NADCA

S-4A-1-03

Standard Tolerances

The values shown represent Standard Tolerances, or normal casting production practice at the most economical level. For greater casting accuracy see Precision Tolerances for this characteristic on the facing page. See Section 7, "Quality Assurance" for more tolerance precision information.

Significant numbers indicate the degree of accuracy in calculating precision. The more significant numbers in a specified tolerance, the greater the accuracy. Significant number is the first non-zero number to the right of the decimal and all numbers to the right of that number. For example, 0.014. The degree of accuracy is specified by the three significant numbers 140. This is not to be confused with tolerance precision. A tolerance limit of 0.007 has a higher degree of precision because it is closer to zero tolerance. Zero tolerance indicates that the part meets design specifications exactly.

Linear Standard and Linear Precision tolerances are expressed in thousandths of an inch (.001) or hundredths of a millimeter (.01).

Notes:

Casting configuration and shrink factor may limit some dimension control for achieving a specified precision.

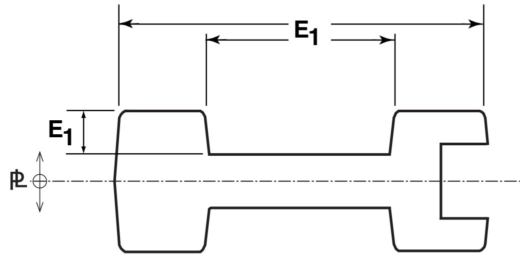
Engineering and Design: Coordinate Dimensioning

Linear Dimensions: Standard Tolerances

The Standard Tolerance on any of the features labeled in the adjacent drawing, dimension "E_1" will be the value shown in table S-4A-1 for dimensions between features formed in the same die part. Tolerance must be increased for dimensions of features formed by the parting line or by moving die parts to allow for movement such as parting line shift or the moving components in the die itself. See tables S-4A-2 and S-4A-3 for calculating precision of moving die components or parting line shift. Linear tolerance is only for fixed components to allow for growth, shrinkage or minor imperfections in the part.

Tolerance precision is the amount of variation from the part's nominal or design feature.

For example, a 5 inch design specification with ±0.010 tolerance does not require the amount of precision as the same part with a tolerance of ±0.005. The smaller the tolerance number, the more precise the part must be (the higher the precision). Normally, the higher the precision the more it cost to manufacture the part because die wear will affect more precise parts sooner. Production runs will be shorter to allow for increased die maintenance. Therefore the objective is to have as low precision as possible without affecting form, fit and function of the part.



Example: An aluminum casting with a 5.00 in. (127 mm) specification in any dimension shown on the drawing as "E1", can have a Standard Tolerance of ±0.010 inch (±0.25 mm) for the first inch (25.4 mm) plus ±0.001 for each additional inch (plus ±0.025 mm for each additional 25.4 mm). In this example that is ±0.010 for the first inch plus ±0.001 multiplied by the 4 additional inches to yield a total tolerance of ±0.014. In metric terms, ±0.25 for the first 25.4 mm increments plus ±0.025 multiplied by the 4 additional 25.4 mm to yield a total tolerance of ±0.35 mm for the 127 mm design feature specified as "E1" on the drawing. Linear dimension tolerance only applies to linear dimensions formed in the same die half with no moving components.

Table S-4A-1 Tolerances for Linear Dimensions (Standard)

In inches, two-place decimals (.xx); In millimeters, single-place decimals (.x)

Length of Dimension "E_1"	Casting Alloys			
	Zinc	Aluminum	Magnesium	Copper
Basic Tolerance up to 1" (25.4mm)	**±0.010** (±0.25 mm)	**±0.010** (±0.25 mm)	**±0.010** (±0.25 mm)	**±0.014** (±0.36 mm)
Additional Tolerance for each additional inch over 1" (25.4mm)	**±0.001** (±0.025 mm)	**±0.001** (±0.025 mm)	**±0.001** (±0.025 mm)	**±0.003** (±0.076 mm)

Engineering and Design: Coordinate Dimensioning

Parting Line: Standard Tolerances

The values shown represent Standard Tolerances, or normal die casting production practice at the most economical level. For greater casting accuracy see Precision Tolerances for this characteristic on the facing page. See Section 7, "Quality Assurance," for more standard tolerance information.

Die Shift:

Parting line die shift, unlike parting line separation and moving die component tolerances, is a left/right relationship with possible ± consequences. It can shift in four directions, based on a combination of part features, die construction and operation factors. It can occur at any time and its tolerance consequences should be discussed with the die caster at the design stage to minimize any impact on the final die casting.

Notes:

All values for part dimensions which run across the die parting line are stated as a "plus" tolerance only. The die casting die at a die closed position creates the bottom of the tolerance range, i.e., 0.000 (zero). Due to the nature of the die casting process, dies can separate imperceptibly at the parting line and create only a larger, or "plus" side, tolerance.

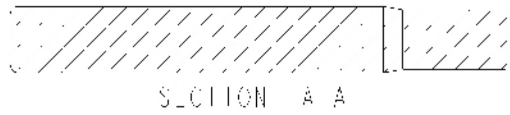
Parting Line Tolerance is the maximum amount of die separation allowed for the end product to meet specified requirements of form, fit and function. This is not to be confused with Parting Line Shift Tolerance which is the maximum amount die halves shift from side to side in relation to one another.

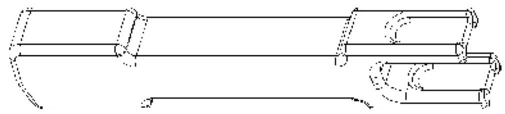
An example of Parting Line Tolerance is the amount a door is allowed to be opened and still call it closed. An example of Parting Line Shift Tolerance is how well the door fits in the frame. Worn hinges may cause the door to drag on the floor so that it can not be closed and therefore not perform its intended function. Parting Line Shift Tolerance will be discussed later in this section.

Parting Line Tolerance is a function of the surface area of the die from which material can flow from one die half to the other. This is also known as Projected Area.

Projected Area is always a plus tolerance since a completely closed die has 0 separation. Excess material and pressure will force the die to open along the parting line plane creating an oversize condition. The excess material will cause the part to be thicker than the ideal specification and that is why Projected Area only has plus tolerance. It is important to understand that Table S-4A-2 (Projected Area Tolerance) does not provide Parting Line Tolerance by itself. Part thickness or depth must be factored in to give a true idea of Parting Line Tolerance. Parting Line Tolerance is a function of part thickness perpendicular to the Projected Area plus the Projected Area Tolerance.

Part thickness includes both die halves to give a volumetric representation of Parting





Line Tolerance. Information from the Projected Area Tolerance table S-4A-2 in combination with the formerly discussed Linear Tolerance table S-4A-1 give a true representation of Parting Line Tolerance. Note that the tolerances in the table apply to a single casting regardless of the number of cavities.

Example: An aluminum die casting has 75 in^2 (483.9 cm2) of Projected Area on the parting die plane. From table S-4A-2, Projected Area Tolerance is +0.012. This is combined with the total part thickness tolerance from table S-4A-1 to obtain the Parting Line Tolerance.

The total part thickness including both die halves is 5.00 in. (127 mm) which is measured perpendicular to the parting die plane (dimension "E2 E1"). From table S-4A-1, the Linear Tolerance is ±0.010 for the first inch and ±0.001 for each of the four additional inches. The Linear Tolerance of ±0.014 inches is combined with the Projected Area Tolerance of +0.012 to yield a Standard Parting Line Tolerance of +0.026/-0.014 in. or in metric terms ±0.35 mm from Linear Tolerance table S-4A-1 plus +0.30 mm from Projected Area Tolerance table S-4A-2 = +0.65/-0.35 mm.

Table S-4A-2 Parting Line Tolerances (Standard) — *Added to Linear Tolerances*

Projected Area of Die Casting inches^2 (cm^2)	**Casting Alloys** *(Tolerances shown are "plus" values only)*			
	Zinc	**Aluminum**	**Magnesium**	**Copper**
up to 10 in^2 (64.5 cm^2)	**+0.0045** (+0.114 mm)	**+0.0055** (+0.14 mm)	**+0.0055** (+0.14 mm)	**+0.008** (+0.20 mm)
11 in^2 to 20 in^2 (71.0 cm^2 to 129.0 cm^2)	**+0.005** (+0.13 mm)	**+0.0065** (+0.165 mm)	**+0.0065** (+0.165 mm)	**+0.009** (+0.23 mm)
21 in^2 to 50 in^2 (135.5 cm^2 to 322.6 cm^2)	**+0.006** (+0.15 mm)	**+0.0075** (+0.19 mm)	**+0.0075** (+0.19 mm)	**+0.010** (+0.25 mm)
51 in^2 to 100 in^2 (329.0 cm^2 to 645.2 cm^2)	**+0.009** (+0.23 mm)	**+0.012** (+0.30 mm)	**+0.012** (+0.30 mm)	—
101 in^2 to 200 in^2 (651.6 cm^2 to 1290.3 cm^2)	**+0.012** (+0.30 mm)	**+0.018** (+0.46 mm)	**+0.018** (+0.46 mm)	—
201 in^2 to 300 in^2 (1296.8 cm^2 to 1935.5 cm^2)	**+0.018** (+0.46 mm)	**+0.024** (+0.61 mm)	**+0.024** (+0.61 mm)	—

For projected area of die casting over 300 in^2 *(1935.5 cm^2)*, **consult with your die caster.**

Engineering and Design: Coordinate Dimensioning

Moving Die Components (MDC): Standard Tolerances

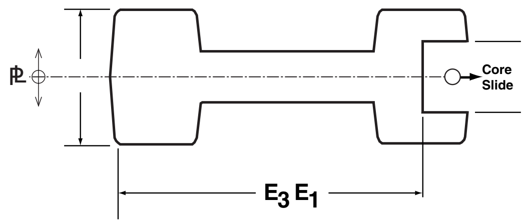
The values shown represent Standard Tolerances, or normal casting production practice at the most economical level. For greater casting accuracy see Precision Tolerances for this characteristic on the facing page. See Section 7, "Quality Assurance," for more standard tolerance information.

Notes:
Moving die components, also called "moving die parts", are most commonly core slides (or pulls) which are used to form inset holes or features in a casting. All values for Projected Area dimensions formed by moving die components are stated as a plus tolerance only. The moving die component at the die closed position is the bottom of the tolerance range, i.e. 0.000 (zero). Due to the nature of the die cast process (parting line separation, wear on moving die components, etc..), a moving die component can back out of a die creating only a larger or plus side tolerance.

Moving Die Components Tolerance can affect final part performance similar to Parting Line Tolerance. When the core is fully inserted into the die, the minimum tolerance is zero. As excess material and pressure are exerted in the die, the core can slide out creating an oversized condition. A MDC Tolerance has been developed to ensure minimal impact on form, fit and function by specifying limits to the oversize condition.

Similar to Parting Line Tolerance, MDC Standard Tolerance is a function of the Projected Area Tolerance plus Linear Tolerance. Linear Tolerance is calculated based on the length of movement of the core slide along dimension "E_3 E_1." Table S-4A-1 is used to determine Linear Tolerance. The linear dimension is not the entire length of "E_3 E_1" but the is only the length of the core slide from where the core slide first engages the die to its full insertion position. Linear dimension is normally perpendicular to the Projected Area.

Projected Area is the area of the core head that faces the molten material. Projected Area Tolerance for moving die components is determined from table S-4A-3. The open area (cavity) on the end view of the part in figure 4A-1B at the beginning of this section shows the projected area. Projected Area Tolerance plus Linear Tolerance provide MDC Standard Tolerance for the volume of the part. Note that the tolerances in the table apply to a single casting regardless of the number of cavities.



Example: An aluminum casting has 75 in^2 (483.9 cm2) of Projected Area calculated from the core slide head facing the molten material. From table S-4A-3, Projected Area Tolerance is +0.024. This is combined with the length of the core slide Linear Tolerance from table S-4A-1 to obtain the MDC Standard Tolerance. The total core slide length of 5.00 in. (127 mm) is measured from where the core engages the part to full insertion in the plane of dimension "E3 E1" to determine Linear Tolerance length. From table S-4A-1, the Linear Tolerance is ±0.010 for the first inch and ±0.001 for each of the four additional inches.

The Linear Tolerance of ±0.014 inches is combined with the Projected Area Tolerance of +0.024 to yield a MDC Standard Tolerance of +0.038/-0.014 in.

MDC Metric Standard Tolerance is *+0.96/-0.35 mm = (±0.35 mm) + (+0.61 mm). on dimensions formed by moving die components.*

Table S-4A-3 MDC Tolerances (Standard) — *Added to Linear Tolerances*

Projected Area of Die Casting inches^2 (cm^2)	Die Casting Alloys *(Tolerances shown are "plus" values only)*			
	Zinc	Aluminum	Magnesium	Copper
up to 10 in^2 (64.5 cm^2)	**+0.006** (+0.15 mm)	**+0.008** (+0.20 mm)	**+0.008** (+0.20 mm)	**+0.012** (+0.305 mm)
11 in^2 to 20 in^2 (71.0 cm^2 to 129.0 cm^2)	**+0.009** (+0.23 mm)	**+0.013** (+0.33 mm)	**+0.013** (+0.33 mm)	—
21 in^2 to 50 in^2 (135.5 cm^2 to 322.6 cm^2)	**+0.013** (+0.33 mm)	**+0.019** (+0.48 mm)	**+0.019** (+0.48 mm)	—
51 in^2 to 100 in^2 (329.0 cm^2 to 645.2 cm^2)	**+0.019** (+0.48 mm)	**+0.024** (+0.61 mm)	**+0.024** (+0.61 mm)	—
101 in^2 to 200 in^2 (651.6 cm^2 to 1290.3 cm^2)	**+0.026** (+0.66 mm)	**+0.032** (+0.81 mm)	**+0.032** (+0.81 mm)	—
201 in^2 to 300 in^2 (1296.8 cm^2 to 1935.5 cm^2)	**+0.032** (+0.81 mm)	**+0.040** (+0.1 mm)	**+0.040** (+0.1 mm)	—

For projected area of die casting over 300 in^2 *(1935.5 cm^2),* **consult with your die caster.**

The formula for draft shown here represents Standard Tolerance, or normal casting production practice at the most economical level. For Precision Tolerance for draft, see the facing page.

Note:
As the formula indicates, draft, expressed as an angle, decreases as the depth of the feature increases. Twice as much draft is recommended for inside walls or surfaces as for outside walls/surfaces. This provision is required because as the alloy solidifies it shrinks onto the die features that form inside surfaces (usually located in the ejector half) and away from features that form outside surfaces (usually located in the cover half). Note also that the resulting draft calculation does not apply to cast lettering, logotypes or engraving. Such elements must be examined individually as to style, size and depth desired. Draft requirements need to be discussed with the die caster prior to die design for satisfactory results.

Engineering and Design: Coordinate Dimensioning

Draft Requirements: Standard Tolerances

Draft is the amount of taper or slope given to cores or other parts of the die cavity to permit easy ejection of the casting.

All die cast surfaces which are normally perpendicular to the parting line of the die require draft (taper) for proper ejection of the casting from the die. This draft requirement, expressed as an angle, is not constant. It will vary with the type of wall or surface specified, the depth of the surface and the alloy selected.

Draft values from the equations at right, using the illustration and the table below, provides Standard Draft Tolerances for draft on inside surfaces, outside surfaces and holes, achievable under normal production conditions.

To achieve lesser draft than normal production allows, Precision Tolerances maybe specified (see opposite page).

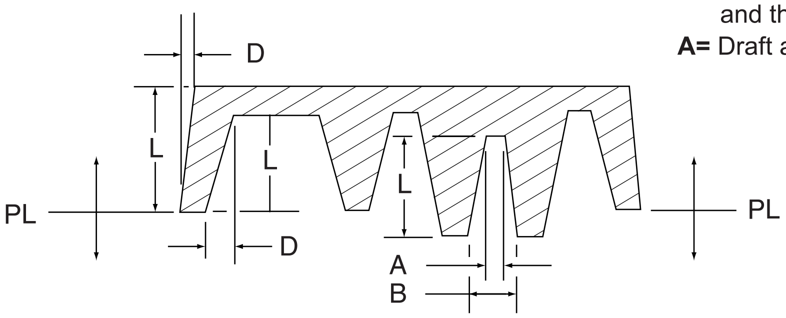
Calculation for Draft

$$D = \frac{\sqrt{L}}{C}$$

Calculation for Draft Angle

$$A = \frac{\left(\frac{D}{L}\right)}{0.01746}$$

Where: **D=** Draft in inches
L= Depth or height of feature from the parting line
C= Constant, from table S-4A-7, is based on the type of feature and the die casting alloy
A= Draft angle in degrees Draft



Drawing defines draft dimensions for interior and exterior surfaces and total draft for holes (draft is exaggerated for illustration).

Table S-4A-7: Draft Constants for Calculating Draft and Draft Angle

Values of Constant "C" by Features and Depth (Standard Tolerances)

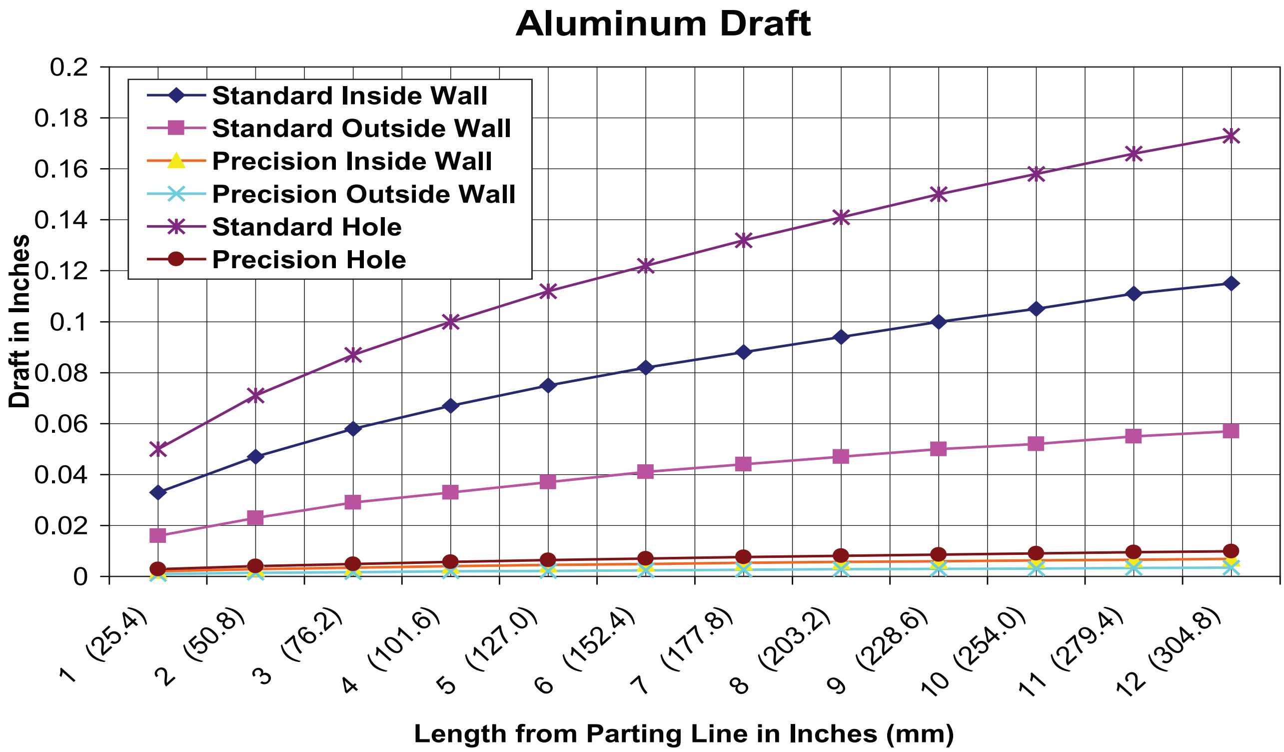
Alloy	**Inside Wall For Dim. in inches** (mm)	**Outside Wall For Dim. in inches** (mm)	**Hole, Total Draft for Dim. in inches** (mm)
Zinc/ZA	**50** (9.90 mm)	**100** (19.80 mm)	**34** (6.75 mm)
Aluminum	**30** (6.00 mm)	**60** (12.00 mm)	**20** (4.68 mm)
Magnesium	**35** (7.00 mm)	**70** (14.00 mm)	**24** (4.76 mm)
Copper	**25** (4.90 mm)	**50** (9.90 mm)	**17** (3.33 mm)

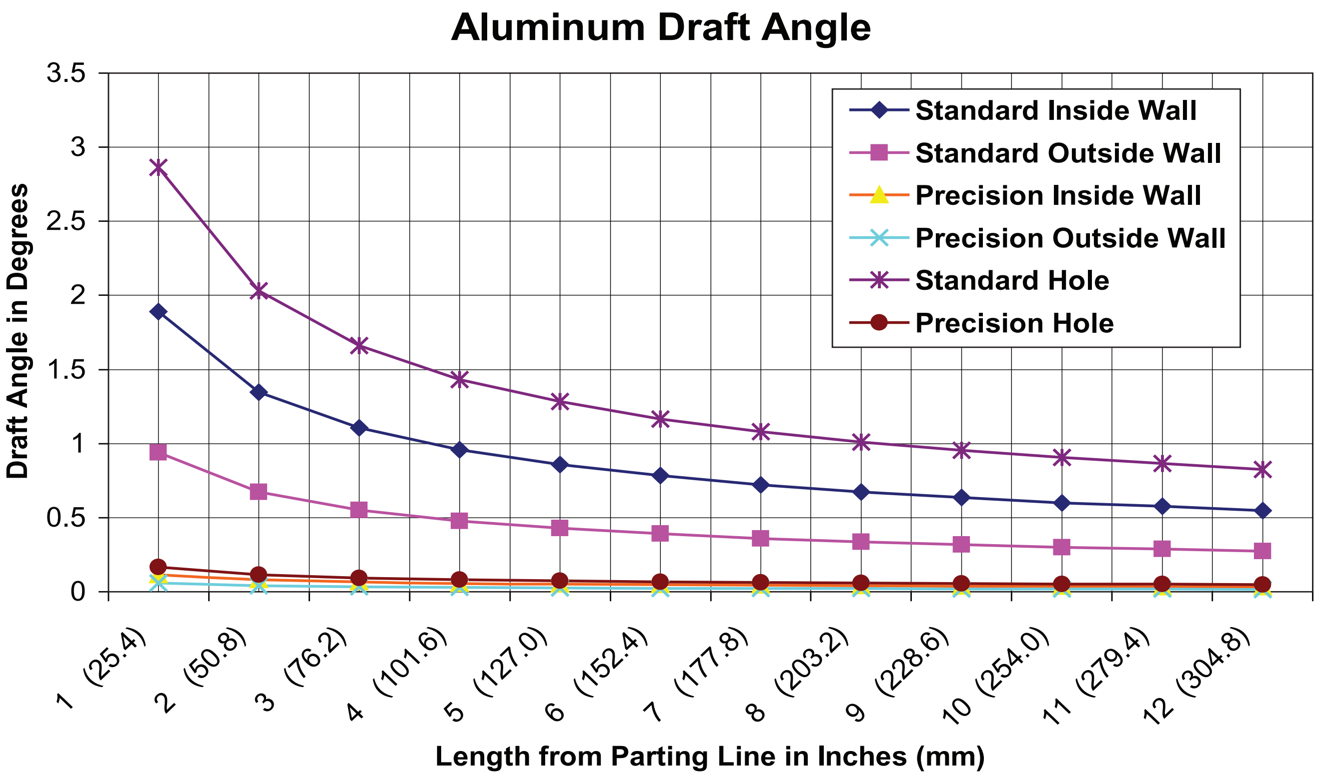
Draft Example (Standard Tolerances):
In the case of an inside surface for an aluminum cast part, for which the constant “C” is 30 (6 mm), the recommended Standard Draft at three depths is:

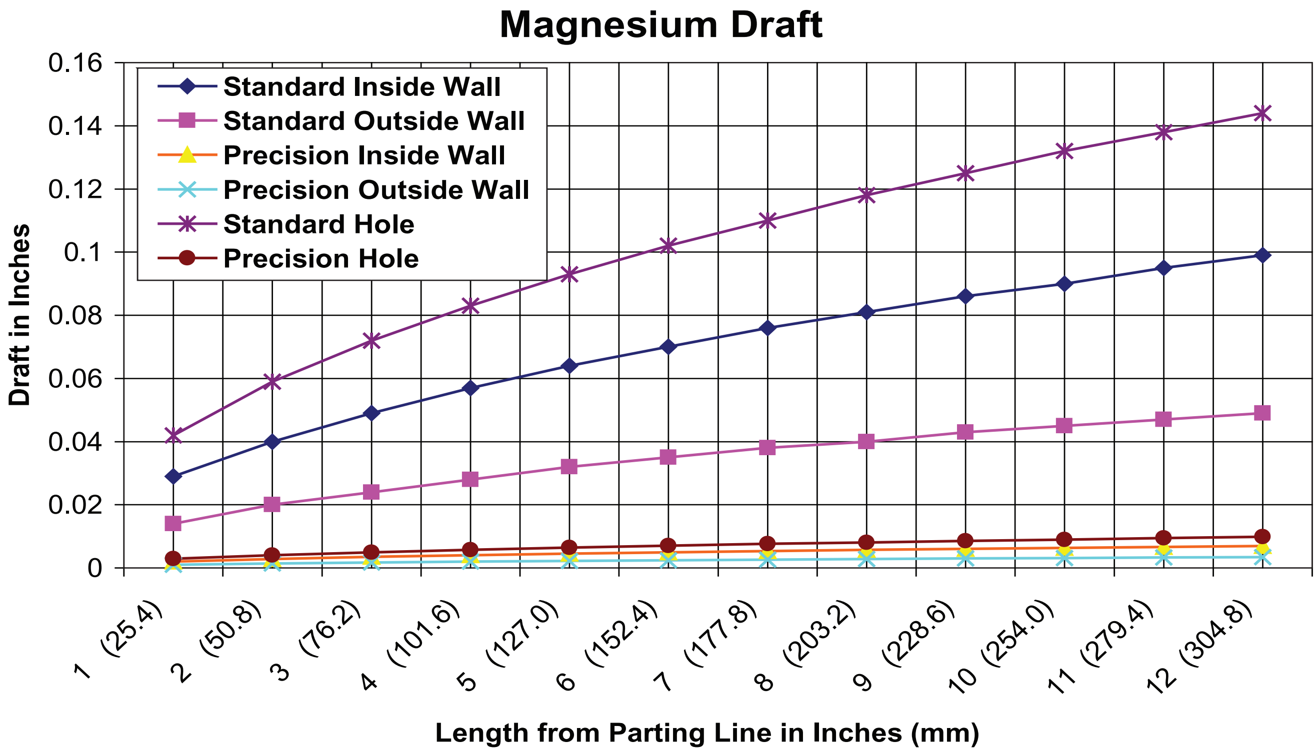
Depth Draft Draft
in. (mm) in. (mm) Degrees
0.1 (2.50) 0.010 (0.250) 6°
1.0 (25) 0.033 (0.840) 1.9°
5.0 (127) 0.075 (1.890) 0.85°

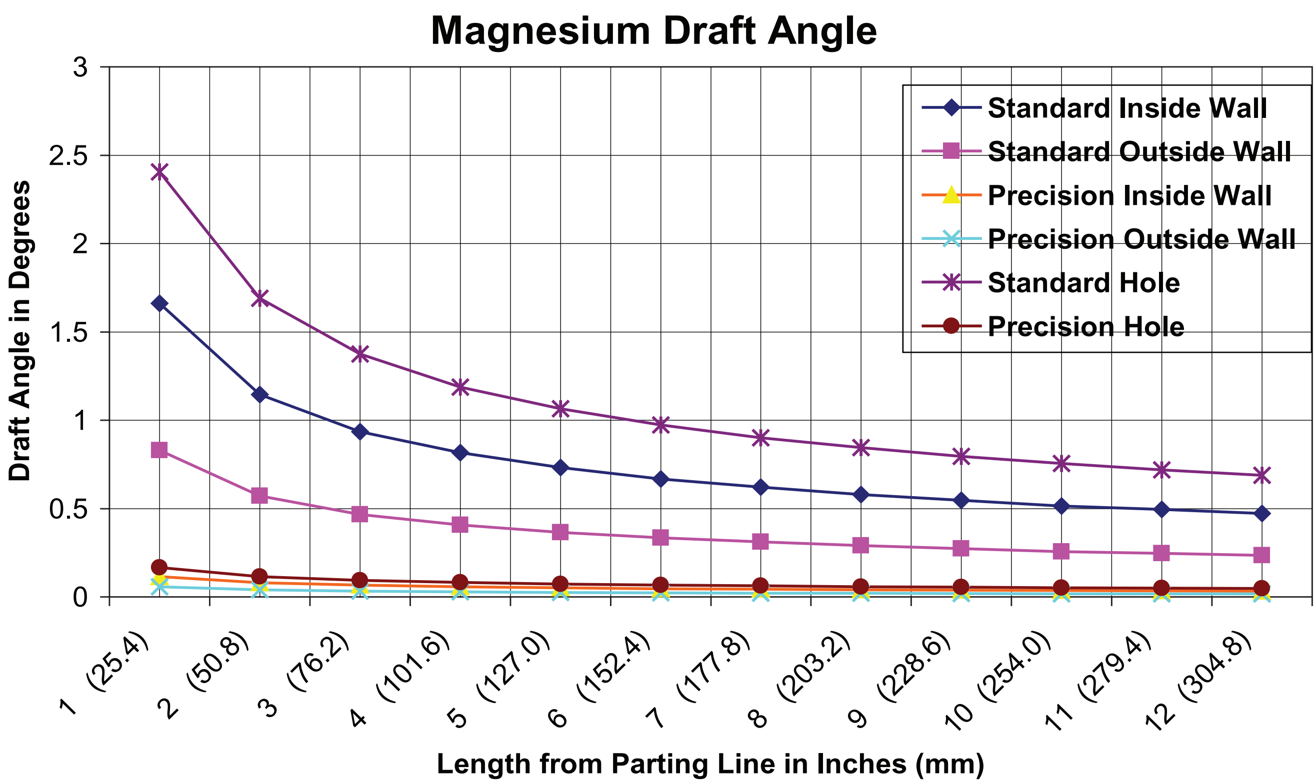
It is not common practice to specify draft separately for each feature. Draft is normally specified by a general note with exceptions called out for individual features. The formula should be used to establish general draft requirements with any exceptions identified.

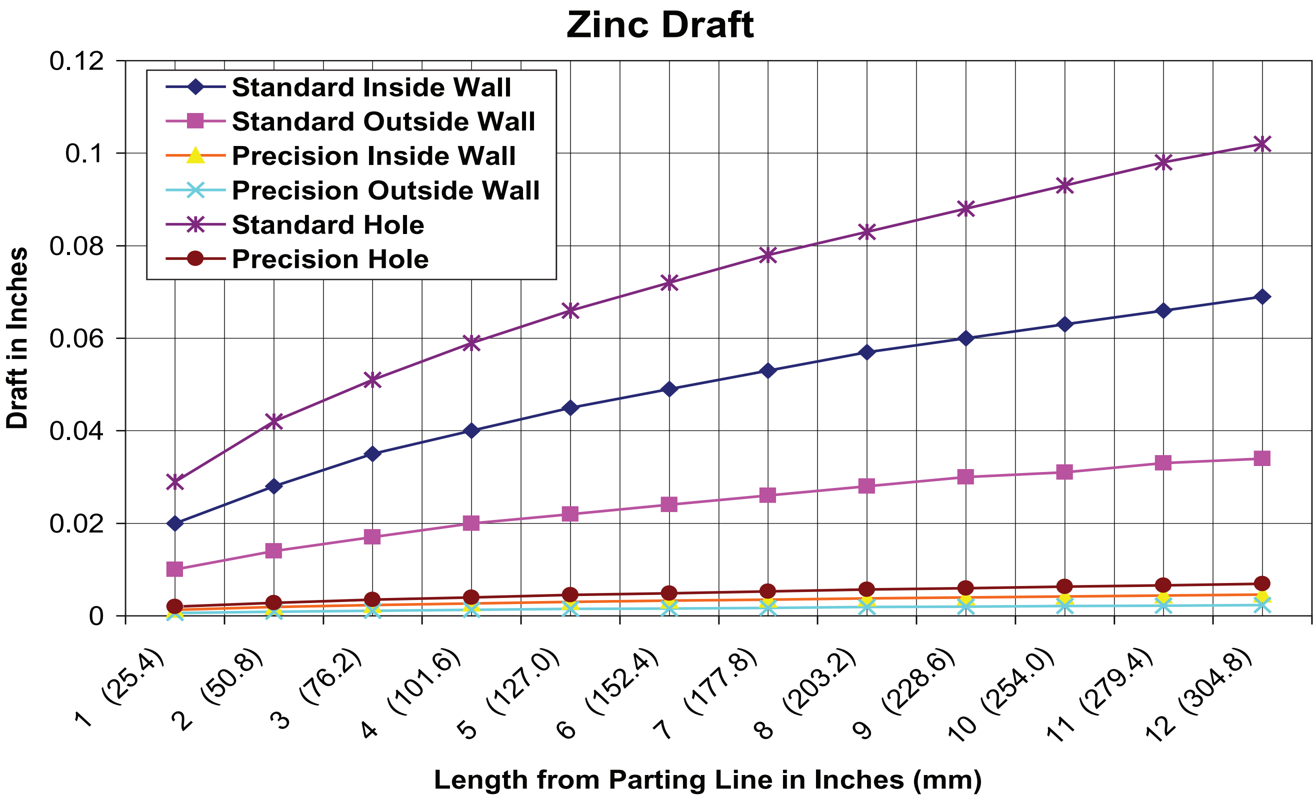
For example, the results at the left indicate that an aluminum casting with most features at least 1.0 in. deep can be covered with a general note indicating 2° minimum draft on inside surfaces and 1° minimum on outside surfaces (based on outside surfaces requiring half as much draft).

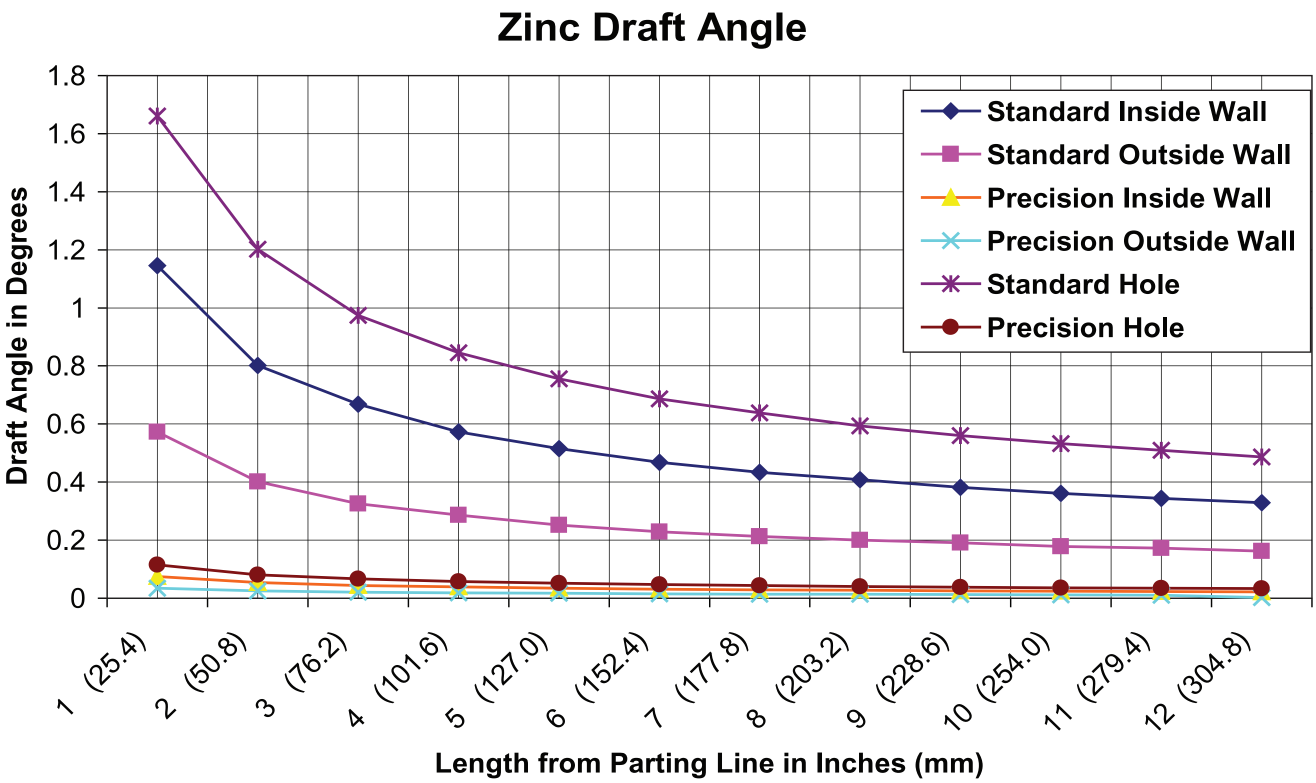

Aluminum Draft
Standard Inside Wall
Standard Outside Wall
Precision Inside Wall
Precision Outside Wall
Standard Hole
Precision Hole
Draft in Inches
0.2
0.18
0.16
0.14
0.12
0.1
0.08
0.06
0.04
0.02
0
1 (25.4)
2 (50.8)
3 (76.2)
4 (101.6)
5 (127.0)
6 (152.4)
7 (177.8)
8 (203.2)
9 (228.6)
10 (254.0)
11 (279.4)
12 (304.8)
Length from Parting Line in Inches (mm)


Aluminum Draft Angle
Standard Inside Wall
Standard Outside Wall
Precision Inside Wall
Precision Outside Wall
Standard Hole
Precision Hole
Draft Angle in Degrees
3.5
3
2.5
2
1.5
1
0.5
0
1 (25.4)
2 (50.8)
3 (76.2)
4 (101.6)
5 (127.0)
6 (152.4)
7 (177.8)
8 (203.2)
9 (228.6)
10 (254.0)
11 (279.4)
12 (304.8)
Length from Parting Line in Inches (mm)


Magnesium Draft
Standard Inside Wall
Standard Outside Wall
Precision Inside Wall
Precision Outside Wall
Standard Hole
Precision Hole
Draft in Inches
0.16
0.14
0.12
0.1
0.08
0.06
0.04
0.02
0
1 (25.4)
2 (50.8)
3 (76.2)
4 (101.6)
5 (127.0)
6 (152.4)
7 (177.8)
8 (203.2)
9 (228.6)
10 (254.0)
11 (279.4)
12 (304.8)
Length from Parting Line in Inches (mm)


Magnesium Draft Angle
Standard Inside Wall
Standard Outside Wall
Precision Inside Wall
Precision Outside Wall
Standard Hole
Precision Hole
Draft Angle in Degrees
3
2.5
2
1.5
1
0.5
0
1 (25.4)
2 (50.8)
3 (76.2)
4 (101.6)
5 (127.0)
6 (152.4)
7 (177.8)
8 (203.2)
9 (228.6)
10 (254.0)
11 (279.4)
12 (304.8)
Length from Parting Line in Inches (mm)


Zinc Draft
Standard Inside Wall
Standard Outside Wall
Precision Inside Wall
Precision Outside Wall
Standard Hole
Precision Hole
Draft in Inches
0.12
0.1
0.08
0.06
0.04
0.02
0
1 (25.4)
2 (50.8)
3 (76.2)
4 (101.6)
5 (127.0)
6 (152.4)
7 (177.8)
8 (203.2)
9 (228.6)
10 (254.0)
11 (279.4)
12 (304.8)
Length from Parting Line in Inches (mm)


Zinc Draft Angle
Standard Inside Wall
Standard Outside Wall
Precision Inside Wall
Precision Outside Wall
Standard Hole
Precision Hole
Draft Angle in Degrees
1.8
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
0
1 (25.4)
2 (50.8)
3 (76.2)
4 (101.6)
5 (127.0)
6 (152.4)
7 (177.8)
8 (203.2)
9 (228.6)
10 (254.0)
11 (279.4)
12 (304.8)
Length from Parting Line in Inches (mm)

Engineering and Design: Coordinate Dimensioning

Flatness Requirements: Standard Tolerance

The flatness values shown here represent Standard Tolerances, or normal casting production practice at the most economical level. For greater casting accuracy see Precision Tolerances for this characteristic on the facing page.

Flatness is described in detail in Section 5, Geometric Dimensioning & Tolerancing. Simply put, Flatness Tolerance is the amount of allowable surface variation between two parallel planes which define the tolerance zone. See the figures below.

Flatness of a continuous plane surface on a casting should be measured by a method mutually agreed upon by the designer, die caster and the customer before the start of die design.

Notes:

The maximum linear dimension is the diameter of a circular surface or the diagonal of a rectangular surface.

Flatness Design Guideline

1. **All draft on walls, bosses and fins surrounding and underneath flat surfaces should be standard draft or greater.**
2. **Large bosses or cross sections can cause sinks and shrinkage distortions and should be avoided directly beneath flat surfaces.**
3. **Changes in cross section should be gradual and well filleted to avoid stress and shrinkage distortions.**
4. **Symmetry is important to obtain flatness. Lobes, legs, bosses and variations in wall height can all affect flatness.**

Flatness defines surface condition not part thickness. See the flatness explanation on the opposite page.

Standard Tolerance is calculated using the largest dimensions defining the area where the tolerance is to be applied. If flatness is to be determined for a circular surface such as the top of a can, the largest dimension is the diameter of the can.

If flatness is to be determined for a rectangular area, the largest dimension is a diagonal.

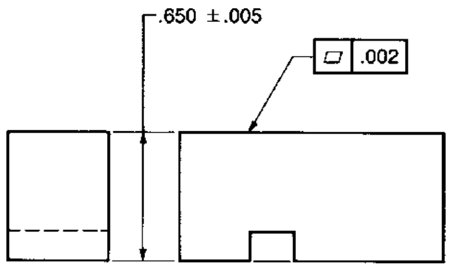
For greater accuracy, see Precision Tolerances for flatness on the opposite page.

Example: Flatness Tolerance - Diagonal
For a part where the diagonal measures 10 inches (254 mm), the maximum Flatness Standard Tolerance from table S-4A-8 is 0.008 inches (0.20 mm) for the first three inches (76.2 mm) plus 0.003 inches (0.08 mm) for each of the additional seven inches for a total Flatness Standard Tolerance of 0.029 inches (0.76 mm).

Table S-4A-8: Flatness Tolerances, As-Cast: All Alloys

Maximum Dimension of Die Cast Surface	Tolerance inches (mm)
up to 3.00 in. (76.20 mm)	**0.008** (0.20 mm)
Additional tolerance, in. (25.4 mm) for each additional in. (25.4 mm)	**0.003** (0.08 mm)

Flatness Example



Explanation