<table>
<thead>
<tr>
<th>Process</th>
<th>Score</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plastic injected parts</td>
<td>60</td>
</tr>
<tr>
<td>Wall thickness</td>
<td>8</td>
</tr>
<tr>
<td>Metal parts</td>
<td>37</td>
</tr>
<tr>
<td>Hole drilling</td>
<td>12</td>
</tr>
<tr>
<td>PCB layout</td>
<td>81</td>
</tr>
<tr>
<td>Assembly</td>
<td>10</td>
</tr>
<tr>
<td>Package design</td>
<td>14</td>
</tr>
<tr>
<td>Packaging/Packout</td>
<td>13</td>
</tr>
<tr>
<td>Product design</td>
<td>14</td>
</tr>
</tbody>
</table>

**249**

DFM GUIDELINES
Curated by Our Best Design Engineers
DFM Guidelines for Plastic Injected Parts

1. If possible, use a uniform wall thickness throughout the part and avoid thick sections. This is essential since non-uniform walls can cause deformation or the part as the molten material cools.
2. The inside radius should be at a minimum of 50 percent of the nominal wall thickness.
3. An outside radius should be the nominal wall thickness plus the inside radius (150 percent of the nominal wall).
4. Sharp corners at the base of the heads and ribs can concentrate stress. The edge where a boss and the nominal wall meet should be rounded to reduce the sharp corner without increasing the thickness of the wall sufficiently to create sinking.
5. To avoid thin sections, the distance between ribs should be at least two and a half times the nominal wall diameter.
6. The draft angle of the ribs should be at least ½° per side to accommodate easier ejection from the mold.
7. To avoid non-uniform shrink that leads to warpage, balance ribs on both sides of the nominal wall.
8. Use connecting ribs and/or support gussets if possible to harden the structural parts. The connection ribs should be 0.6 times the nominal wall thickness at its base to avoid sinking.
9. To improve the strength of hollow sections, use ribs to design structures of equal strength and stiffness but reduced wall thickness.
10. Use ribs to improve the stiffness of horizontal sections without increasing their thickness.
DFM Guidelines for Plastic Injected Parts

11. Design ribs with a maximum thickness equal to 0.5 times the wall diameter.
12. Design ribs with max. height same to 3x the wall thickness.
13. Support bosses with ribs or connect them to a central wall.
14. The radius at the base of a boss should be ¼ of the nominal wall with a radius no smaller than 0.015.
15. Keep a wall thickness of less than 5mm. Thick walls lead to long cycle times and poor mechanical properties.
16. Avoid significant variations in wall thicknesses to simplify flow patterns and minimize differences in shrinking that lead to bending.
17. Avoid abrupt changes in the thickness of the wall; this can create stress concentration areas that may reduce a part’s impact strength. Wall thickness changes should have transition zones that minimize the possibility of stress concentrations, sinks, voids, and warp.
18. Avoid gating near an area with a significant variation in wall thickness because hesitation and race tracks can create non-uniform flow and shrinkage.
19. The maximum diameter of a rib should be 0.5 to 0.75 percent of the nominal wall to avoid sinking.
20. The maximum rib height should be no higher than three times the nominal wall thickness to avoid significant differences in wall diameters.
DFM Guidelines for Plastic Injected Parts

21. To maintain uniform wall thickness, the bosses should be cored to the bottom of the boss.
22. When designing molded threads, avoid feathered edges and include rounded roots to minimize tension concentrations and keep walls uniform.
23. Sharp edges can be stress concentrators in plastic parts. Thread designs should consider this.
24. Design parts with a minimum of $\frac{1}{2}$° per side-draft to accommodate easier ejection from the mold.
25. Core out parts with reasonably thin walls where possible.
26. Check if the application of mini tools can be applied.
27. Define parting line definitions in any areas where required.
28. Check if there are any missing surface finishes/textures.
29. Check if any parts that need a better definition for ejector pin location/size.
30. Make sure there are no component materials incompatible with intended processes.
31. See if any parts require specialized cooling systems and/or mold design for deep or critical areas.
32. Check if there are alternate resins that result in cost-saving that can be used (i.e., ABS to PP).
33. Make sure there are no areas where gating locations will cause aesthetic issues.
34. Make sure there are no inadequate draft angles on any of the molded parts.
35. Eliminate irregular parting lines where possible.
DFM Guidelines for Plastic Injected Parts

36. When different thicknesses are required, smoothen the transition using a chamfer or fillet with the length that is 3x the difference in thickness.
37. For interior edges, use a minimum radius of 0.5 times the wall diameter.
38. For exterior sides, add a radius equal to the inner radius plus the wall thickness.
39. To facilitate the ejection of the mold piece, a draft angle must be added to all vertical walls.
40. A minimum draft angle of 2° is recommended.
41. For features over 50 mm, increase the angle of inclination by one degree every 25 mm.
42. Avoid designing bosses that merge into main walls.
43. For bushings with inserts, use an outside diameter equal to 2 times the nominal insert size.
44. Do not add threads directly to your injection molded part.
45. Design bosses with an external diameter equal to 2 times the nominal diameter of the screw.
46. Add a relief of 0.8 mm at the edges of the thread.
47. Use a thread with a pitch greater than 0.8 mm (32 threads per inch).
48. Add a shot angle to the vertical walls of your quick-fit joints.
49. Design the pressure fittings with a diameter bigger than 0.5 times the wall thickness.
50. Adjust width and length to control deviations and allowable forces.
DFM Guidelines for Plastic Injected Parts

51. Design hinges with a thickness between 0.20 and 0.35 mm.
52. Select a flexible material (PP, PE, or PA) for live hinged parts.
53. Use shoulders with a thickness equal to the diameter of the main wall.
54. Add the steaks as large as possible.
55. Add a minimum interference of 0.25 mm between the crushing rib and the component.
56. Do not add a shot angle on the vertical walls of a rib.
57. Use embossed text (0.5 mm high) instead of recorded text.
58. Use a font with uniform thickness and a minimum font size of 20 points.
59. Align the text perpendicular to the separation line.
60. Use a height (or depth) greater than 0.5 mm.
DFM Guidelines for Wall Thickness of Different Plastic Materials

1. For Polypropylene, a wall thickness of 0.8 – 3.8 mm is recommended.
2. For ABS, a wall thickness of 1.2 – 3.5 mm is recommended.
3. For Polyethylene, a wall thickness of 0.8 – 3.0 mm is advised.
4. For Polystyrene, a wall thickness of 1.0-4.0 mm is recommended.
5. For Polymurethane, a wall thickness of 2.0 – 20.0 mm is advised.
6. For Nylon, a wall thickness of 0.8 – 3.0 mm is recommended.
7. For Polycarbonate, a wall thickness of 1.0 – 4.0 is recommended.
8. For Silicone, a wall thickness of 1.0 – 10.0 mm is recommended.
DFM Guidelines for Metal

1. In a sheet metal design, it is essential to specify the size of the holes, the locations, and their alignment.
2. The space between holes is essential. It must be at least twice the thickness of the sheet.
3. In cases where the holes must be near the edge, the minimum space between the side and the holes must be at least the thickness of the sheet.
4. Add Bends at edges to reduce the likelihood of metal tearing.
5. Put chamfers at corners and beads on bends to increase stiffness and reduce the spring-back effect.
6. Add collars around pierced areas to increase stiffness.
7. The minimum inner bend radius should be at least $1 \times \text{material thickness}$.
8. The depth of the relief of the curve must be greater than or equal to the inside radius of the curvature.
9. The width of the bend relief should be equal to or greater than the sheet metal thickness.
10. The diameter of the hole must be equal to or greater than the thickness of the sheet.
11. The outer radius of a curl should not be less than two times the material thickness.
12. For tear hems, the inside diameter should be equal to the thickness of the material.
DFM Guidelines for Metal

13. Notch Width should not be narrower than 1.5 * t.
14. As a rule of thumb, flexion perpendicular to the rolling direction is more relaxed than flexion parallel to the rolling direction. Folding sideways to the rolling direction can often lead to a fracture in hard materials, so it is not recommended to bend parallel to the rolling direction for cold-rolled steel.
15. No bending is acceptable for cold-rolled steel.
16. Hot rolled steel can be bent parallel to the rolling direction.
17. The minimum width of the flange must be at least four times the thickness of the material plus the radius of curvature.
18. The holes or grooves must be located a minimum of 3 times the thickness of the material plus the radius of curvature.
19. Sufficient burring holes height is needed to accommodate the stress during loading. It is recommended that the minimum burring height should be at least two times the sheet metal thickness.
20. The internal diameter of the milling hole should be sufficient to accommodate the pins, bolts, etc.
21. Burring holes that are too close to the bend will distort during the bending process. Distortion will be minimal if the distance between the edge of the Hole to the beginning of the inside bend radius is at least four times the material thickness.
22. It is recommended that the minimum distance between the burring hole edge and the bend should be at least four times the sheet thickness.
DFM Guidelines for Metal

23. It is advised that the minimum length from the edge of the milling hole and the side of the piece be at least four times the thickness of the sheet.
24. It is recommended that the minimum distance from a burring hole edge and the cutout edge should be at least four times the sheet diameter.
25. It is recommended that the minimum length between the edge of the milling hole and another edge of the milling hole is greater than or equal to 4 times the thickness of the sheet.
26. Interference in the flat pattern should be avoided for ease of manufacturing as there should not be any interference in sheet metal component design when the sheet metal part is unfolded.
27. It is recommended that a maximum ratio of half-shear depth to sheet metal thickness should be less than or equal to 0.6 times the sheet metal thickness.
28. It is recommended that minimum distance from notch to bend should be three times the material thickness plus inside bend radius.
29. It is recommended that the minimum distance between two Notches should be two times the material diameter.
30. It is recommended that the minimum distance between Notch and Hole should be 1.2 times the material diameter.
31. It is recommended that the minimum distance between the holes in a perforated metal should be equal to 1.2 times the material thickness.
32. It is recommended that the minimum distance from the edge of a circular hole to a bend should be three times the material thickness.
33. It is recommended that the minimum distance from the edge of a rectangular hole to a curve is 3.5 times the thickness of the material.
DFM Guidelines for Metal

23. It is advised that the minimum length from the edge of the milling hole and the side of the piece be at least four times the thickness of the sheet.
24. It is recommended that the minimum distance from a burring hole edge and the cutout edge should be at least four times the sheet diameter.
25. It is recommended that the minimum length between the edge of the milling hole and another edge of the milling hole is greater than or equal to 4 times the thickness of the sheet.
26. Interference in the flat pattern should be avoided for ease of manufacturing as there should not be any interference in sheet metal component design when the sheet metal part is unfolded.
27. It is recommended that a maximum ratio of half-shear depth to sheet metal thickness should be less than or equal to 0.6 times the sheet metal thickness.
28. It is recommended that minimum distance from notch to bend should be three times the material thickness plus inside bend radius.
29. It is recommended that the minimum distance between two Notches should be two times the material diameter.
30. It is recommended that the minimum distance between Notch and Hole should be 1.2 times the material diameter.
31. It is recommended that the minimum distance between the holes in a perforated metal should be equal to 1.2 times the material thickness.
32. It is recommended that the minimum distance from the edge of a circular hole to a bend should be three times the material thickness.
33. It is recommended that the minimum distance from the edge of a rectangular hole to a curve is 3.5 times the thickness of the material.
DFM Guidelines for Metal

34. It is recommended that the minimum distance (D) between the primary diameter of a countersink and a form should be four times the material thickness.

35. It is recommended that the minimum recommended distance between the diameters of the main relief and the edge be four times the thickness of the material plus the base radius of the relief.

36. It is recommended that return flange length to the sheet metal thickness should be greater than or equal to four times the material thickness.

37. A certain minimum distance should be maintained between two extruded forms to avoid fracture. Care needs to be taken when placing formed features close to each other. If a station does not clear away already put in a part, the form will get flattened out. The minimum recommended distance between the types should be greater than or equal to eight times the material thickness.
DFM Guidelines for Hole Drilling

1. Avoid unusual hole sizes as they would require custom tools.
2. Reduce variation in holes size as this will reduce the number of tools required during assembly.
3. Through holes are preferred over blind holes. Blind holes do not provide as much freedom of action for the exit and cooling of the chip. Operations such as reaming and tapping after drilling are more easily done in a through-hole.
4. Make blind holes with conical and not flat bottoms.
5. For drilling operations, it is recommended to avoid holes intersecting with cavities. If such an intersection is unavoidable, the centerline of the hole should be outside of the cavity.
6. Avoid partial holes, if unavoidable, make sure that at least 75% of the hole area is within the material.
7. Avoid deep and small diameter holes as they're difficult to machine.
8. The hole depth to diameter ratio should be less than 3.
9. Make sure drills enter and exit surfaces that are perpendicular to the centerline of the hole.
10. If possible, do not use holes that are smaller than an eighth of an inch in diameter. Drills for small holes usually break and are not recommended for mass production.
11. For large holes, try to make a preliminary hole that should only be drilled according to the specifications. This saves material, transportation costs, and drilling costs.
12. In the drawings, multiple holes should be placed on a flat surface from the same horizontal and vertical datums.
DFM Guidelines for PCB Layout

1. The V-cut should run across the entire length of the panel and not interfere with any component placement.
2. For panels with V-cut, the recommended board thickness is more prominent than 3.0mm.
3. A clearance area of 1.0mm on both sides of the V-cut is necessary for PCB panels that need automated depaneling.
4. Milling grooves are recommended to be 2mm.
5. The distance between the origins of adjacent stamp holes should be 1.5mm.
6. For small PCBs (smaller than 80mm x 80mm), panelization is recommended.
7. For irregular shape PCBs with missing parts, use block filler to create a more rectangular outline.
8. If the PCB doesn't have a clearance area of 5mm along the edge of the board, add margin bars along its perimeter.
9. Use standard component orientation and placement to minimize soldering shadows.
10. For irregular boards with padding pieces larger than 35 mm x 35 mm, use SMT and wave PCB. Size is defined by the equipment that your EMS provider is using in production. Size is important, and bigger is not always better. The larger the panel, generally, the more difficult it will be to process it.
11. PCB Shape must have two parallel sides (longest sides) to process through automation.
12. 125inkeep out area along board edges or rails/break off tabs is required.
1. Avoid unusual hole sizes as they would require custom tools.
2. Reduce variation in holes size as this will reduce the number of tools required during assembly.
3. Through holes are preferred over blind holes. Blind holes do not provide as much freedom of action for the exit and cooling of the chip. Operations such as reaming and tapping after drilling are more easily done in a through-hole.
4. Make blind holes with conical and not flat bottoms.
5. For drilling operations, it is recommended to avoid holes intersecting with cavities. If such an intersection is unavoidable, the centerline of the hole should be outside of the cavity.
6. Avoid partial holes, if unavoidable, make sure that at least 75% of the hole area is within the material.
7. Avoid deep and small diameter holes as they're difficult to machine.
8. The hole depth to diameter ratio should be less than 3.
9. Make sure drills enter and exit surfaces that are perpendicular to the centerline of the hole.
10. If possible, do not use holes that are smaller than an eighth of an inch in diameter. Drills for small holes usually break and are not recommended for mass production.
11. For large holes, try to make a preliminary hole that should only be drilled according to the specifications. This saves material, transportation costs, and drilling costs.
12. In the drawings, multiple holes should be placed on a flat surface from the same horizontal and vertical datums.
DFM Guidelines for PCB Layout

13. 25 in rail/break off minimum tab size.
14. Allow the EMS provider to optimize the lowest cost by having flexibility in panelization.
15. The panels become less stable as the size of the matrix increases. EMS providers will consider the batch size requirements when establishing the matrix size.
16. Include information on protruding parts (delineated and remote areas) The V-Groove score is applied to both sides of the board. It is a “V” shaped groove that leaves a 0.015in band of material to support the board. Components or other features should not be too close to the edge; otherwise, damage may occur.
17. Typically 0.03in – 0.05in should be allowed. -45 deg 0.015in V-Groove scoring guidelines.
18. Routing and perforated tabs refer to IPC-700. Perforated tabs are formed by three holes, each one 0.040in in diameter. Indent holes by 0.025in to avoid manual operation after depanelization.
19. The location of the cut tab shall be specified, if critical, as it may cause interference upon assembly into the application box or hardware.
20. The layers of the PCB must be balanced both in the layer count and in all areas to avoid deformation.
21. Specify surface finishes.
22. HASL is preferred for Tin-Lead applications.
23. Exceptions do apply for CSP, QFN, or ultra-fine pitch parts where pad coplanarity is critical. In these cases, HASL is not the preferred method due to pad doming.
DFM Guidelines for PCB Layout

24. Electroless Nickel Immersion Gold (ENIG) is preferred for RoHS applications.
25. ENIG has several advantages over more conventional (and cheaper) surface plating processes such as HASL, including excellent surface planarity, good oxidation resistance, and usability on untreated contact points.
26. Fiducials: 3 fiducials on corners of the board or panel. 0.050in round dot with 0.100in solder mask clearance.
27. Fiducials should be at least 0.190in from the rail edges to avoid clogging the clamps.
28. In large matrices, add local fiducials near any fine tone component.
29. The separation of the welding mask from the edges of the copper pad is typically 0.003in with a minimum of 0.002in.
30. Use Soldermask dams between pads to minimize solder bridging – 0.005in is preferred, but 0.003in is the industry minimum.
31. Connect standard pad sizes (IC & QFN) outside the pad area with a trace or dam with a mask.
32. Trace welding mask on common ground.
33. Component spacing 0.060in clearance around BGAs for inspection and rework.
34. The footprints of the components must be according to the specification of the manufacturer’s device part and IPC-7351.
35. For lead-free Quad-Pitch components: keep pad sizes consistent with each other to avoid tilting.
36. It is recommended to extend the pad size beyond the shape of the component for inspection purposes.
37. Via masking options: Covered, Flooded, Plugged, or Tented.
DFM Guidelines for PCB Layout

38. Cover the track to prevent welding from getting lost in the track and starving the joint.
39. Avoid bias within the ground pattern, under the components, or closely connected to the pads. If the tracks are near a pad, join the track or provide a welding mask dike between the pad and the track to prevent the weld from absorbing the track and stealing the weld from the intended joint.
40. Via size will boost the cost of PCB, the standard dimension is 0.010in.
41. Identify the polarities in silkscreen and drawings for assembly.
42. Indicate preference (or non-preference) in non-polarized parts.
43. Place the polarity consistently in one direction to reduce possible errors and inspection time.
44. Radial through-hole components should have a hole space typically of 0.100in, 0.200in, or 0.300in in the center.
45. Dual in-line (DIP) package components typically have a lead separation of 0.300in, 0.400in, or 0.600in.
46. Axial through-hole components shall have a center to center spacing of 0.300" to 0.800". using " as an inch indicator here, instead of the in you’ve been using so far?
47. Avoid hand-inserted “hairpin” axial device requirements.
48. Maintain a “Keep out” area of 0.125 inches? between the silver wire of the through-hole (PTH) and the surface support.
49. Technology components (SMT) facilitate selective welding.
50. Locate all PTH on the primary side of the plate to avoid manual welding.
DFM Guidelines for PCB Layout

51. Place all the large mass parts on the first side of the PCB and the smaller mass parts on the second hand. This will make manufacturing easier for the EMS provider.
52. Don’t reflect the components of the ball grid matrix (BGA) on opposite sides. This makes inspection and reworks very hard.
53. Minimize SMT component height on the secondary side of the PCB to a maximum of 0.25in.
54. In general, the secondary side should contain mainly passive components and smaller / low mass devices.
55. The large/high mass parts do not go from the secondary side. Large parts placed on the secondary side could overcome the surface tension of the solder paste and fall off.
56. Test pads should be 0.030in from other components. All test points on the secondary side of the PCB.
57. Provide drawings of test specifications to the EMS provider Define functional test methods.
58. Define the requirements for environmental test detection (ESS) if necessary.
59. Define PCB fabrications: copper weights and rolled type.
60. The common thickness for copper is 1 ounce (1.4mm or 0.0014in).
61. For solder mask type, liquid photo imageable in green is most common.
62. White is the most common silkscreen color.
63. FR4 RoHS laminate should be specified for RoHS applications.
64. 0.0625in, 0.031in & 0.093in are common board thickness.
65. Define workmanship standard: IPC 610 Class 2 or Class 3
66. Specify compliant coating requirements, if necessary Acrylic, polyurethane, or silicon.
DFM Guidelines for PCB Layout

67. Specify any “Keep out” area for coating (for example, headers, connectors, etc.)
68. Specify the potting requirements.
69. Specify any special adhesive, RTV, or tamper seals, if necessary. (Note on drawings)
70. Define labeling requirements for plate assembly and programmed parts.
71. Consider special packaging if necessary for finished products.
72. Specify any requirements of UL, CSA, ATEX, or other regulatory agencies.
73. Provide a list of imperative components.
74. Through holes that need to be “un-filled” or free of solder should be identified.
75. Often, parts “without material” must be added to the OEM and must have open holes.
76. Define flux and cleaning requirements or provide an assignment to use “No-Clean” flux.
77. Indicate if there are “non-hermetically” sealed components that cannot be washed.
78. A flying probe has “air exclusion zones,” which are high components that shade the access points. This can make the automatic inspection and test access difficult.
79. The handling of moisture sensitive devices (MSD) should be considered for delicate components and bare boards if they are shipped, keep the materials sealed per the manufacturer’s guidelines for MSD, and the audit process.
80. Components that need special preparation must be specified (e.g., parts must be maintained at a certain height tolerance on the PCB)
81. If sections of different thicknesses are required, make the transition as smooth as possible with a chamfer or fillet. In this way, the material will flow more evenly into the cavity, ensuring that the entire mold is filled.
DFM Guidelines for Assembly

1. Make sure joining or assembly processes don’t compromise the aesthetics (i.e., sink marks).
2. Make sure all parts are designed to meet stack assembly requirements.
3. Avoid using components that can be confused with other parts of similar shape (i.e., symmetrical shapes).
4. Make sure there are no parts that are too large and heavy which can lead to worker fatigue.
5. Avoid parts/labels that are hard to grasp, pick up, hold or require a tool for assembly.
6. Keep track of parts that can get tangled during assembly.
7. See if there are large base parts where other elements can be added to.
8. Modularize multiple parts into single subassemblies where possible for a more comfortable final assembly.
9. Be aware of any special cleanliness requirements that should be considered.
10. Check if any parts or assemblies require a particular environment during manufacturing (i.e., dust proofing, grounding).
DFM Guidelines for Package Design

1. See if there are packaging materials that are considered a standard that are not being used.
2. Check if the package structure is non-standard.
3. Make sure the intended package structure construction and pack out methods are clearly defined.
4. Check if there is an opportunity to reduce any package material thickness.
5. Make sure the package die lines are optimized to best utilize sheet or roll sizes of paper and plastic material.
6. Check if the draft is adequate for blister design.
7. Check if the flange is suitable for blister design.
8. Make sure if the design tolerance meets the process capability.
9. Make sure instructions can be printed on the package if possible.
10. Check if existing guidelines can be used instead of creating new instructions.
11. Check if the number of inserts/attachments be reduced.
12. Check if double-sided printing can reduce the amount of cardboard used.
13. Check if tab-ins and clip art can be combined and/or reduced.
14. Check if the package is using new materials or a new printing process.
DFM Guidelines for Packaging/Packout

1. Replace glue or tape for clip art with a standard mechanical locking approach where possible.
2. Secure the product by utilizing existing methods where possible.
3. Check if access is restricted during assembly.
4. Design the packaging to minimize the labor needed to secure the product in it.
5. See if the package structure can be assembled using a less labor-intensive method.
6. If the use of keylocks for securement can be beneficial, do so.
7. If the packaging operations can be automated, do so.
8. If a J hook is required, check if existing parts and attachment methods are not being used.
9. Make sure the packet can’t potentially damage the product.
10. Check if product components can easily be detached or damaged by the consumer.
11. Place the product, so it’s easier to remove for end consumers if possible.
12. Use spot welding instead of glue where possible.
13. Check if there is a more cost-effective tie-down material that can be used.
DFM Guidelines for Product Design

1. Minimize the number of (different) parts in a product
2. Use standard available components
3. Design parts that can be used in multiple product lines
4. Design components that are simple to produce
5. Minimize the number of flexible components
6. Design components that are multifunctional in a product
7. Design parts that cannot be assembled incorrectly
8. Maximize symmetry of parts or design them deliberately asymmetrical
9. Minimize loose fasteners (such as screws)
10. Ensure that all parts can be assembled in one direction. Preferably vertically from top to bottom
11. Ensure that parts align themselves for assembly
12. Make sure that the assembly is well visible and accessible
13. Design products that fit in standard packaging
14. Design the product modular