SAE Aerospace
An SAE International Group

AEROSPACE INFORMATION REPORT

SAE AIR1269 REV. B

Issued 1974-06
Revised 1997-12
Reaffirmed 2003-05

Superseding AIR1269A

Molded Rigid Plastic Parts

1. SCOPE:

This SAE Aerospace Information Report (AIR) provides recommendations for the best practices to be followed in the molding of rigid thermoplastic and thermosetting molding compounds. The molding processes covered in this document are injection, transfer and compression molding. Recommendations are made on the design details of the part such as draft angles and processing considerations such as gate location in order to yield an easily producible, high quality, functional part.

2. APPLICABLE DOCUMENTS:

The following publications form a part of this document to the extent specified herein. The latest issue of SAE publications shall apply. The applicable issue of other publications shall be the issue in effect on the date of the purchase order. In the event of conflict between the text of this document and references cited herein, the text of this document takes precedence. Nothing in this document, however, supersedes applicable laws and regulations unless a specific exemption has been obtained.

2.1 ASTM Publications:

Available from ASTM, 100 Barr Harbor, West Conshohocken, PA 19428-2959.

ASTM D 1693 Environmental Stress-Cracking of Ethylene Plastics

TO PLACE A DOCUMENT ORDER: **Tel: 877-606-7323 (inside USA and Canada)**
Tel: 724-776-4970 (outside USA)
Fax: 724-776-0790
Email: custsvc@sae.org
SAE WEB ADDRESS: http://www.sae.org

2.2 Other Publications:

"Plastics Engineering Handbook," Society of Plastics Industry, Inc., latest edition, VanNostrand Reinhold Publishing Company, New York

"Plastics Mold Engineering," J. H. DuBois and W. I. Prebble, sponsored by the Society of Plastics Engineers, Inc., Reinhold Publishing Corp., New York

"Plastic Product Design," R. D. Beck, VanNostrand Reinhold Company, New York

"Engineering Design for Plastics," E. Baer, Society of Plastics Engineers, Inc., Polymer Science and Engineering Series, Reinhold Publishing Corporation, New York, 1964

3. DEFINITIONS:

3.1 PROCESS COVERED:

Parts described in this document are those produced by molding in a closed, matched-metal-die mold by injection, transfer, or compression molding.

3.2 MATERIAL COVERED:

The material from which the parts are molded is classified as a rigid thermoplastic or thermoset with or without fillers, modifiers, or reinforcement.

4. GENERAL:

Molded plastic part failure is often blamed on the material when failure may actually be due to poor design of the part, poor mold design, or poor shop practices used to mold the part. For instance, the shop practices that are used are typically those that are most expedient or lowest in cost, although quality parts for high-performance applications may really require better or specialized techniques which are more expensive. The following recommendations cover some of the more common design and shop practice problems encountered and possible solutions; these are problems which are very often overlooked. Material manufacturers and/or their literature should be consulted for additional information on specific materials.

5. RECOMMENDATIONS:

5.1 Configurational:

5.1.1 Tolerances: Extreme accuracy of dimensions in molded articles is expensive to achieve. The closer the tolerances demanded, the greater the cost of the molds will be because of the precision required, and also the higher the operational costs of molding will be because of the greater care required to maintain uniformity of conditions. In some cases, a further expense arises from the need of using cooling fixtures after the molding.

5.1.1.1 Dimensional tolerances in a molded article are the allowable variations from a nominal or mean dimension.

5.1.1.2 This information is given in the form of charts to be used as a basis for establishing standards for individual molded articles, by agreement between the design engineer and the molder.

5.1.1.3 These charts are published in the “Plastics Engineering Handbook” (see 2.2).

5.1.1.4 The use of the charts is illustrated by an example of articles molded from ABS (acrylonitrile-butdiene-styrene). Figure 1 presents standards of tolerances for articles of the typical shape shown, molded from this material.

5.1.1.5 Note that the typical article shown in cross-section in the charts may be of round, rectangular, or other shape. Thus, dimensions A and B may be diameters or may be lengths and widths.

5.1.1.6 “Fine tolerance” represents the narrowest possible limits of variation obtainable under close supervision and control of production.

5.1.1.7 “Standard tolerance” is that which can be held under average conditions of manufacture.

5.1.1.8 “Coarse tolerance” is acceptable when accurate dimensions are not important.

5.1.1.9 Unless otherwise stated, the dimensions of the part normally are measured at least 24 hours after molding and after conditioning to a stable condition equivalent to 23 °C ± 1 (73 °F ± 2) and 50% ± 1 relative humidity.

5.1.2 Dimensions:

5.1.2.1 Measurement: Tight tolerances often lead to measurement problems if agreements are not reached beforehand as to atmospheric conditions during measurement. Dimensions can be correct immediately after molding in the molder’s plant and be wrong in the customer’s plant. The atmospheric temperature and humidity and the moisture content of the part may have to be stipulated to get agreements on measurements. Polyamide parts, for instance, may have to be conditioned by boiling several hours in water to simulate use conditions and larger dimensions at greater moisture content. Material manufacturers can supply information on proper conditioning of molded parts.

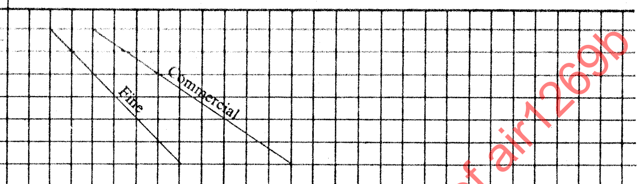
5.1.2.2 Dimensional Stability: Dimensions can change with environmental conditions such as temperature extremes, humidity, stress, and long term aging. Often post-mold baking is required to minimize the effects on the parts of these conditions. Baking will complete the cross-linking reaction of thermosets and will stress relieve parts molded in all materials to achieve more stable dimensions.

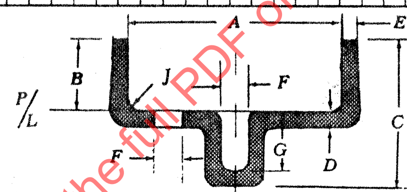
STANDARDS AND PRACTICES OF PLASTICS CUSTOM MOLDERS

Engineering and Technical Standards

ABS

NOTE: The Commercial values shown below represent common production tolerances at the most economical level. The Fine values represent closer tolerances that can be held but at a greater cost.

Drawing Code	Dimensions (Inches)	Plus or Minus in Thousands of an Inch 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28	
A = Diameter (see Note #1) B = Depth (see Note #3) C = Height (see Note #3)	0.000 0.500 1.000 2.000 3.000 4.000 5.000 6.000		
	6.000 to 12.000 for each additional inch add (inches)	Comm. ± .003	Fine ± .002
D = Bottom Wall (see Note #3)		.004	.002
E = Side Wall (see Note #4)		.003	.002
F = Hole Size Diameter (see Note #1)	0.000 to 0.125	.002	.001
	0.125 to 0.250	.002	.001
	0.250 to 0.500	.003	.002
	0.500 & Over	.004	.002
G = Hole Size Depth (see Note #5)	0.000 to 0.250	.003	.002
	0.250 to 0.500	.004	.002
	0.500 to 1.000	.005	.003
Draft Allowance per side (see Note #5)		2°	1°
Flatness (see Note #4)	0.000 to 3.000	.015	.010
	3.000 to 6.000	.030	.020
Thread Size (class)	Internal	1	2
	External	1	2
Concentricity (see Note #4)	(T.I.R.)	.009	.005
Fillets, Ribs, Corners (see Note #6)		.025	.015
Surface Finish	(see Note #7)		
Color Stability	(see Note #7)		



REFERENCE NOTES

1 – These tolerances do not include allowance for aging characteristics of material.

2 – Tolerances based on ⅛" wall section.

3 – Parting line must be taken into consideration.

4 – Part design should maintain a wall thickness as nearly constant as possible. Complete uniformity in this dimension is impossible to achieve.

5 – Care must be taken that the ratio of the depth of a cored hole to its diameter does not reach a point that will result in excessive pin damage.

6 – These values should be increased whenever compatible with desired design and good molding technique.

7 – Customer-Molder understanding necessary prior to tooling.

NOTE: All SI tables have been intentionally omitted.

FIGURE 1

5.1.2.3 Flatness: Out-of-flatness and warpage often occur and may be a problem even though dimensions are within tolerance. This problem could affect proper mating of parts or optical and visual values. Large flat areas, for example, are a problem and should be avoided. Such surfaces should be convex or the deviation from flatness and allowable warpage should be agreed on before the mold is made. Minimum warpage depends upon a balance of several factors. These include uniform part thickness, location of knockout pins, and optimum molding conditions. When minimum warpage is desired, cooling parts on a jig or fixture may be necessary.

5.1.3 Part Design: Parts are often designed with corner radii that are too sharp, allowing stress concentration points; sections heavier than necessary, causing excessively long molding cycles and high costs; greatly varying wall section thicknesses, causing warpage on ejection from the mold; improper anchoring of inserts, etc.

5.1.3.1 Ribs and Bosses: Ribs and bosses, frequently used on molded products, should be designed properly for best results. Parts which are not properly designed are weak at the base where they join the body of the piece; sharp corners must not be used on small ribs and bosses. When the height is too great in proportion to the width, the pieces do not fill out well, and this lack of density greatly reduces their strength. Figure 2 shows proportions for ribs which have been found to be generally satisfactory. Figure 3 shows correct proportions for bosses.

5.1.3.2 If many ribs are to be inserted, they should be staggered to reduce the distortion caused by unequal shrinkage. Ribs should not be located in the center of large areas unless required by the design. When central ribs must be inserted, reeds or flutes should be added to the outer surface of the piece to conceal resulting flow lines.

5.1.3.3 Holes: Flow of compound around mold pins causes considerable wear and some distortion. Mold pins should be built to the maximum diameter and replaced when they wear down to the minimum. Pins are frequently broken when flow conditions are particularly bad. Pin breakage is more prevalent in compression molding than in transfer and injection molding. A common practice is to mold deep blind holes to their maximum molding depth allowable for a specific pin diameter (normally having a length twice the diameter), and drill them to the full depth after molding.

Note that through-holes may be molded twice the depth of blind holes. The through-hole pins may butt in the center or be supported at each end by entering the matching section of the mold.

Mold builders frequently use two pins which butt in the center to produce deep holes. It is common practice to make one pin 0.030 inch (0.76 mm) larger than the other to compensate for mold wear, misalignment and deflection of the pins.

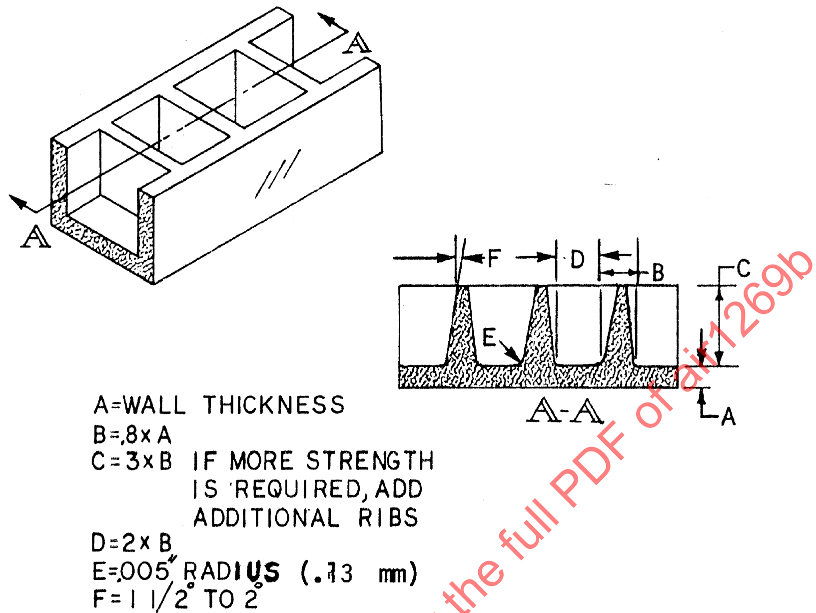


FIGURE 2 - Rib Design

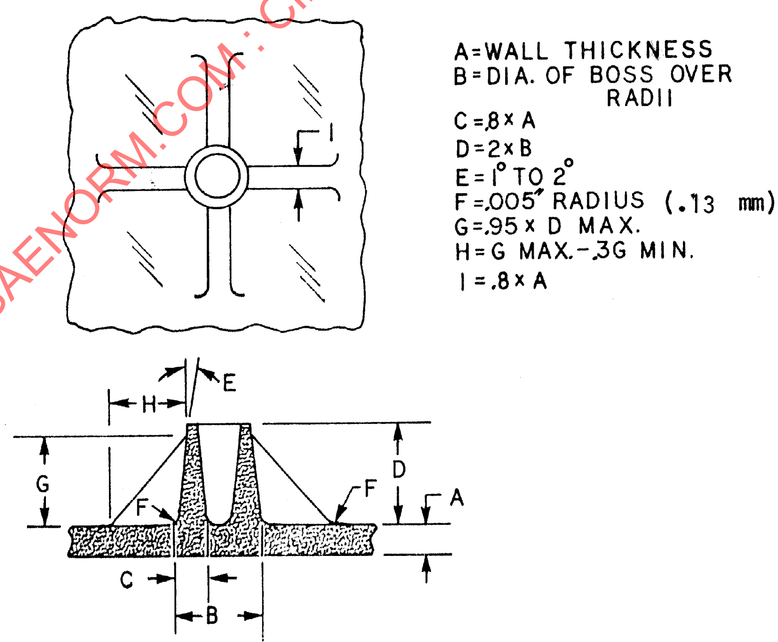


FIGURE 3 - Boss Design

5.1.3.4 Drilled Holes: When holes are to be drilled after molding, they should generally be spotted by the mold. This facilitates drilling without the use of drill jigs. A tapered, recessed bottom section will minimize chipping when the drill comes through.

Use of flat-head screws requiring countersunk holes should be avoided whenever possible if brittle plastics are specified. When these materials are used, the angle must be correct or the wedging action which takes place when the screw is tightened may break the piece. When countersunk holes are required, the screw head must seat below the surface by at least 0.016 inch (0.4 mm) to compensate for variations in the screws.

Table 1 shows the proper size of clearance holes for counterbores, screws, and nuts.

Bosses should be added around screw holes if the product is to be mounted on an uneven surface. Three holes or mounting points are best, as three points will always make contact on an uneven surface. If more than three mounting points are required, the bosses may be cut by sanding on a flat surface to ensure a fit without distortion of the piece.

5.1.3.5 Drilled and Tapped Holes: Figure 4 shows the preferred practice for trouble-free production. Holes which are to be tapped should be provided with a molded countersink to minimize edge chipping during the tapping operation.

Blind holes are frequently molded in plastic materials and are tapped after molding. Experience has shown that complaints often result from insufficient depth because this condition fails to provide adequate space for chips that accumulate in tapping. Because the depth of hole required is usually beyond the limits of economical production, holes are generally molded to the maximum molding depth and drilled to full depth before tapping.

5.1.3.6 Molded and Tapped or Self-Tapping Screws: Many products now use molded holes for self-tapping screws. It is frequently desirable to drill these molded holes for additional depth. Table 2 shows the correct molded-hole size, drilled-hole size, driving torque, and holding power of such screws. It will be well to check the hole sizes on products that are to use self-tapping screws.

The holes for the screws may be molded or drilled, whichever is more practical. If the material is brittle or friable, molded holes should be formed with a rounded chamfer and drilled holes should be machined chamfered.

TABLE 1 - Recommended Clearance Holes

Screw Size in Inches	Screw Dia. in Inches	Hole Std. Dia. in Inches	Hole Min. Dia. in Inches	Counterbores in Inches Standard	Counterbores in Inches Socket Wrench Hex Head	Counterbores in Inches Socket Wrench Hex Nut
No. 4	.112	+.000 .128-.003	+.003 .115-.000	11/32	---	---
No. 6	.138	+.000 .157-.007	+.003 .141-.000	13/32	---	---
No. 8	.164	+.000 .182-.007	+.003 .167-.000	7/16	---	21/32
No. 10	.190	+.000 .209-.007	+.003 .193-.000	1/2	---	11/16
No. 12	.216	+.000 .234-.007	+.003 .219-.000	9/16	---	---
1/4	1/4	+.000 .265-.007	+.004 .253-.000	5/8	3/4	15/16
5/16	5/16	11/32	+.004 .317-.000	3/4	15/16	1-1/16
3/8	3/8	13/32	+.004 .380-.000	13/16	1-1/16	1-3/16
1/2	1/2	17/32	+.004 .506-.000	1-1/16	1-1/4	1-3/8

NOTE: All SI tables have been intentionally omitted.

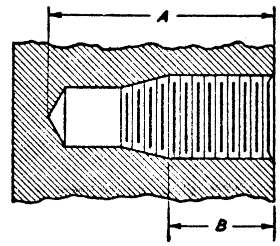


FIGURE 4 - Preferred Depth of Holes for Threaded Parts

TABLE 2 - Self-Tapping Screws for Plastics

Thermosetting Materials Screw Diameter	Thermosetting Materials Hole Required) (Inches)	Thermosetting Materials Drill Size No.	Thermosetting Materials Depth of Penetration[1] Minimum (Inches)	Thermosetting Materials Depth of Penetration[1] Maximum (Inches)	Thermoplastic Materials Hole Required (Inches)	Thermoplastic Materials Drill Size No.	Thermoplastic Materials Depth of Penetration[1] Minimum (Inches)	Thermoplastic Materials Depth of Penetration[1] Maximum (Inches)
No. 2-56	.073	49	7/32	3/8	Not Recommended (See Type Z)	Not Recommended (See Type Z)	Not Recommended (See Type Z)	Not Recommended (See Type Z)
No. 4-40	.098	40	1/4	5/16	.093	42	1/4	5/16
No. 6-32	.116	32	1/4	5/16	.116	32	1/4	5/16
No. 8-32	.144	27	5/16	1/2	.144	27	5/16	1/2
No. 10-32	.166	19	3/8	1/2	.166	19	3/8	1/2
No. 10-24	.161	20	3/8	1/2	.161	20	3/8	1/2
1/4 inch-20	.228	1	3/8	5/8	.228	1	3/8	1

[1] Includes taper.

5.1.3.7 Lettering on Molded Products: Molded letters, figures, and decorative designs are often used on molded parts. Lettering should be applied perpendicular to the parting line or be put on side walls which provide sufficient draft for the letters to “draw” out of the mold. These letters may be raised or depressed, or they may face up from a depressed panel. If the mold cavities are to be hobbed, the letters should be depressed. When cavities are to be machined, raised letters may be had at lowest cost. Depressed lettering requires raised mold letters; these are easily damaged and may be costly to maintain.

Small raised letters are generally 0.008 to 0.015 inch (0.20 to 0.38 mm) high. Larger letters may be 0.016 to 0.032 inch (0.4 to 0.8 mm) high. In all cases, it is desirable to use considerable draft on the sides of letters and to have a good fillet where they join the molded piece. Failure to observe this practice will produce letters that will be fragile and may break off in the mold. Letters that are to be painted should have no greater width than 0.032 inch (0.8 mm). Wide, depressed letters and lines that are to be painted should be avoided if possible, as paint will wipe out when the excess is removed from the surface. Wide letters are frequently simulated by forming them with a series of parallel lines.

5.1.3.8 Draft: Draft is necessary for the removal of parts from the mold. It is possible, when absolutely necessary, to produce some surfaces without draft. In most jobs, however, failure to provide adequate draft will be the source of many difficult molding problems. A minimum of 1/2 degree taper per side is generally satisfactory although 1 degree per side is most desirable for production jobs. Side wall taper of large units must permit flow of material from the bottom of the mold up to the parting line when compression molds are used.

5.1.3.9 Wall Thickness: The wall thickness of all new products should be checked to make sure that it is adequate and uniform. Especially thin sections may require reinforcement with resin-treated paper or cloth (molded laminated). Good molding practice requires uniform section and minimum wall thickness to produce a fast and complete cure. Heavy sections attached to thin sections are troublesome and produce distortion and undercured parts. Thermoplastic products will often show concave depressions or “sinks” on heavy sections. Sinks are caused by internal shrinkage that takes place as the center hardens, thereby pulling in the outer surface. If thermosetting materials are specified, transfer or injection molds are best for a combination of heavy and light sections. All heavy sections should be cored out if permissible. Frequently, this is accomplished by locating blind holes on the underside, the round pins forming these holes being easily inserted in the mold. Designers should always core out the underside of heavy sections to expedite the cure and save material.

A wall thickness of 0.062 inch (1.6 mm) may be used on small pieces made from thermosetting materials, but 3/32 inch (2.4 mm) is a much safer minimum. Larger pieces such as business machines use thicknesses from 0.125 to 0.375 inch (3.2 to 9.5 mm). Thermoplastic crystalline polymers such as polyethylene, polypropylene, and nylon may be molded in sections as thin as 0.010 inch (0.25 mm). Amorphous polymers such as acrylics, polycarbonate, and polystyrene require a minimum wall thickness of 0.050 inch (1.27 mm).

The cloth-filled phenolic materials require considerable thicker sections to facilitate material flow. Ribs should be added whenever possible to act as feeders.

Brittle materials, such as the cold-molded and mineral-filled phenolics, require a heavy wall thickness; not less than 0.125 inch (3.2 mm) should be considered. In the absence of previous experience, ask for recommendations from the raw material supplier. Considerable loss may be experienced by the use of overly thin wall sections, and this practice is poor economy.

5.1.3.10 Fillets: Radii or fillets must be used in all inside corners to assist material flow and strengthen the part, as fractures start easily from sharp corners. Sharp corners on interior surfaces of all plastic pieces should be avoided. Molds which require sharp corners are frequently more costly and fragile than those made with generous radii in the corners. Part designers should eliminate troublesome features of this kind when designing the product unless the mold construction or assembly requires sharp corners where the sections join.

Designers frequently specify radii which cannot be machined. These may be changed in most cases to radii which are machinable. In the few cases where there is no alternative, such radii must be cut in the mold by hand - a slow and extremely costly procedure. Designers should always try to avoid corners which cannot be machined.

5.1.3.11 Sharp Outside Corners: Sharp corners are required at the parting lines but are usually undesirable at all other points. All outside corners should be radiused as much as possible to help material flow, reduce mold cost, and avoid sharp molded corners that chip and break easily during finishing operations. Where beveled or rounded edges are required, a flat area must be provided at the edge to eliminate feather-edge mold sections, which are easily broken. Mold designers must correct all corners and edges which would produce a knife- or feather-edge in the part.

5.1.3.12 Surface Finishes: The best finish for any mold is a highly polished surface protected by chromium plating. A highly polished mold surface minimizes sticking, improves flow conditions in the mold, and produces nicely finished pieces. Many special surface textures are demanded for decorative effect. The effectiveness of the finish depends on the ingenuity of the mold-maker. Dull or “sand-blasted” finishes are often specified for the purpose of minimizing light reflection. These finishes are produced by grit blasting or acid etching. Surfaces treated in this manner will “polish up” in time, however, and, because of this, better results may be obtained by grit blasting the chromium plate after the mold is finished.

All special surface finishes should be limited to flat areas, or to sides which provide considerable draft.

5.1.3.13 Parting Line Placement and Flash: The location and design of the parting line can affect function as well as appearance. The parting line should not be located on a bearing surface or a critical matting surface and should be specified. The maximum allowable flash length and thickness should be specified because this will increase as the mold wears. Flash dimensions must not exceed dimensional requirements. It should be noted that special procedures for avoiding or removing flash add to the cost of the part.

5.1.3.14 Knockout Pin Placement and Dimensions: Knockout pins may be necessary to eject the part from the mold. Improper placement will cause warpage of the part due to distortion involved during part ejection. Knockout pins that are too small may push holes through softer materials. Larger pin diameters may be necessary and sometimes may be the full diameter of small parts to minimize problems. Knockout pins often are difficult to align flush with the part surface. It is preferable to locate the pins flush to 0.010 inch (0.25 mm) below the part surface.