SUNRISE

SUNRISE METAL ALUMINUM DIE CASTING DESIGN GUIDE

Low Volume
High Precision
High Quality
Reliable Partner

Address
No.1456 Fushikang Road,Kunshan.
Jiangsu Province,P.R.China.

Phone
+86-512-55291075
+86 13912651818

E-contact
www.sunrise-metal.com
sales@sunrise-metal.com

Contact us by Whatsapp

CONTENTS

Ultimate Design Guide

For Aluminum Die Casting Parts

A proper design approach can substantially enhance your manufacturing experience. If you are new to the aluminum die casting industry or want to boost your knowledge of casting part designing then this guide is for you.

Mechanical engineers and product designers would find this guide useful. We highlighted the important design factors and limitations that can significantly simplify the aluminum die casting process thus reduce the production cost.

Design Requirement

When designing an aluminum die cast part, you must take into consideration its application, appearance, performance, precision, and most importantly the cost. First, you have to decide what you want to achieve in your parts and balance your requirements to comply with your budget.

Here we have highlighted the major things you should be concerned with when designing a part.

Using Function

When designing a product, you need to consider its possible application. Aluminum die cast parts can serve both structural function and cosmetic function. So, it has become a popular alternative to others.

As modern computers are far more powerful than before, the accuracy attainable by die casting has remarkably increased multifold. Along with its superior structural importance, die cast parts also serves a very good cosmetic function.

You should clearly state the application of your parts to the die caster. He can help you choose the right material and decide proper tolerances for the design parameters based on your requirements. You should also consider properties such as corrosion resistance, strength-to-weight-ratio, conductivity, etc.

But the customers often end up paying for quality and strength that far exceeds their need. So, having a good idea of the using function of your parts will provide you better insight into the die casting process.

Appearance Requirement

The appearance of an internal die aluminum die casting part doesn’t hold much significance. But it is important when it is an exterior casting part such as a housing or casing.

End users will always have an appeal for aesthetic looking products. So, regardless of the performance of a part, consumers will prioritize the looks as well. Thus, external die cast components should have a good appearance.

As a result, you should design your parts keeping in mind the aesthetic aspects. Plan in advance the type of finishing you want to achieve. A good finishing can provide additional protection against extreme weather conditions.

Assembly Method

Assembly of aluminum die cast parts can be relatively simple or highly complicated depending on the complexity of the parts. Conventional die casting equipment had some limitations on what type of parts can be cast. So, casting parts with intricate details was difficult before.

But, complex parts can be divided into suitable segments and then joined together after casting by a suitable assembly method. Some of the common die casting assembly techniques include:

- Fastening
- Threading
- Welding
- Injected Metal Assembly
- Cored Holes, etc.

You must choose a specific assembly technique for your die cast parts before you start designing. Because the design will be heavily influenced by the assembly method. Choose a suitable assembly option that meets your requirement.

Cost Budget

You should run a detailed cost budget analysis of your project. Because every manufacturing business is directly influenced by budget concerns. Your design must be done accordingly to comply with your budget.

An experienced designer can greatly reduce the die casting expenses without compromising the quality of your parts. Certain design parameters should be followed to avoid overdesigning your parts and end up incurring unnecessary costs.

For example, adding pockets allow you to design lighter parts without sacrificing performance. It allows you to cut down material costs. Then reducing or eliminating undercuts and sharp corners can greatly reduce the cost and difficulty of tooling and casting.

Product Structure Design

Properly designing an aluminum die cast part comes with many challenges. Even the smallest features in a design can have a great impact on the casting operation. So, each of the details should be designed with proper care according to the recommended guidelines.

We have focused on the major features present in aluminum die casting design. You will find recommended precision for many important features and learn about the design considerations you should be following during product design.

Material Selection

The product design can significantly vary depending upon your material choice. There will be certain limitations imposed for each type of alloy. Optimum integrity and strength of your aluminum die cast parts require careful design and execution.

Depending upon the composition of alloying elements used with aluminum, the properties such as weight, fluidity, strength, conductivity, melting point, etc. can vary. But not all of them are suitable as a die casting material.

Some of the popular aluminum alloys you can use for die casting include,

- A380
- A383 (ADC12)
- A413

There are numerous other aluminum alloys available as well. You have to choose the right aluminum alloy based on your requirements and budget constraints.

Aluminum Die Casting Alloys

Commercial: ANSI/AA	360 360.0	A360 A360.0	380 380.0	A380 Ⓔ Ⓕ A380.0	383 Ⓔ 383.0	384 384.0	390* B390.0	13 413.0	A13 A413.0	43 C443.0	218 518.0
Mechanical Properties											
Ultimate Tensile Strength ksi	44	46	46	47	45	48	46	43	42	33	45
(MPa)	(303)	(317)	(317)	(324)	(310)	(330)	(317)	(300)	(290)	(228)	(310)
Yield Strength Ⓐ ksi	25	24	23	23	22	24	36	21	19	14	28
(MPa)	(170)	(170)	(160)	(160)	(150)	(165)	(250)	(140)	(130)	(97)	(193)
Elongation % in 2in. (51mm)	2.5	3.5	3.5	3.5	3.5	2.5	<1	2.5	3.5	9.0	5.0
Hardness Ⓑ BHN	75	75	80	80	75	85	120	80	80	65	80
Shear Strength ksi	28	26	28	27	–	29	–	25	25	19	29
(MPa)	(190)	(180)	(190)	(190)		(200)		(170)	(170)	(130)	(200)
Impact Strength ft-lb	–	–	3	–	3 Ⓓ	–	–	–	–	–	7
(J)			(4)		(4)						(9)
Fatigue Strength Ⓒ ksi	20	18	20	20	21	20	20	19	19	17	20
(MPa)	(140)	(120)	(140)	(140)	(145)	(140)	(140)	(130)	(130)	(120)	(140)
Young's Modulus psi x 10^6	10.3	10.3	10.3	10.3	10.3	–	11.8	10.3	–	10.3	–
(GPa)	(71)	(71)	(71)	(71)	(71)		(81.3)	(71)		(71)	
Physical Properties											
Density lb/in³	0.095	0.095	0.099	0.098	0.099	0.102	0.098	0.096	0.096	0.097	0.093
(g/cm³)	(2.63)	(2.63)	(2.74)	(2.71)	(2.74)	(2.82)	(2.71)	(2.66)	(2.66)	(2.69)	(2.57)
Melting Range °F	1035-1105	1035-1105	1000-1100	1000-1100	960-1080	960-1080	950-1200	1065-1080	1065-1080	1065-1170	995-1150
(°C)	(557-596)	(557-596)	(540-595)	(540-595)	(516-582)	(516-582)	(510-650)	(574-582)	(574-582)	(574-632)	(535-621)
Specific Heat BTU/lb °F	0.230	0.230	0.230	0.230	0.230	–	–	0.230	0.230	0.230	–
(J/kg °C)	(963)	(963)	(963)	(963)	(963)			(963)	(963)	(963)	
Coefficient of Thermal Expansion µ in/in°F	11.6	11.6	12.2	12.1	11.7	11.6	10.0	11.3	11.9	12.2	13.4
(µ m/m°K)	(21.0)	(21.0)	(22.0)	(21.8)	(21.1)	(21.0)	(18.0)	(20.4)	(21.6)	(22.0)	(24.1)
Thermal Conductivity BTU/ft hr°F	65.3	65.3	55.6	55.6	55.6	55.6	77.4	70.1	70.1	82.2	55.6
(W/m °K)	(113)	(113)	(96.2)	(96.2)	(96.2)	(96.2)	(134)	(121)	(121)	(142)	(96.2)
Electrical Conductivity % IACS	30	29	27	23	23	22	27	31	31	37	24
Poisson's Ratio	0.33	0.33	0.33	0.33	0.33	–	–	–	–	0.33	–

Aluminum Die Casting Alloy-Typical material properties

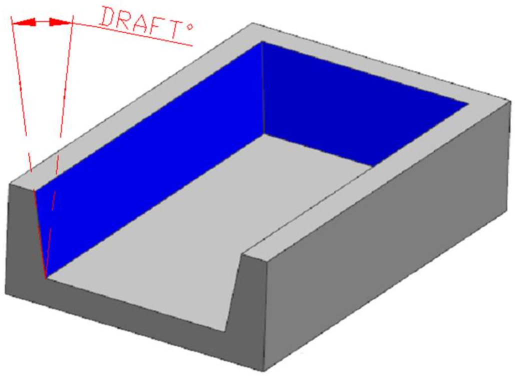
Aluminum Die Casting Alloys

Commercial: ANSI/AA	360 360.0	A360 A360.0	380 380.0	A380 A380.0	383 383.0	384 384.0	390* B390.0	13 413.0	A13 A413.0	43 C443.0	218 518.0
Resistance to Hot Cracking Ⓐ	1	1	2	2	1	2	4	1	1	3	5
Pressure Tightness	2	2	2	2	2	2	4	1	1	3	5
Die-Filling Capacity Ⓑ	3	3	2	2	1	1	1	1	1	4	5
Anti-Soldering to the Die Ⓒ	2	2	1	1	2	2	2	1	1	4	5
Corrosion Resistance Ⓓ	2	2	4	4	3	5	3	2	2	2	1
Machining Ease & Quality Ⓔ	3	3	3	3	2	3	5	4	4	5	3
Polishing Ease & Quality Ⓕ	3	3	3	3	3	3	5	5	5	4	1
Electroplating Ease & Quality Ⓖ	2	2	1	1	1	2	3	3	3	2	5
Anodizing (Appearance) Ⓗ	3	3	3	3	3	4	5	5	5	2	1
Chemical Oxide Protective Coating Ⓘ	3	3	4	4	4	5	5	3	3	2	1
Strength at Elevated Temp. Ⓙ	1	1	3	3	2	2	3	3	3	5	4

Aluminum Die Casting Alloy-Other characteristics

Draft

The draft is one of the most important design parameters for aluminum die casting. It is the tapering or inclination provided to the cores and surfaces of a part that are perpendicular to the parting line of the die. It is also referred to as a draft angle.



Draft in an Aluminum Die Casting Part's Design

A designer must provide a sufficient draft wherever it's necessary. Because without an adequate draft, the casting will be hard to eject after solidifying and there remains a possibility to damage the part or even the die itself.

Design Considerations for Draft

The following considerations are taken when calculating draft requirements for parts,

- Normally a common draft angle is adopted for most of the geometric features.
- Some exception is applicable for internal walls and surfaces. The draft is normally doubled than the outer walls in this case.
- The draft requirement can vary depending on the alloy used for casting as well. You may have to calculate the draft according to your choice of aluminum alloy.

The standard tolerances of the draft for an inside surface of an aluminum cast part at different depths are shown below as an example.

Depth	Draft Distance	Draft Angle
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°

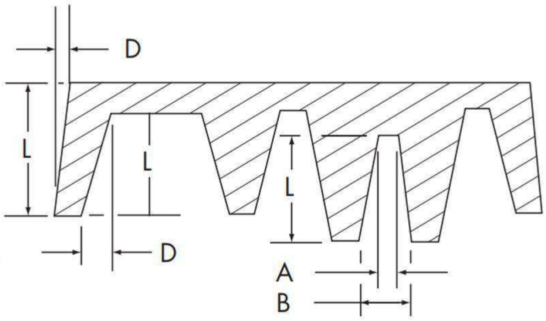
The standard tolerances for any alloy can be calculated using the following equation.

Calculation for Draft Distance

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

Calculation for Draft Angle

$$A = \frac{\left(\frac{D}{L}\right)}{0.01746} \quad \text{or} \quad \frac{57.2738}{C\sqrt{L}}$$



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

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)

Equation for Calculating Standard Tolerances of Draft

If you want to achieve a smaller draft then precision tolerances can be used. But they will involve more precise machining and will be costlier. So, it is advised to avoid precision tolerances unless necessary.

The precision tolerances of the draft for an inside surface of an aluminum cast part at different depths are shown below.

Depth	Draft Distance	Draft Angle
in. (mm)	**in. (mm)**	**Degrees**
0.1 (2.50)	0.006 (0.150)	3.6°
1.0 (25)	0.020 (0.510)	1.1°
2.5 (63.50)	0.032 (1.140)	0.72°

Here's is the equation for calculating the precision tolerance of a part.

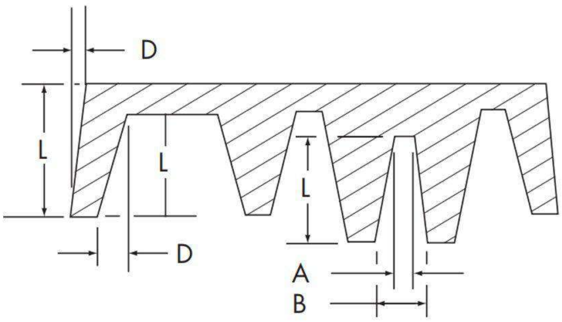
Calculation for Draft Distance

$$D = \frac{\sqrt{L} \times 0.8}{C}$$

Calculation for Draft Angle

$$A = \frac{\left(\frac{D}{L}\right)}{0.01746} \quad \text{or} \quad \frac{45.819}{C\sqrt{L}}$$

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



Values of Constant "C" by Features and Depth (Precision 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	**60** (12.00 mm)	**120** (24.00 mm)	**40** (7.80 mm)
Al/Mg/Cu	**40** (7.80 mm)	**80** (15.60 mm)	**28** (5.30 mm)

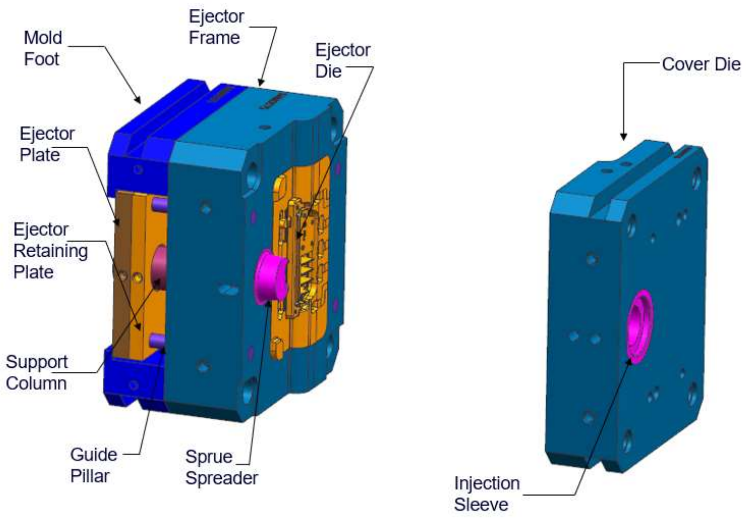
Equation for Calculating Precision Tolerances of Draft

Do note that the above-drawn representation of the draft is a bit exaggerated to give you a better understanding of the concept. The draft is very small in reality and often not even noticeable without careful observation.

Moving Die & Fixed Die

It is critically important to design the moving and fixed die in a harmony. Inconsistency in even one of them can hamper the aluminum die casting process. While we are at it, the moving die design usually comes with more challenges.

The fixed die has a relatively simpler construction. But the moving die has more components to worry about. When the material is injected into the die, the core can slide out and create an oversized condition due to the pressure exerted by the excess material.



Components of a Typical Moving and Fixed Die

The moving die components tolerance is a function of the Linear Tolerance and Projected Area tolerance. Here the linear tolerance is the length of the core slide and the projected area is the head of the core slide facing the molten material.

The shifting can take place along a linear direction perpendicular to the projected area. So, it is desirable to keep a minimum tolerance of zero for the moving die components.

Due to the construction of the die casting equipment, only a larger or positive tolerance is possible during the process. The standard and precision tolerances of a die design according to some variable Projected Area are shown below.

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)	—

Standard Tolerances for Moving Die Component

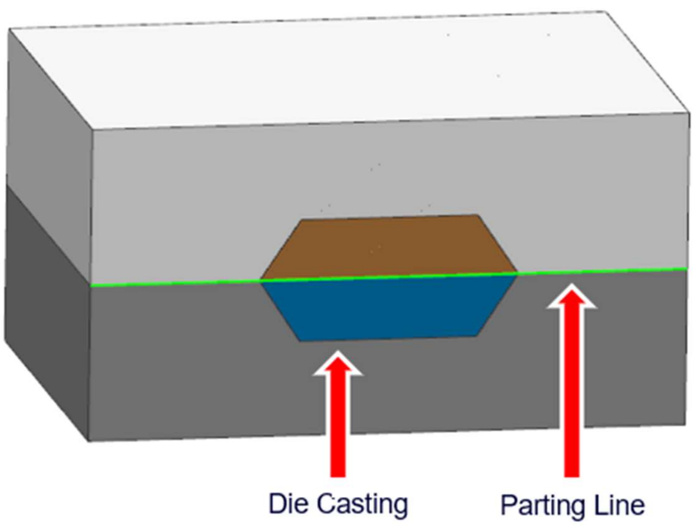
Table P-4A-3 MDC Tolerances (Precision) – Added to Linear Tolerances

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)	+0.005 Ⓐ (+0.127 mm)	+0.006 (+0.152 mm)	+0.005 (+0.127 mm)	+0.010 (+0.254 mm)
11 in^2 to 20 in^2 (71.0 cm^2 to 129.0 cm^2)	+0.007 (+0.178 mm)	+0.010 (+0.254 mm)	+0.007 (+0.178 mm)	—
21 in^2 to 50 in^2 (135.5 cm^2 to 322.6 cm^2)	+0.010 (+0.254 mm)	+0.014 (+0.356 mm)	+0.010 (+0.254 mm)	—
51 in^2 to 100 in^2 (329.0 cm^2 to 645.2 cm^2)	+0.014 (+0.356 mm)	+0.018 (+0.457 mm)	+0.014 (+0.356 mm)	—
101 in^2 to 200 in^2 (651.6 cm^2 to 1290.3 cm^2)	+0.019 (+0.483 mm)	+0.024 (+0.61 mm)	+0.019 (+0.483 mm)	—
201 in^2 to 300 in^2 (1296.8 cm^2 to 1935.5 cm^2)	+0.024 (+0.61 mm)	+0.030 (+0.762 mm)	+0.024 (+0.61 mm)	—

Precision Tolerances for Moving Die Component

Parting Line

The Parting Line is the location along which two die halves meet together to form the full product structure. The formation of the parting line is inevitable due to the way die casting is done. Because the design always consists of a minimum of two parts.



Die Casting Parting Line Location

The parting line is a clear indication to distinguish between the moving half and fixed half of a die. The Parting Line Tolerance refers to the maximum amount of die separation allowed to ensure proper execution of the aluminum die casting process.

When the material pressure is trying to force the die halves apart, the material will flow out from separation created along the parting line. This is known as the flash defect of die casting. The cast parts require an additional trimming process to remove the flash, runner, gate, and overflow.

Before and After Trimming Flash of an Aluminum Die Cast Part

The parting line tolerance is a function of the Projected Area of the die, which represents the separating surface where the molten material moves from one die half to another.

A completely closed die has zero separation from each other, so the projected area tolerance always has a plus value. The extent of die separation is dependent upon the die casing pressure and the extent of clamping force applied to keep the die halves together.

The parting line tolerance can vary depending on the alloy, size, and depth of the parts. The recommended standard and precision tolerance values for die casting parting lines are given below.

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)	—

Standard Tolerances for Parting Line

Table P-4A-2 Parting Line Tolerances (Precision) — Added to Linear Tolerances

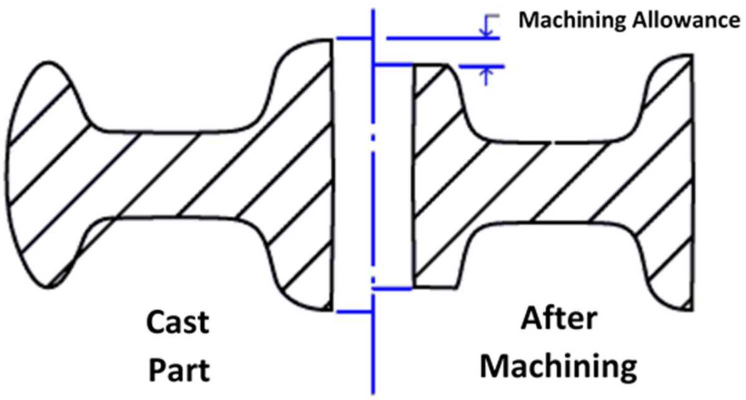
Projected Area of Die Casting inches^2 (cm^2)	Die Casting Alloys (Tolerances shown are "plus" values only)			
	Zinc	Aluminum	Magnesium	Copper
up to 10 in^2 (64.5 cm^2)	+0.003 Ⓐ (+0.076 mm)	+0.0035 (+0.089 mm)	+0.0035 (+0.089 mm)	+0.008 (+0.20 mm)
11 in^2 to 20 in^2 (71.0 cm^2 to 129.0 cm^2)	+0.0035 (+0.089 mm)	+0.004 (+0.102 mm)	+0.004 (+0.102 mm)	+0.009 (+0.23 mm)
21 in^2 to 50 in^2 (135.5 cm^2 to 322.6 cm^2)	+0.004 (+0.102 mm)	+0.005 (+0.153 mm)	+0.005 (+0.153 mm)	+0.010 (+0.25 mm)
51 in^2 to 100 in^2 (329.0 cm^2 to 645.2 cm^2)	+0.006 (+0.153 mm)	+0.008 (+0.203 mm)	+0.008 (+0.203 mm)	—
101 in^2 to 200 in^2 (651.6 cm^2 to 1290.3 cm^2)	+0.008 (+0.203 mm)	+0.012 (+0.305 mm)	+0.012 (+0.305 mm)	—
201 in^2 to 300 in^2 (1296.8 cm^2 to 1935.5 cm^2)	+0.012 (+0.305 mm)	+0.016 (+0.406 mm)	+0.016 (+0.406 mm)	—

Precision Tolerances for Parting Line

However, consult with your die caster if the projected area of die casting is over 300 in^2 (1935.5 cm^2).

Machining Allowance

Machining Allowance is the extent of stock material that can be removed from a finished aluminum die cast part. A cast part may have surface roughness and geometric deviations from the actual design to some extent. So, secondary machining is required after the die casting process to correct these errors.



Machining Allowance of an Aluminum Casting

An important matter to be addressed is that the optimum mechanical properties and density of casting are at or close to the surface. So, machining allowance should be carefully determined so as not to penetrate the less dense portion.

However, a certain Machining Allowance must be specified for the machining and casting variables during the design stage. Leaving a small machining allowance may not be able to meet the surface quality requirement and risks leaving defects in the parts.

On the other hand, an unnecessarily large machining allowance for a part will increase the time, labor, and cost of production. Consulting your die casting supplier in advance will help you decide on a proper machining allowance.

Usually, the minimum machining allowance is taken as 0.010 in. (0.25 mm) to reduce tool wearing and minimize porosity in casting. The maximum allowance is the sum of this minimum and the casting deformation.

Here is a comparative example of the machining allowance for two different datum point locations.

Machining Stock Allowance Comparative Example: Precision Tolerances

	Example A Datum Points In Same Die Half	Example B Datum Points In Opposite Die Half
Minimum Machine Stock Allowance inches (mm)	0.010 (0.25 mm)	0.010 (0.25 mm)
Machining Allowances (± 0.001 in. or ± 0.026 mm)	0.002 (0.05 mm)	0.002 (0.05 mm)
Linear Casting Allowance on 5.000 in. (127 mm) Dimension Precision Tolerance Ⓐ	0.012 (0.356 mm)	0.012 (0.356 mm)
Across Parting Line Precision Tolerances Ⓑ	–	0.008 (0.020 mm)
Maximum Stock	0.026 (0.56 mm)	0.034 (0.86 mm)
Casting Dimension Ⓒ	5.017 ± 0.006 (127.45 ± 0.18 mm)	5.026 +0.014/-0.006 (127.66 +0.38/-0.18 mm)

Ⓐ ±0.007 (±0.18 mm) P-4A-1-03 Precision Tolerance
Ⓑ ±0.008 (±0.20 mm) P-4A-2 Precision Tolerance
Ⓒ Casting dimension would not be needed if drawing was a combined drawing, only finish dimension of 5.00 ± 0.001 in. (127 ± 0.025 mm) would be needed.

Precision Tolerance Values for Machining Allowance

However, some additional consideration is needed for flat and large parts. You can consult with your caster to assure the machining allowance values in this case.

Wall Thickness

Always try to keep a uniform wall thickness throughout the part. Because uniform thickness allows better metal flow and solidification. So, casting quality and integrity are much better.

However, if you must provide a variable wall thickness to your design, you should introduce a gradual transition in the form of a fillet/radii instead of abruptly changing the thickness. Otherwise, you will leave sharp edges in your design.

It is not desirable to have any sharp edges in product design. Because it will affect the metal flow and cause difficulty in ejection after casting. However, you can leave the edges as it is if the walls meet at the parting line.

Recommended Wall Thickness

While there are no absolute values for how thick or thin you should make the walls, it is wise to keep it within a limit. The typical wall thicknesses for aluminum die casting design can range from 0.787 in. (2.0mm) to 0.1737 in. (3.5mm). It also depends on the part's size and structure.

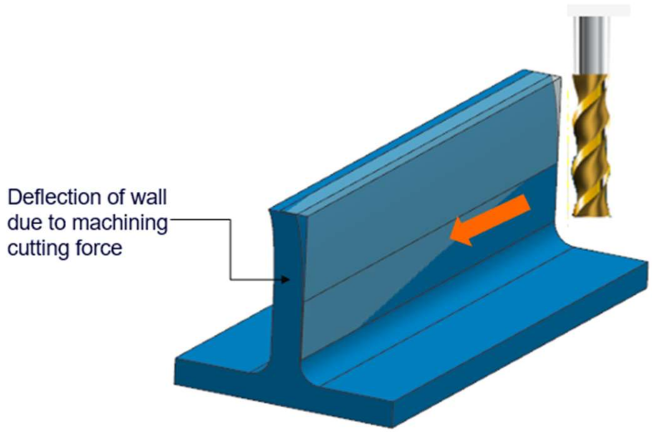
But this is subject to change depending on the alloy, part configuration, part size, and application of your die casting parts. For instance, if the part size is smaller then you can cast walls sections as thin as 0.020 in. (0.50 mm).

However, there may be an exception to the maximum and minimum wall thickness for small and large aluminum die cast parts. You can consult with your die caster or Sunrise Metal if you are having trouble with it.

Avoid Too Thick and Thin Wall

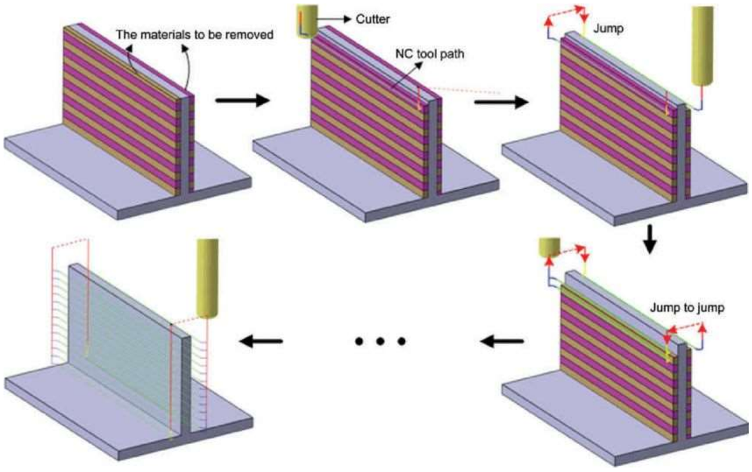
Thicker walls will increase the stiffness of your parts. But making them too thick will delay the cooling thus hampering the solidification process. So, this can result in poor casting quality unless proper measures are taken.

Thick walls also add extra weight to your product. So, product designers with a focus on making the parts lighter will prefer thin walls. But, if the walls are made thinner beyond a certain limit, the stiffness will be too low and it will be prone to warping when subjected to further machining.



Warping of a Thin Wall Section Due to Machining Stress

The warping issue can be dealt with by machining step by step. But thin walls in a cast part lack stiffness and strength. Providing ribs will substantially improve the thin wall's stiffness and make it more stable.



Steps to Machine a Thin Wall Section

However, modern die casting technologies are advanced enough to deal with most of the critical design parameters. But you should only consider them if it will ensure better performance or economy for your parts.

Lightweight Design

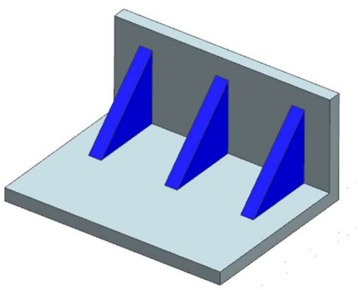
Metal Savers and Pockets are two commonly adopted design features for lightweight part design. These can significantly reduce the volume of material needed for producing a part without affecting the integrity and strength.

Metal savers are hollow spaces that are usually provided in the ribs to reduce the amount of material used thus making the parts lighter as well. The portion between the ribs doesn't serve much purpose, so it can be safely removed from your design.

Design Recommendation for Metal Savers

One should keep these following things in mind when designing a metal saver for a part.

- Avoid sharp edges along with the metal saver, use fillets/radii with a radius as large as possible. Consider a minimum radius of 0.06 inch (1.524mm).
- Keep a uniform wall thickness around the metal saver. Try to keep the thickness close to the typically recommended value.
- Provide a draft angle as larger as possible.



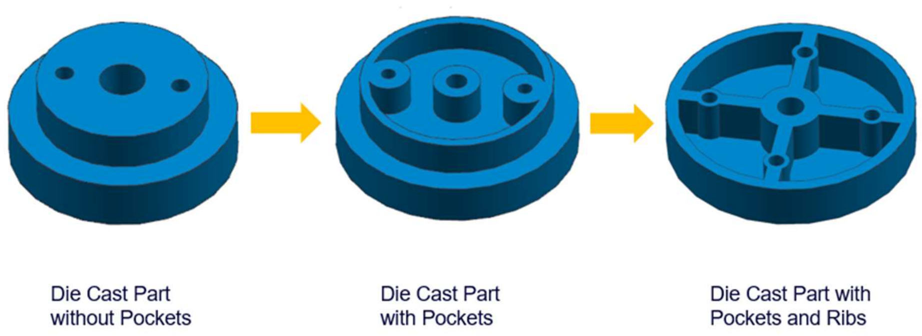
Ribs designed with Metal Savers

Pockets can Considerably Reduce the Weight

Pockets are another weight reduction technique used by designers. Thicker sections with holes can be replaced with thin-walled sections to reduce the amount of material needed for production. However, pockets can sometimes cause irregular shrinkage.

So, you should carefully decide where to use pockets. You can strengthen the pocket features with ribs as well. Doing so will add more stiffness to it and also allow better metal flow. A reduced amount of metal will also increase the cooling rate thus boosting the production cycle.

Here are some figures showing pockets in a part with and without ribs added to them.



Pockets Used in Parts for Weight Reduction

Fillets & Radii

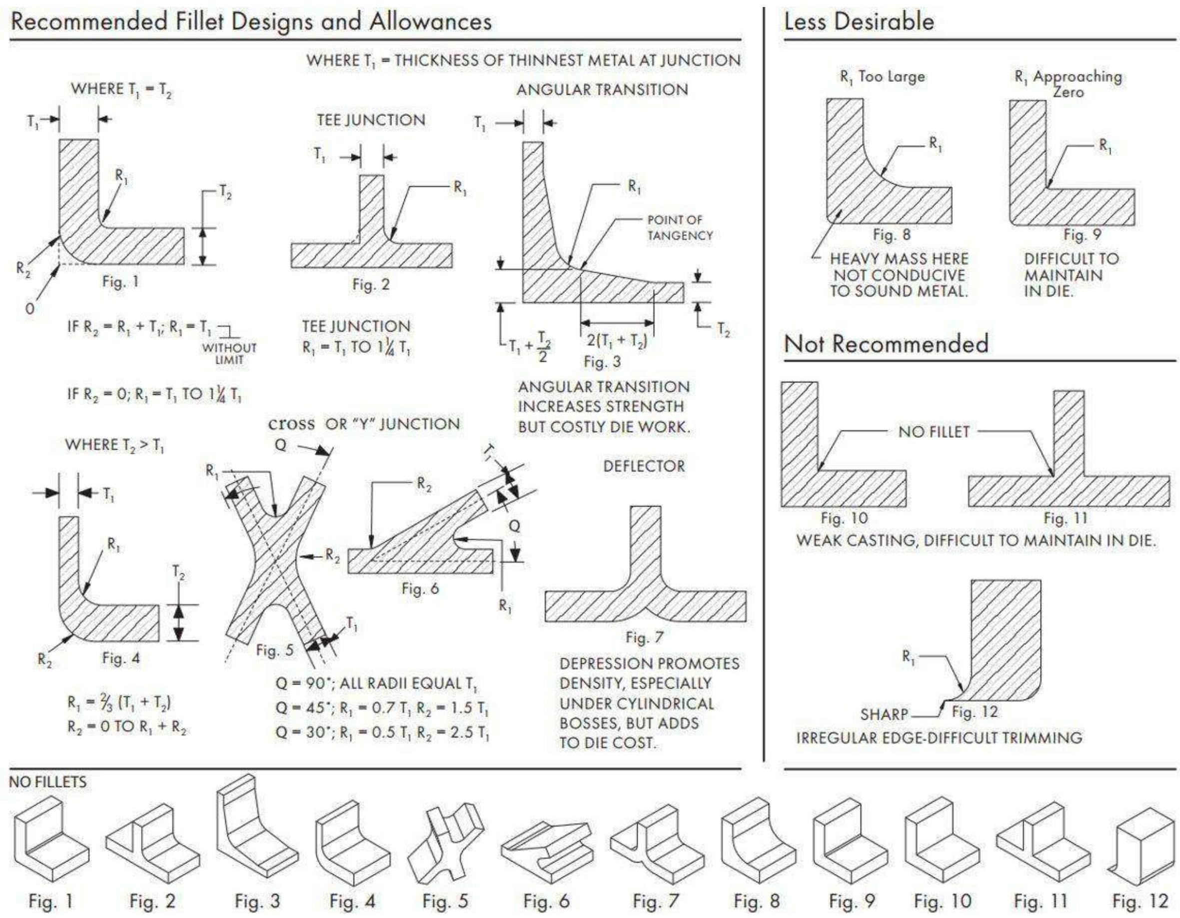
Despite common belief, fillets and radii are not the same things. While both of them refer to the rounded edges of aluminum die cast part design, the rounded inside corners are called fillets and the rounded outer edges are called radii.

Fillets and Radii are an extremely important feature for any aluminum die cast part design. They can significantly reduce the turbulence created during a metal injection and ensures smoother metal flow. So, the parts can attain better structural integrity.

Recommendations for Designing Fillet/Radii

- Two intersecting surfaces will have sharp corners and it is best to join them using fillet/radii. It will not allow high-stress concentration in that part of the die or the part.
- Fillet/Radii is not needed for any edges or corners that are along the parting line of the die.
- Provide adequate draft for fillets that are perpendicular to the parting line.

Fillet/Radii in a part can be designed according to the following guidelines.



Fillet/Radii Design Guidelines for a Die Casting Part

Shrinkage

Shrinkage is a very common and unavoidable phenomenon in aluminum die casting. Any metal alloy will undergo some extent of shrinkage when the molten metal starts to cool down and solidify. So, necessary adjustments must be made to the product design to allow room for shrinkage.

Porosity formed by Shrinkage of an Aluminum *Die Cast Part*

Thicker sections are prone to shrinkage and cause internal pores to form. Local overheating also cause shrinkage to take place which in turn results in porosity. Such spots need to be locally cooled by improvising the die design. But it may increase the casting cycle time.

Tips to Reduce Shrinkage in Castings

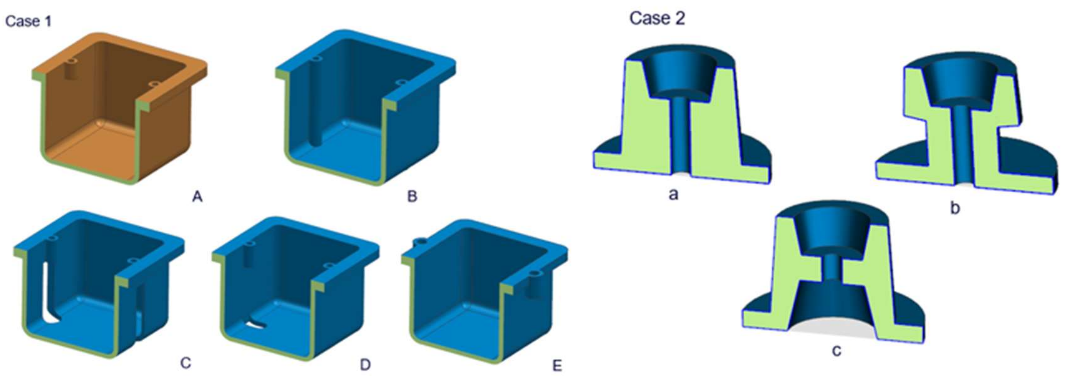
The designer should abide by the following design considerations to reduce shrinkage in aluminum die cast parts.

- Avoid large/thick sections in your design. Redesign them with thinner sections and metal saving cores if possible.
- Adding flat or vertical ribs to the walls can improve the feeding characteristics and reduce the tendency to shrink.
- Shrinkage porosity in the localized area can be reduced by adding squeeze pins.

Boss

Bosses are required for parts that will be mounted elsewhere. They function as stand-offs and mounting points. But improper design and positioning of bosses can lead to manufacturing difficulties which will in turn increase the cost.

Bosses can also increase the material requirement and increase the weight of the aluminum casting. Bosses can be redesigned in the following manner for obtaining lighter parts.



Cases Showing Alternative Boss Redesign Techniques

Design Considerations for Boss Design

The following design considerations should be adopted for including bosses in a part design.

- Try to add a hole to the center of the bosses to obtain a uniform wall thickness if a hole is required.
- Bosses must be given large fillets to allow the proper flow of the molten metal into it.
- Adding ribs is also recommended as it can help the boss be filled well and also provides additional strength to the bosses.
- A sufficient draft should be provided to the bosses to allow easier ejection of the casting.

Here is a video with a short description of bosses explaining its purpose and design process in a die casting.

https://www.youtube.com/watch?v=7XZKQwwz0_U

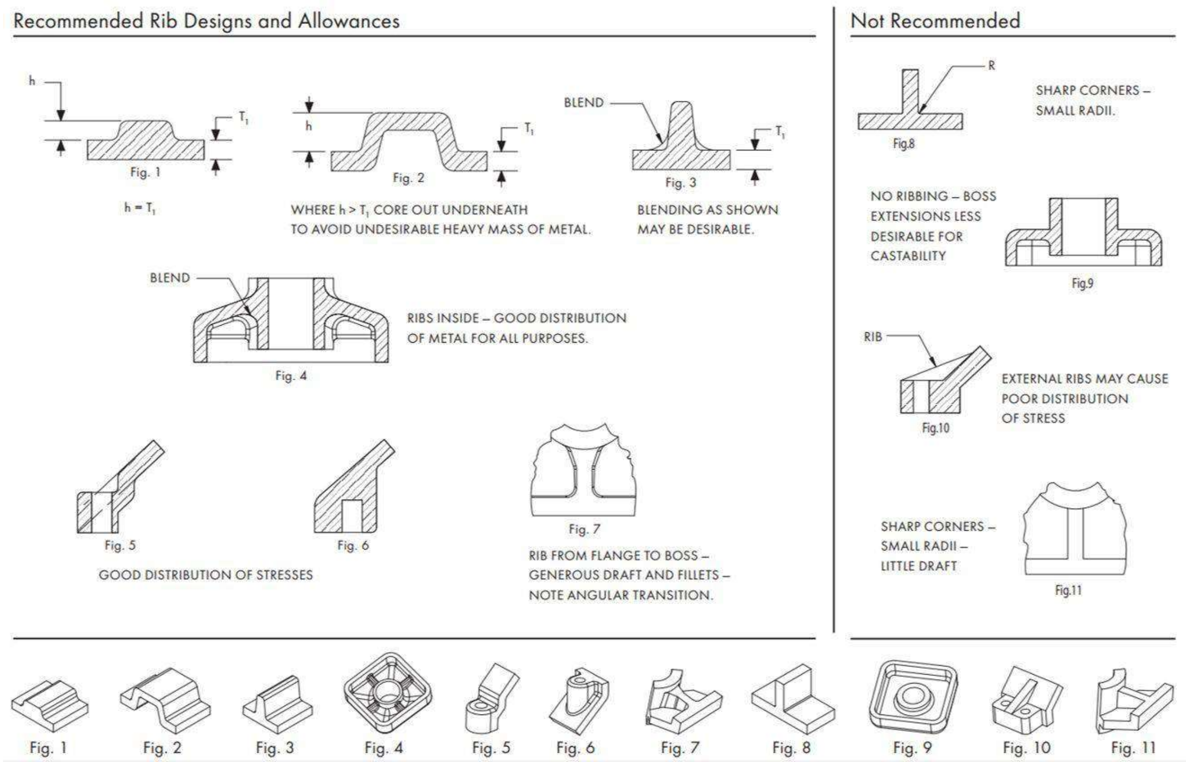
Ribs

Ribs are incorporated in a design to increase the stiffness thus add strength to aluminum die casting. So, ribs can assist in producing sound castings. Ribs are mostly paired along with other weaker sections such as thin walls, to impart additional strength to them.

It can often provide more strength than a thicker solid section, as thicker sections tend to have more porosity in them which reduces their structural capacity. However, overusing ribs can cause stress concentration at the edge of the ribs.

Ribs are often designed with hollow sections known as metal savers. They are a method to reduce the material usage in ribs and reduce the weight of the part.

The recommended rib dimensions for some common scenarios are shown in the following diagrams along with some conditions where ribs should not be used.



Rib Design Guidelines for an Aluminum Die Casting Part

Hole to Edge Space Design

Holes located too close to the edges of an aluminum casting will result in a weaker section. A minimum spacing must be maintained between the hole and edge to avoid excessive stress concentration in that zone. So a proper hole to edge space should be determined based diameter of the hole. A minimum clear distance should be maintained for two adjacent holes as well. Take it into account the diameter of both holes and their stress concentration zone.

Enough spacing is to avoid weaker section. You can also consider the second operation for holes if no sufficient hole to edge space.

Hole and Window

As far as design difficulty is concerned, holes and windows are usually the least of your worries. However, even the simplest of the features in the aluminum die casting part must be designed with proper attention to the details. You must ensure manufacturability when designing.

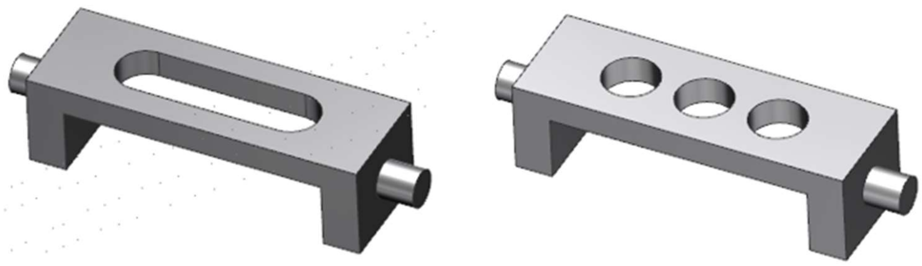
The most common application of Holes and Windows are different electronic device casings such as laptops, calculators, etc. There are a lot of holes positioned closely in these devices. Such a pattern causes problems for the metal flow.

You can have a better visualization of this problem from this video.

https://www.youtube.com/watch?v=imWjJDIWEsI

Holes and windows can also make it difficult to eject the casting. Because the solidification shrinkage of the part will cause the casting to grip onto the die. You can deal with these issues by following these tips when designing holes and windows.

- Provide an adequate amount of draft to counter the ejection issue. You will notice from the draft calculation that; holes and windows require more draft than any other features. This is because of the flat and closed wall setup along the inner perimeter.
- To avoid problems during metal flow, you can use bridge-like features to ensure continuous metal flow through the holes and windows. Providing cross feeders, flash-overs, overflows will ensure smooth metal flow in the part. You can easily trim down these extra features later.
- If your design allows, then you should remove large windows and replace them with consecutive smaller holes. Because long windows can disrupt the metal flow and compromise the casting’s integrity.



Long Windows Replaced with Multiple Holes

Side Cores/ Slides

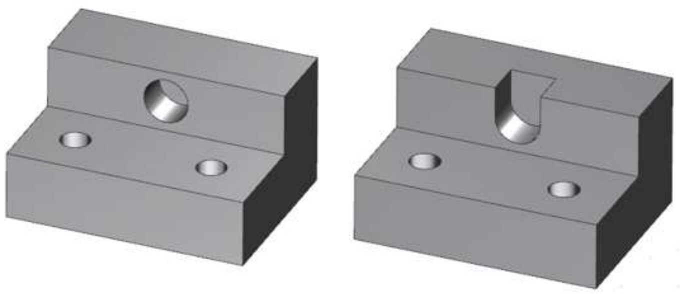
Any Holes and undercuts in the design parallel to the parting line can greatly increase the complexity of aluminum die casting or even make it impossible to cast by conventional means. Side Cores/ Slides makes it possible to easily manufacture parts consisting of holes and undercuts.

Cores are used to form holes in a part and slides are used when there is an undercut present in the design. However, they add a significant cost to die construction. The impact on the casting cycle of parts due to separate pulling out of the cores and slides other than the main die half.

The slides used within a die can also cause a parting line shift. It is caused due to the force applied by mechanical locks that hold the slide in place during casting. It is more common in the case of the unit dies.

Avoid Them if Possible

Designers should try to align such geometric features parallel to the die pull out direction or redesign the parts to eliminate the need for cores/slides. An example is given below showing how a part is redesigned to eliminate the need for a side core.



Aluminum Part Redesigned to Remove Core Slide Requirement

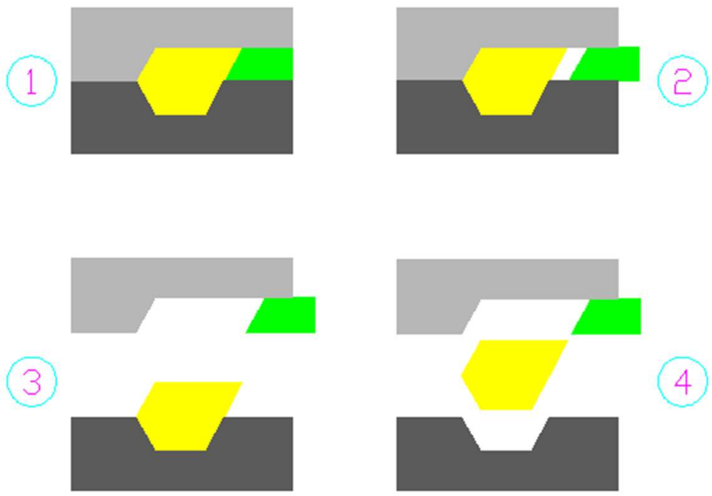
But, there will be cases where you must introduce a core/slide to cast a feature without having to machine it later. Core slides or pulls can be designed in such a manner that can eliminate the need for most if not all of the secondary machining operation.

So, the difficulties posed by initial increased tooling cost and slower casting cycle is offset by the reduced secondary machining operations. The repeatability for parts is also greatly extended this way.

Working Mechanism

Side core pulling and slide movements are usually powered by angle pins or hydraulic cylinders. The angle pin is a mechanical mean for the core/slide movement. It is activated by the opening and closing sequence of the main die.

So, an additional power source is not required for angle pins to function. It is also economical to produce. However, angle pins can interfere with the casting removal, only suitable for a short slide



Working Mechanism of a Simple Side Core

It is also difficult to use angle pins for the top slide and is only possible using springs. These issues can be resolved by using hydraulic methods. You can define a cycle of your choice, and make use of top slides with it. It doesn't interfere with the casting retrieval as well.

There are other methods of motion available as well that can be used for the core/slide. You have to choose the right one by analyzing your budget, production volume, size of parts, length of core/slide travel into the casting, etc.

It is recommended that you discuss it with your die caster for getting proper suggestions about designing a side core pulling/ slide mechanism. You are welcome to consult Sunrise Metal as well, we will be glad to assist you.

Thread Forming

When we are talking about thread forming, we will be mostly talking about casting external threads. While it is theoretically possible to cast internal threads, they are not desirable due to the complexity and cost of manufacturing.

External threads can be easily manufactured with a regular aluminum die casting setup with proper alignment with the parting line or with a simple slider mechanism. Internal threads will require a mechanism to rotate the core in the die.

This increases the cost of tooling and piece part. Internal threads are usually tapped as a secondary operation for speed and economy of production. This removes the need to remove cutting chips from the hole.

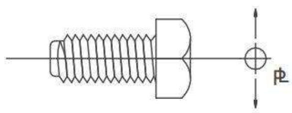
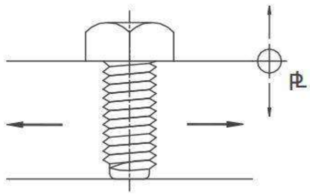
Ideal Tolerances for Threads

Threads can be easily formed with aluminum die casting equipment. Cast threads are usually limited to external threads where precision class fits are not required.

If you must have a precision class fit for your parts, you can always consult with your die caster. Secondary machining may be required for achieving better precision. Also, the major diameter should be following the specified thread form definition as agreed upon by both parties.

The maximum and minimum tolerances for some ideal thread forming operation are given below:

Table S-4A-12: Die Cast Threads Tolerances

Method of Forming Threads	Figure 1		Figure 2	
				
Tolerances	**Zinc**	**Aluminum/ Magnesium**	**Zinc**	**Aluminum/ Magnesium**
Minimum pitch or maximum number of threads per inch	32	24	32	24
Minimum O.D.	0.187" (4.763 mm)	0.250" (6.350 mm)	0.187" (4.763 mm)	0.250" (6.350 mm)
Tolerance on thread lead per inch of length	±.005" (±.127 mm)	±.006" (±.152 mm)	±.005" (±.127 mm)	±.006" (±.152 mm)
Minimum Pitch Diameter Tolerance	±.004" (±.102 mm)	±.005" (±.127 mm)	±.005" (±.127 mm)	±.006" (±.152 mm)

Die Cast Thread Tolerances Chart

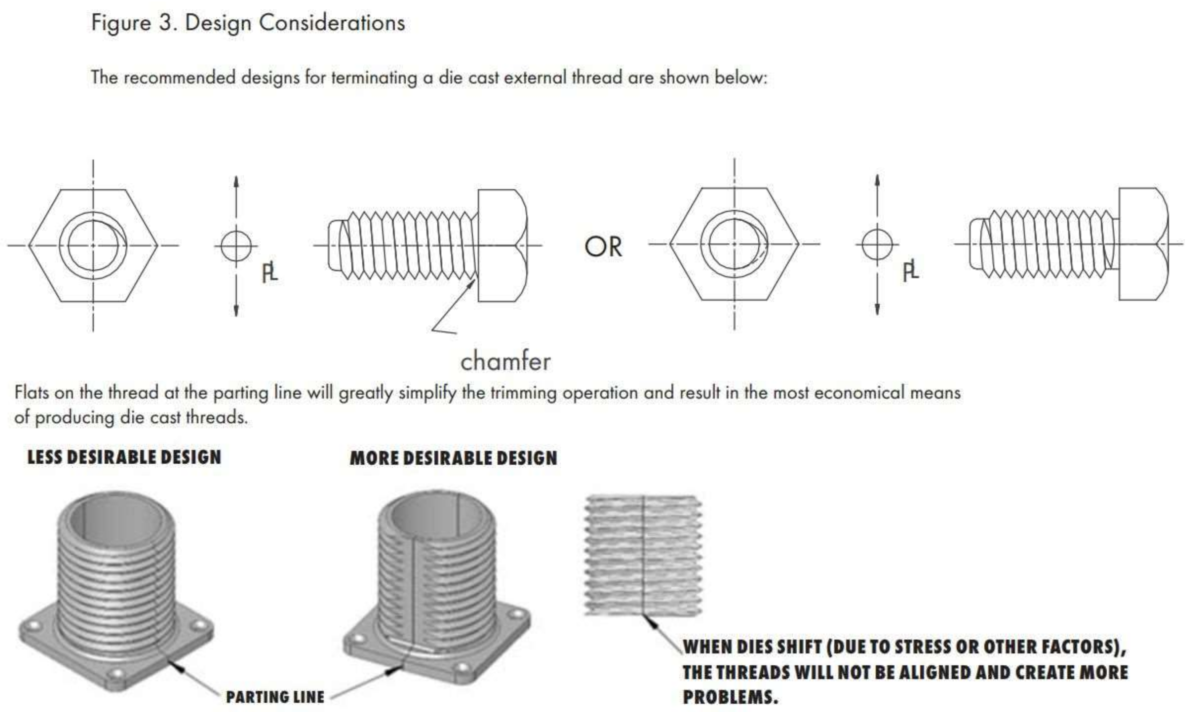
Design Considerations for Threads

However, do keep the following things in mind when casting threads.

- An additional trimming operation may be required for removing any flash formed between threads.
- Try to apply direct tolerances when possible instead of specifying thread class of fit.
- These values include tolerance limits for the moving die component, parting line, and linear dimension.
- Consult the die caster when tighter tolerances are required for the threads.

- Keeping the threads flat at the parting line can considerably simplify the manufacturing process. Because full diameter threads are not really necessary. Keeping it flat will allow minor die shifting without affecting the parts.

Here are some figures showing the recommended external thread configuration.



Recommended External Thread Design Considerations

Insert

Insert is a piece of solid material set in the die that becomes integrated into the aluminum die casting. It is necessitated when the selected alloy cannot meet a requirement and the design requires the integration of components made from other materials.

There are specialized systems available to make use of inserts in aluminum die casting. The insert is loaded inside the die cavity and the molten aluminum flows and surrounds the insert to complete the die casting.

You may need to incorporate threaded inserts in your design when you face the following situations:

- The bearing points are prone to abrasion and wear.
- Threads are subjected to excessive wear due to removing and inserting the fasteners too often.
- When you need threads with a higher tensile strength to deal with concentrated loads.

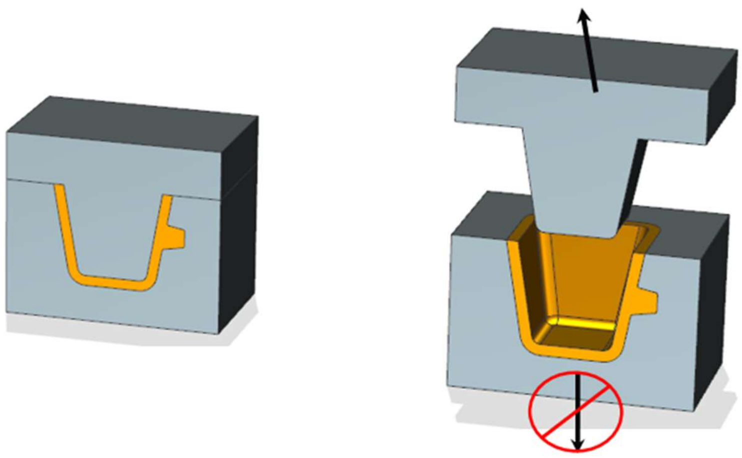
Insert die casting is costlier than regular casting and the complexity of insert setup will affect the cost of production.

Keep the following considerations in mind when dealing with inserts:

- Clearly define all the required specifications to the die caster. Often the inserts will require tighter tolerances due to the clearances in the die. Seek the die caster's approval to make sure the insert tolerances are sufficient.
- If a customer wants to supply the inserts, discuss with the die caster to ensure that the provided tolerances are within the recommendation of the caster. Because inserts without improper tolerances can severely damage the die.
- Analyze the stresses induced by the inserts. Make sure that the stresses won't interfere with your product's performance in the long run.
- The insert may be shaped any way you need to ensure that it can develop enough anchorage for the anticipated loading condition.
- Avoid sharp corners and other features that can cause stress concentration in the parts.

Undercut

Undercut usually refers to a recessed geometric feature or surface of a part that is not accessible with a straight cutting tool. In the case of die casting, undercuts are features that restrict ejecting the casting with a single pull mechanism.



Undercut Restricting Removal of a Casting

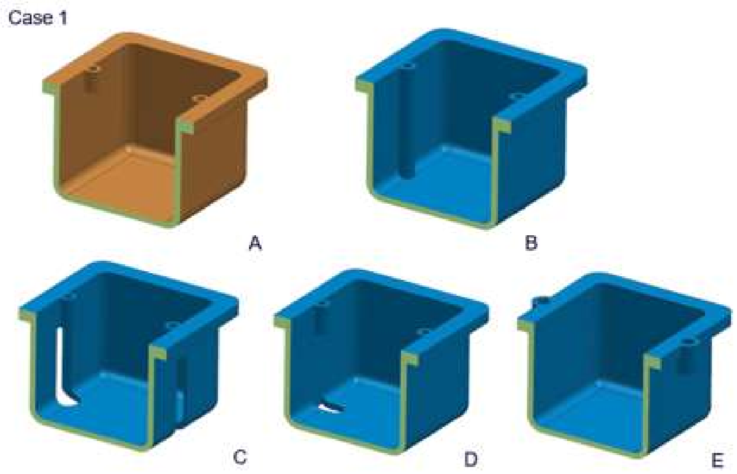
So, when you are designing a part, you must consider the difficulties that may arise during tooling and casting. Sometimes you can negate the effect of undercut by cleverly choosing an orientation for aluminum die casting. But most of the time it will be impossible without introducing side cores.

Adding side cores in your design will complicate both the die construction and the casting process. The mechanism is more complex thus costlier and the setup will take more time than conventional aluminum die casting.

Here are some important things you should keep in mind when designing undercuts.

- Discuss with your manufacturer whether he can make use of specialized cutting tools such as a T-shaped or V-shaped tool that can reach difficult places.

- Keep the number of external undercuts to a minimum. Because they will require side cores, which will increase the tooling cost.
- Some undercuts can be solved by adjusting the parting line.
- Redesign your part to remove internal undercut.
- Avoid undercuts that are not facing the die pull direction, they can't be ejected without placing side cores.
- Undercuts beneath bosses will obstruct the ejection of your aluminum casting.



Different Ways to Redesign the Undercuts Beneath Bosses

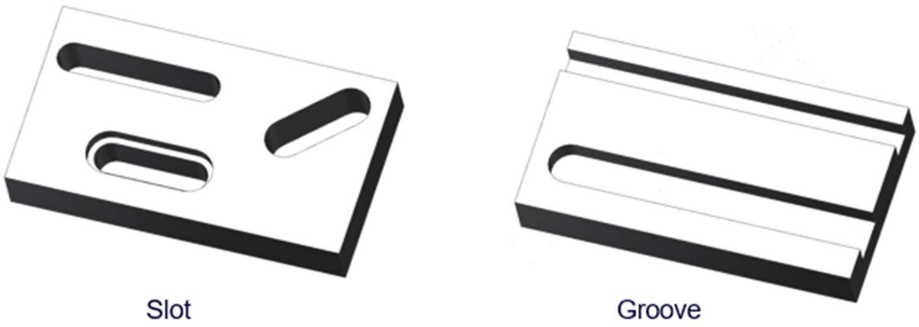
However, it would be best to avoid any sort of undercuts in your design if possible.

Slot & Groove design

Slot is an elongated hole that is may or may not have a round edge at the ends. Slot is mainly used in flat, rectangular aluminum parts. It will usually have a limited length. Slot is always through meaning that it will fully penetrate the part.

Slots can be differentiated into different types such as length bilaterally limited, length unilaterally limited, or semicircular elongated slot depending on their length and shape.

Grooves can consist of different shapes and sizes such as T-slot, Dovetail, Rectangular, Flat-bottom, V-shaped, Radius, etc. It is usually cut along the edges and works as a feature for mounting components made of other materials.



Slot and Grooves in a Die cast Section

Slots and grooves in a design act as a clamping element for other components. It also provides an opening to pass through other components such as switches, levers, etc.

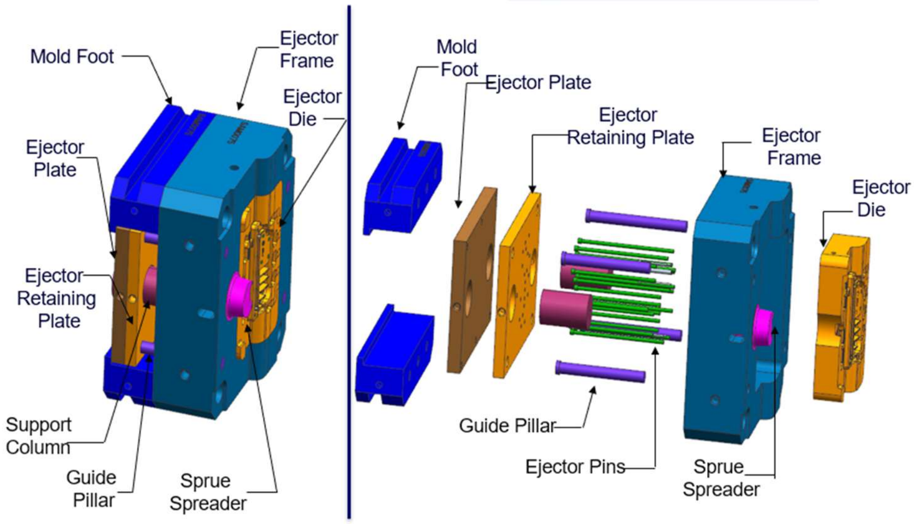
When designing slots and grooves the following tips can be useful.

- Keep in mind the dimension of other components that might be clamped or pass through the slot and groove.
- Avoid very closely spaced slots in your design. They can hamper the integrity of the parts and raise issues during finishing.
- Don't keep any sharp edges in rectangular/flat-bottomed grooves. Round all the internal or external edges as much as possible. It will help minimize plating costs and minor issues.
- The angle of V-shaped grooves should also be rounded for the same reason.

Ejector design

The solidified aluminum casting tends to clamp into the die due to the solidification shrinkage. So, additional force is applied from within the die through some ejector components to ensure removal of the casting.

The ejector mechanism of die casting equipment can comprise multiple components. A die casting mold mainly consists of two parts namely the cover die half and ejector die half. The ejector die houses the ejector mechanism.



Components of a Typical Ejector Die Half

The parting line acts as the separation point of the two die halves. The ejector die comes apart relative to the parting line after casting is finished. However, there is more complex equipment with multiple die setup which makes the ejection system more complex.

We will only be discussing a convention two-part die as this is more common and economical. In this case, the ejector half of the die contains the ejector pins, ejector plate, inserts, runners, and any sort of engraving present in the design.

The ejection mechanism mainly depends on two components of the die.

1. Ejector Pins
2. Ejector Plate

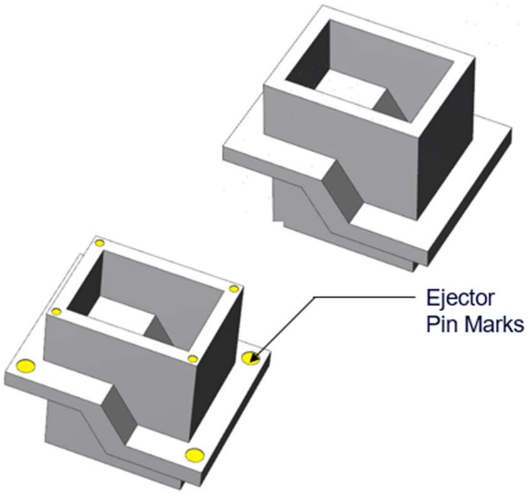
Ejector Pins

Ejector pins push out the solidified casting from the die. Another function of the ejector pin is to clamp the casting so that it doesn't bend due to the stress concentrated during solidification. But, a downside is that they leave a mark in the casting.

So, the location and size of ejector pins should be carefully selected considering the size, configuration, and some other aspects of the casting. The designer should follow these guidelines when designing the ejector pins.

- Ejector pins should be located in a non-functional area of the casting such as in the bottom of deep pockets, overflow, on a boss, or the bottom of ribs, etc.
- Ejector pins will leave a footprint upon ejecting a casting. So, do not keep the cosmetic surface of a casting facing the ejector pins.
- The recommended tolerance for a raised or depressed pin mark is .015" (.381 mm).
- Pay attention to your die casters recommendation, he can help you choose a better size, location, and the number of ejector pins.

Proper toolmaking practices can reduce the marks left by ejectors to a great extent. However, they will still be visible. The OEM and die caster should come to a mutual agreement upon where the pins can be provided.



Ejector Pin Marks in a Casting

Also, note that ejector pins will also form flash around it. It is normally left alone unless it is objectionable by the customer. The pin flashes can be crushed, flattened to minimize their footprint.

Ejector Plate

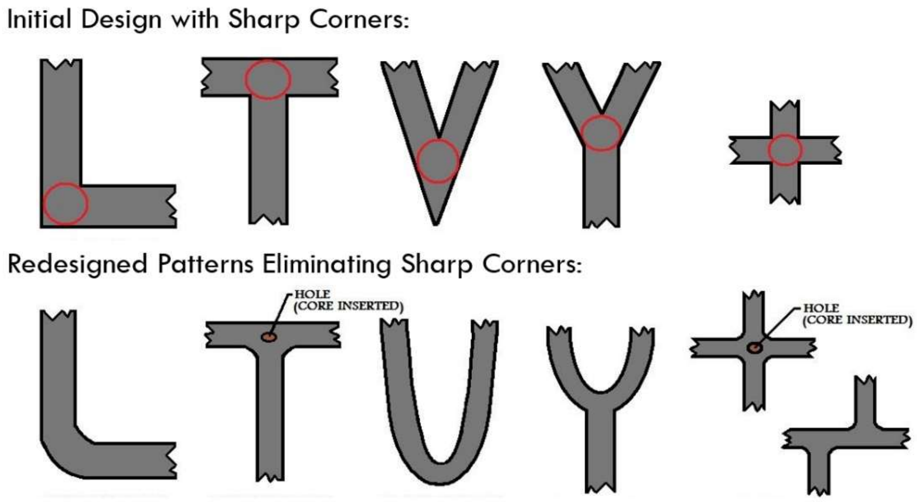
Ejector plates can function either as a complementary part for the ejector pins or function on their own. Usually, ejector plates are used as a mounting surface for the ejector pins. As pressure is only applied on the ejector plate, it simultaneously pushes the ejector pins forward and the casting is retrieved.

Ejector plates may also function alone without any ejector pins. However, it is usually seen in miniature die casting only. The part is ejected by the force exerted by the plate. This is better in the sense that ejector plates don't leave any marks on the casting like ejector pins.

Sharp Edge

Sharp edges are not welcome in aluminum die casting parts design. They create hot spots in the casting where stress concentration occurs due to the solidification shrinkage. It makes the corners prone to defects. It is also difficult to apply coatings on sharp edges.

So, designers tend to round all the sharp corners even with the minimum radius possible. Apply fillet, radii, or chamfers to all internal and external sharp corners. Here are some sharp corner designs that can be redesigned in the following manner.



Redesigning Sharp Corners of Aluminum Die Casting

Another problem with internal sharp edges is that they significantly increase the tooling cost. As you know that machining is pretty expensive, and the cost will substantially increase for higher precision requirements.

A perfectly sharp edge means that it will have zero tolerance. While it is achievable for external edges, achieving such precision in internal edges is nearly impossible. Internal edges will always have a minimum radius even if the most precise tools are used.

However, you can safely apply sharp edges along the parting line to ensure that the two die halves are perfectly closed. Other than that, keep sharp edges to a minimum.

Pressure Tightness

The pressure tightness is a measure of the integrity of aluminum die casting that indicates its ability to withstand a certain degree of fluid pressure. The purchaser may require the castings to have a specified pressure tightness in some cases.

Pressure tightness heavily depends on its density and porosity. Even a small amount of porosity can affect a part's pressure tightness and leaks may form during application. Multiple factors may affect the density and porosity of aluminum casting.

Entrapped air inside the aluminum die casting mold is a major concern for any manufacturer. When molten metal is injected under high pressure, the resistance formed by the entrapped air creates porosity. The gas pores present in the castings also reduce their density thus reducing the structural integrity.

There are so many factors involved that induce porosity is what makes it quite impossible to make a casting completely pore-free. Effective DFM and good quality control during the whole process is needed to obtain proper density and minimal porosity.

Die Design Considerations

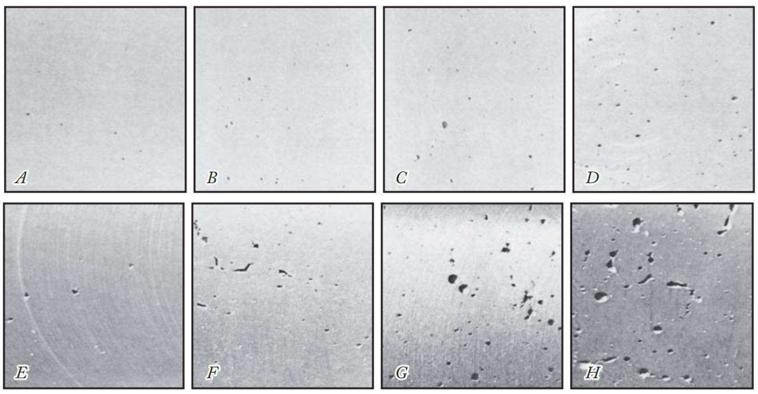
The designer should consider the following design parameters for obtaining the optimum pressure tightness for the aluminum die casting.

- Carefully follow the guidelines concerning the design of fillets, ribs, and corners. Otherwise, the metal flow won't be smooth and the turbulence can induce porosity.
- Try to keep a uniform wall thickness to promote smooth metal flow. Avoid heavy sections and any abrupt changes in the thickness.
- A hole that must be pressure tight should be cored instead of machining to minimize the effect of porosity.
- Provide sufficient draft in all holes and passages that won't be machined later. However, provide a minimum draft if a cored hole is going to be machined after casting.
- Simulation of the whole aluminum die casting process will help you identify the potential issues beforehand. Run a mold flow analysis to find out which features in the die should be improvised to improve the casting quality.
- Vacuum-assisted die casting equipment can exceptionally reduce the porosity in aluminum die castings. So, the parts have fewer defects and have better density properties.

Secondary Machining Operations

The secondary machining operations can also affect the pressure tightness of aluminum castings. The following guidelines should be adopted for machining.

- The densest part of any casting is at or near its surface. The inner part is usually less dense and has more porosity. So, a minimum machining allowance should be specified to avoid exposing the inner porous section of the casting.
- If a part requires greater machining allowance than allowed, it can expose the inner porous section. So, it may require impregnation later for optimum pressure tightness.
- Avoid machining both sides of a casting which needs to be pressure tight.
- Avoid a large draft where a large amount of stock will be machined. Especially the cored holes should be given minimum draft.



Porosity in a Part Reveals as it is Machined

Aside from the above considerations, the alloy selection also plays an important part in ensuring the pressure tightness of the aluminum castings. Certain alloys will give better results when aiming for pressure tightness of aluminum parts.

The pressure tightness of aluminum parts can be tested using various inspection equipment. Usually, the testing of pressure tightness is done in the range of 5 to 40 psi pressure. If higher pressure tightness is required the designer should discuss it with the die caster.

Part Strength

The strength requirement for your aluminum parts will have a significant impact on the overall production cost and time. So, the part's strength requirement should be clearly discussed with the die caster to select a viable aluminum die casting design approach.

The strength of aluminum die casting parts depends on many factors. Below we have provided some tips regarding ensuring the strength of die casting parts.

- First of all, start with the proper aluminum alloy selection. The alloy used for production will have a great impact on the strength of parts. However, mechanical strength is not the only key parameter here. The selected alloy must have sufficient workability and meet other conditions as well.
- It is desirable to cast-in-place as many features as possible. Having to machine any feature later will induce machining stress on the aluminum parts.
- The aluminum die casting process can be vacuum-assisted to reduce porosity in parts. It will increase the density of the parts resulting in better structural integrity and strength.
- Adding ribs to thin wall sections and bosses will increase their stability.
- Sharp edges are hot spots for stress concentration. So, these areas are prone to failure. It should be replaced with fillets to increase the rigidity of the parts.

Following these guidelines can help you increase the die cast parts strength. However, there are many more minor things left to be considered. Discuss with your die caster for suggestions regarding increasing the part strength.

Tiny Features

Tiny features require more sophisticated cutting tools to machine. The time, cost, and difficulty of tooling are much greater for tiny features. Features with standard tolerances can be achieved at a relatively low cost and are precise enough for any regular consumers.

However, you might need precision tolerances for your parts if they have a very sophisticated application. But precision machining beyond a certain limit is known as micromachining and it will not be achievable with standard machining tools.

Micro-machining deals with machining features with tolerances way less than a millimeter. It has a surprisingly higher cost compared to any standard machining operations. The designer must avoid such a level of precision in their design unless it's really necessary.

Premature defects taking place in the die will add considerable repair costs. So, keep tiny features to a minimum to reduce the tooling cost.

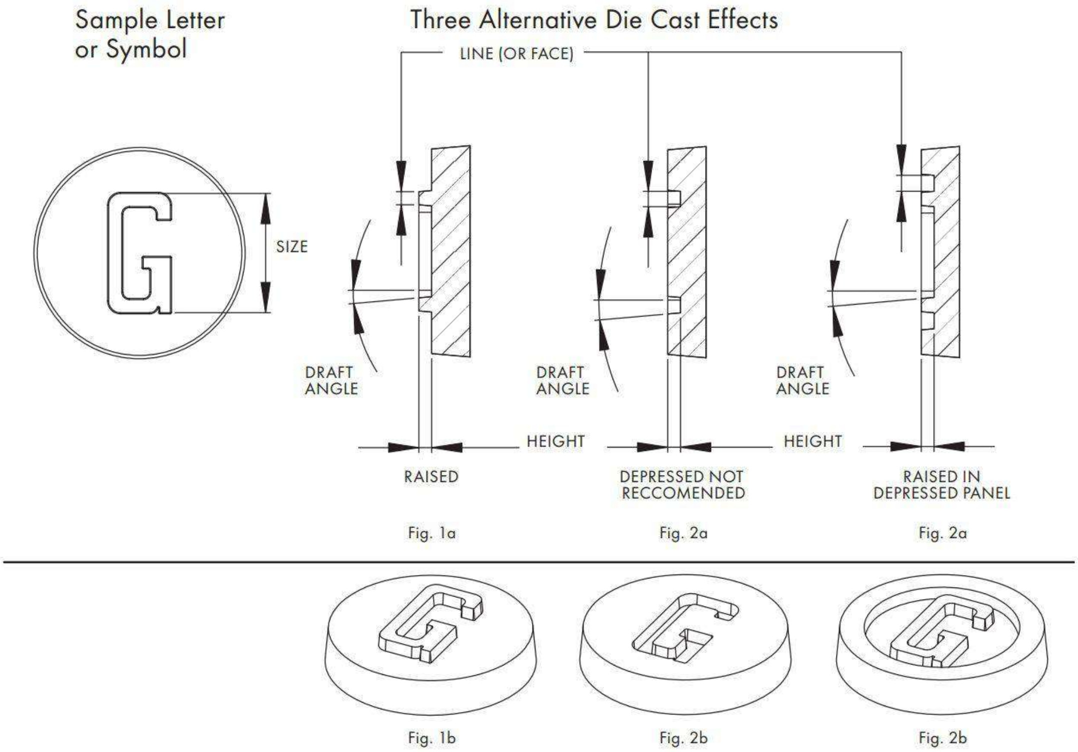
Die Casting Lettering

Most of the aluminum die cast parts designed for a company will have some sort of lettering or ornamentation to it. It can be letters, logos, trademarks, date code, or production number that allows convenient supply line tracking, branding, and many other purposes.

Die casting lettering or ornamentations may be implemented in a part in three ways.

1. As a Raised Feature
2. As a Depressed Feature
3. Raised lettering on a depressed panel

The most economical out of these three methods is keeping the lettering as a raised feature. A raised feature in the aluminum casting will require a depressed feature for the die.



Methods of Aluminum Die Casting Lettering and Ornamentation

Such a feature is easy to construct in a die and causes minimal wear during operation. So, Construction cost and maintenance cost over the lifetime of a die is less for raised (depressed in the die) lettering is less.

On contrary, depressed lettering in an aluminum casting requires raised features that protrude into die steel. It is a bit more complex to construct and requires more maintenance.

However, if the designer wishes to keep a leveled surface then the lettering can be done as a raised feature inside a depressed panel. This is much more practical as you can apply depressed features without worrying about damaging the die.

The additional depressed portion can be filled with a coating later. So, avoid directly depressed features in your parts. Here are some additional guidelines you can follow when designing.

- The face (line thickness) of any letters or symbols should be at least 0.010 in. (0.254 mm) or greater.
- The height of cast lines or symbols should be equal or less than its line thickness.
- If any lettering or ornamentation includes complex details or fine serifs may not be clear when cast. So, it is advised to simplify them if possible.
- The draft angle should not be less than 10°.

Following these guidelines should be helpful for most of the cases. If you have any other requirements then you can always consult with your die caster or Sunrise Metal for proper suggestions.