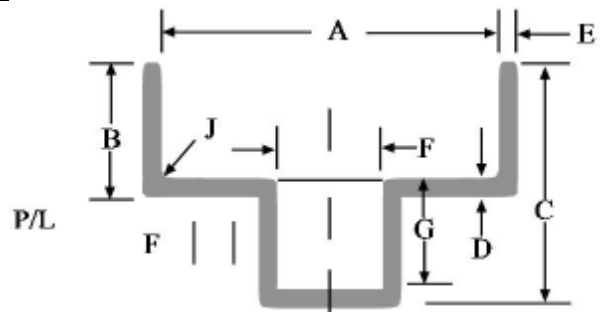


NOTE: The Commercial values shown below represent common production tolerances a of customer supplied critical dimensions.
The Fine values represent closer tolerances that can be held but at a greater cost.

Drawing Code			
A = Diameter (see Note #1) B = Depth (see Note #2) C = Height (see Note #3)	<p style="text-align: center;">Plus or Minus in Thousands of an Inch</p>		
	6.000 to 12.000 for each additional inch add (inches)	Comm. ±	Fine ±
D = Bottom Wall (see Note #3)		.005	.003
E = Side Wall (see Note #4)		.004	.003
F = Hole Size Diameter (see Note #1)	0.000 to 0.125	.003	.002
	0.125 to 0.250	.003	.002
	0.250 to 0.500	.004	.003
	0.500 & Over	.005	.003
G = Hole Size Depth (see Note #5)	0.000 to 0.250	.004	.003
	0.250 to 0.500	.005	.003
	0.500 to 1.000	.006	.004
Draft Allowances per side (see Note #5)		0°	0°
Flatness (see Note #4)	0.000 to 3.000	.016	.011
	3.000 to 6.000	.031	.021
Thread Size (class)	Internal	Inserts	Inserts
	External	1	2
Concentricity (see Note #4)	(T.I.R.)	.010	.006
Fillets, Ribs, Corners (see Note #6)		.026	.016
Surface Finish	(see Note #7)		
Color Stability	(see Note #7)		



REFERENCE NOTES

1. These tolerances do not include allowances for aging characteristics of material.
2. Tolerances based on 1/8" 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 design and good molding technique.
7. Customer-Molder understanding necessary prior to tooling