep
36. Acceptable deflection
37. Wear resistance
38. Sterilization and methods, i.e. chemical, steam, radiation
39. Living hinges
40. Second operation machining
41. Part weight
42. Packaging, nesting and shipping
43. Selection of the proper injection molder, decorator and assembly service

HENRY PLASTIC MOLDING INC.
41703 Albrae Steet, Fremont, CA. 94538 Phone 510-490-7991 Fax 510-490-3548

Injection Molding: Plastic Part Design Checklist

X	Item	Specification	Purpose
☐	Draft	0.5 degrees or more 1.0 degree per inch of depth	Release part from mold
☐	Draft	Toward ejector pins	Improved cosmetics
☐	Wall thicknesses: Thicker is not necessarily better.	.030ò to .150ò Avoid large variations in wall thicknesses. Avoid abrupt changes in wall thickness.	Thin walls may result in weak knit lines, voids, short shots. Thick walls may exhibit uneven texture, sink, internal voids, excessive stress, warping and may result in an excessively long molding cycle time, which will add cost to the part price.
☐	Wall to Rib Ratio	50% to 75%	Proper wall ratios reduce sink and cosmetic issues
☐	Rib Height	3 times nominal wall thickness	Filling and sink
☐	External Wall Radii	Inside Radius: Approx. 50% of the nominal wall thickness Outside Radius: Approx. 150% of the nominal wall thickness	Radius corners strengthen parts. Improved flow, reduced stress, greater impact resistance
☐	Internal Rib & Boss Radii	Approx. 50% of the nominal wall with a minimum of .015ò	Radius corners strengthen parts. Improved flow, reduced stress, greater impact resistance
☐	Parting line edges	Sharp	Reduced mold cost
☐	Undercuts	Design to eliminate undercuts and features that require undercuts, or design to allow pass-through cavity and core with 5 degrees of shut off	Undercuts increase the cost of the mold
☐	Gates: Number, Location & Type	Diaphragm, Edge, Fan, Flash, Hot Edge, Hot tip, Pin, Sprue, Sub, Tab, Valve, Ejector Pin	Consult molder for best understanding. Gates can affect many issues involving the part
☐	Weld Lines/Knit Lines	Eliminate, reduce or reposition	Weld or knit lines are generally a cosmetic issue which should be considered during the design process
☐	Ejection location and type	Pin, sleeve, air assist, hand	Reduce cycle time, minimize cosmetic issues
☐	Ribs and other features	Encourage flow	Improved injection process, improved cosmetics & improved knit lines
☐	Threads	Avoid feathered edges & include radii at thread roots	Reduce molded-in stress and reduced stress concentration
☐	Textures or polished surfaces	See Mold Tech information and specifications at: www.henryplastic.com/links.php or www.mold-tech.com for guidelines. Increased draft may be required.	To provide tactile grip, cosmetic appeal, finger mark masking, no-glare surface and other benefits. Polished surfaces should be applied to hard steel molds.
	Engraving	Types and placement	Provides molded-in identity, labeling and artwork
	Proper files	Native files, .step files and .pdf files	The mold maker will need these files for proper mold build

HENRY PLASTIC MOLDING INC.
41703 Albrae Steet, Fremont, CA. 94538 Phone 510-490-7991 Fax 510-490-3548

Molder's, Toolmaker's and Tool Designer's Decision Checklist

1. Material type
2. Temperature requirements
3. Mold temperature
4. Pressure requirements
5. Clamping pressure required
6. Press size
7. Material drying time and temperature
8. Material shrinkage rate
9. Melt temperature
10. Efficient Process Conditions
11. Barrel size
12. Barrel speed in RPM
13. Runner size
14. Runner type
15. Hot runner
16. Runner geometry
17. Runner shut-offs
18. Cold Slugs
19. Cold slug traps
20. Runner balancing
21. Cavity balancing
22. Shot speed
23. Cycle time
24. Cooling method
25. Cooling period
26. Weld line location
27. Sprue valve
28. Tolerances
29. Weld line appearance
30. Weld line integrity
31. Hiding weld lines
32. Balanced fill
33. Material Burning
34. Barrel residence time
35. Minimize Clamp Force Requirements
36. Fiber Orientation
37. Even Part Shrinkage
38. Family tool shrinkage problems
39. Cooling volume, size and type
40. Plated cooling system requirements
41. Warp location and prevention
42. Cycle time for warp reduction
43. Packing requirements
44. Gate location, size and type
45. Number of gates
46. Scrap rates

47. Tool type
48. 2 plate, 3 Plate
49. Texture type
50. Engraving requirements
51. Engraving locations
52. Mold-tech or equivalent
53. Draft requirements for features & texture
54. Draft for part ejection
55. Ejection location
56. Ejection type
57. Drag points and marks
58. Flash problems & tolerances
59. Runner and or sprue ejection
60. Witness lines
61. Wall thickness transitions
62. Gas trapping
63. Core pin requirements
64. Slide requirements
65. Lifter requirements
66. Lifter angles
67. Hand loaded inserts
68. Parting line locks and parting line concerns
69. Vent size
70. Vent locations
71. Guide Pin size
72. Spring size
73. Interference ejection
74. Polished areas
75. Tool material
76. Stainless steel requirements
77. Soft, pre-hard or hardened tool
78. Folding runner
79. Oil cooling or water cooling
80. Foaming requirements
81. Tool inserts or direct cut
82. Over-molding requirements
83. Injection pressure
84. Pressure hold period
85. Part cooling fixtures for dimensional stability
86. Part holding features
87. Back pressure
88. Screw RPM
89. Surface laking
90. Pin or ejector cracking