Aluminium casting design guidelines

Are you new to the casting process? Learn how to effectively implement design tactics for optimal manufacturability here. Design for manufacturability Optimizing your component design to take advantage of the die casting process is key to seeing a return on your investment. Whether your project is best suited for conventional die casting, multi-glass closet casting or injected metal mounting, it's best to design your component with the production process in mind. In other words, engineers should approach each project with the intention of designing for optimal manufacturability. Design for manufacturing (DFM) is a core methodology that ensures that cast cast parts perform according to specification and reduce the need for secondary operations. Given that these operations can often account for as much as 80% of component costs, it is important to minimize them during the design stage. DFM is more than just a concept—it's a way to remove costs and eliminate inefficiencies before the project moves toward production. In this blog, we'll go through three ways to design your die cast component to get the most ROI. Reduce weight and wall thickness In die casting, two of your highest cost dribear are the material and machine time. You can reduce the need for both by adding weight-saving pockets and thinning your walls. Reducing weight and wall thickness in cross sections may seem like an obvious answer. Less weight means less material, and less material means lower material cost. It also means reduced solidification time, which means you get more shots per minute. But some companies find themselves sacrificing performance for the cost. With some performance in mind, it's important to be intentional about reducing weight and wall thickness while maintaining some strength. When designing your component, you must use the mechanical and physical requirements of your project to select the most appropriate alloy that will work effectively with thin walls. For example, if your part must be corrosion resistant and stable, thin wall aluminum is a good fit. Aluminum is corrosion resistant and retains a high dimensional stability and hardness. Want to learn which alloy is best for your project? Use our dynamic metal selector tool to filter for your necessary mechanical and physical properties! Maintain consistent wall thickness While striving for a reduced wall thickness, it is perhaps even more important to maintain uniformity. This will go a long way to ensuring consistently stable, repeatable casting that is optimized for manufacturing. Varying wall thickness can lead to porosity from both varying flow pressures and non-uniform solidification. At Dynacast, our engineers have many tricks to achieve a mesh-shaped component with die casting while maintaining consistent wall thickness. In Figure 2, you can see that the component on the left has several walls that are much thicker than the thinnest part of the component. cast in this way, it would provide a weaker, porous part. Instead, our engineers will core the thicker walls to achieve more uniformity and incorporate ribs into the cores to ensure some strength. Consider draft angle and tolerance zones When designing your component, it is important to consider the possible ejection angles and tolerances for the project materials to avoid delays in redesign. For draft angles, generally 0.5º is achieved for zinc 1º-2º is attainable for aluminum. For exact tolerances, generally between ±0,001 and ±0,002 is possible for zinc, whereas aluminum can hold between ±0,002 and ±0,004. With attainable draft angles and tolerances in mind, you are better equipped to avoid constructing unnecessary costs in the design. Too often, companies will request demanding tolerances and minimal draft angles when such features are not needed to maximize some performance. As a result, their castings fail. Instead, take a more holistic approach to your design. Determine the non-critical dimensions of your component to allow for more mild tolerance zones. In addition to extending the life of your tool because there are fewer precise geometries that are torn down, allowing tolerance zones also makes it easier to plan tolerance stack-up of your entire component. This will help you avoid machining and secondary operations where possible, making your design work for you to get the most out of the die casting process. Work smarter, don't harder Change your part design to take advantage of the die casting process not only allows you to fully utilize the efficiencies of die casting, but can also better suit your business needs. Are you interested in learning more about how to effectively design your die cast component for optimal manufacturability? Sign up for our webinar, Work Smarter Not Harder: Remove cost in the design stage. Tags Design Innovation Die Casting Your biggest component production costs should not be wasted. Still, that's what many secondary operations will result—in fact, up to 80% of a component's cost can be attributed to secondary operations that can be made more efficient—or eliminated altogether. DOWNLOAD NOW Get your copy of the Die Casting Design Guide. Just enter your details below. Download your free copy of our Die Casting Design Guide to Learn: How to Reduce The Weight of Components Why Wall Thickness Adds Cost What the Best Draft Angle Is for Inside vs. Outside Surfaces It a critically important factor often overlooked by designers And much more. Enter your details for immediate access. From the Guide: Design for Manufacturing (DFM) is a core methodology that ensures that your cast cast parts perform according to specification and require minimum secondary operations. This is important when you consider that these can often account for as much as 80% of component costs. Download your copy to get the complete guide. Almost every part of a product, or the entire product, has its genesis the press casting process. Die casting describes a manufacturing process that allows manufacturers to create sharply defined smooth or structural surface metal parts. Using a cold or warm chamber manufacturing technique based on high pressure, the process forces and sprays molten metal into a reusable steel dying at a rate of 60-100 miles per hour. A number of clamps hold the mould in place during the injection, cooling and solidification stage. Similar to the injection molding process, which uses a different class of materials, die casting produces parts from a sustainable range of non-ferrous metals, such as zinc, magnesium, aluminum and a range of composite materials. The type of metal chosen to fabricate the part determines whether manufacturers will use the hot chamber or cold chamber method to inject the metal into the die. Many manufacturers prefer die casting over other manufacturing processes due to the ability to create such a variety of parts and products at high speed and with precision. Current applications for die castings include machinery, vehicles, appliances, toys, sporting goods, plantand fray for office equipment, enclosures and many other applications. In addition, die casting allows the production of components with fine details such as letters, textured surfaces and other functions without the need for further processing. The ability to maintain close tolerances, which often eliminates all machining, makes die casting suitable for products with lower volume as well. In recent years, innovations in manufacturing technology and materials science have eliminated many of the old design assumptions and process challenges. These advances have resulted in new specifications for essential design elements, including dimensional control, drafts and flatness. For designers involved in die casting, here are some tips and tips on how to design your part or product efficiently and economically. The departure lines for Die Cast Components and Products One of the most important parts of the die casting design will be to decide on any type of parting line that will divide the part and produce a contact surface between two or more components. Where the designer places this line depends on the geometric shapes and tolerances of the different surfaces. The designer has two choices — either a straight farewell line or a broken farewell line. You should try to design parts with a straight bill line because it is the cheapest option as far as tooling costs. When making a decision on the farewell line, the designer should consider the following factors: Customer specifications — Usually, the part or product specification of the customer will make it challenging to place the line. The customer's tolerance requirements also become an issue because line surfaces tend to have a lower quality compared to other surfaces. Die costs — A straight line of department can lower tooling costs. In some cases, however, it will be more design a broken line. This is because adding features to the part that would require side die draws, which are usually more expensive than stepping off the line. Processing — Many parts require post-production processing. Areas that have critical tolerance or finishing requirements should be located to one side of the die In addition, the area near the line should not have essential cosmetic requirements because gates and valves that will be placed along this line will be visible. Furthermore, this area will need further processing to minimize or remove material from the casting. Metal flow — The importance of the filling process depends on the correct position of the port inlet. The inlet, which must be located in the cutting line, determines whether the casting is filled properly with the molten metal. During high pressure djutning, the injection or filling mechanism may push the metal into the casting to prevent the part from shrinking during solidification. Cores — The positions of cores (grains forming holes in the part) will determine the location of the deseningen line. The designer must consider the location of the core, as well as the size of the diameter and length of each core for each hole needed in the casting. Knockout Pins (also known as ejector pins) — The mode of the avsegraden determines the degree of power required to knock out the part when printing thief casting. You should try to avoid undercutting whenever possible, but especially parallel to the deseningen line. These features may require additional die components or processing. In the process of designing die casting, metal flow is one of the most important considerations. If the shape is not filled correctly, it may result in defects such as visible surface flow lines and air pockets that create inner porosity in the part. You will also need to consider ejector pins that will eject the casting part from die. You need to design the pins in a way that leaves a minimum number of remaining pin marks on the surface of the casting. In addition, the ejector pins keep the part from being bent. Ejector pin marks result in depressed or raised impressions of about 015 (.381 mm). The diameter of the ejector pin marks will vary depending on the size of the casting. For optimal manufacturability, use raised ejector pin marks. Flash formation and location At the deflection line in the two dying halves, an extension of metal may form on the casting or where separate press parts cast a function. In addition, a metal seam may be the result of regular operation of the ejector pins. You can account for this flash in the design phase and determine the amount of metal you need to remove as well as the method of removal. Managing issues early in the design phase can result in cost savings for the overall manufacturing process. Correct Draft for Die Casting Parts The designer must incorporate drafts into the casting. The draft refers to the taper or slope assigned to the cores and other parts of the die cavity. elements prevent casting from getting caught in mold or tools during the ejection of casting, making it much easier to open die and easily eject casting from the döningsdö. Always try to introduce drafts into the process as early as possible. The drafts are based on the foreclosure line. The location of the draft – on an inner wall, outer wall and/or hole – will have different calculations. The variations in shrinkage will determine the correct calculation (the amount of drafts). Generally, in the formula, the number is always a constant. It depends on the alloy used and the depth of the surface. However, any molded surface that is parallel to the initial direction of die should be tapered for proper ejection of the part from die. In this scenario, the draft resultresults in an angle and is not constant. Off wall placement requires the least amount of drafts because casting tends to shrink away from die steel that forms outside the surfaces. However, unused holes require the most drafts. As the casting shrinks during solidification, it exerts large force around the matrix steel, which forms the internal surface of the hole. The inside wall also undergoes casting shrinkage on die steel that creates the surfaces on the inside walls. A die that is easy to open and get the part thrown out will result in a part that is more accurate for straightness/flatness and one that has a higher surface quality. Radii & Fillet The use of both fillet and radii can increase structural integrity. To promote metal flow, use a liberal radii and transition. For intersecting surfaces that meet at a sharp corner or edge, fillets can prevent high stress concentrations at the time, in both the die casting and the parts. Fillets reduce the concentration of heat in the die and part. Proper use of fillets will reduce the cost of die maintenance and increase the life of the tool. For a fillet projected into a location perpendicular to the breeding parade, you need to add drafts. The amount of drafts depends on the draft of the intersecting surface. To maintain the continuity of the edges and smoothness of the components, create fillets with a constant radius. Shallow castings tend to have smaller fillets. Deep pockets and other inter-interrogation corners require larger fillets. Wall thicknesses Generally, die castings consist of thin wall structures that have no hard and fast rules for minimum and maximum wall thicknesses. It is important to design uniform walls throughout the part and where variations occur. This will ensure a smooth metal flow during filling and minimize distortion caused by cooling and shrinkage. A good mold filling will produce parts with excellent properties and get defects. The key is to design the casting so the entire shape fills before solidification begins. Failure to fill the entire shape first can lead to cold rods (poor surface finish) in the casting. You can reduce the risk of cooling closing without any or unnecessary corners, which prevent melting flow in the mold, by using radii. Innovations in die casting technology make it possible to produce parts with minimum and maximum thickness that were unattainable a few years ago. Utilize this ability only when you find it necessary to improve performance or to achieve economic benefits. Otherwise, stick with uniform wall thicknesses. You can make the metal flow better through the mold with thicker walls and ribs. When the main wall has protruding features, make sure they do not add significantly to the thickness of the wall. Excessive bulk can delay cooling. When looking at the part from the die opening direction, make sure that the features of the project from the sidewall are not behind each other so you can avoid dying casting depressions. Although design casting allows the production of intricately detailed components, the designer should avoid the use of interior undercuts when designing the parts because the moving interior core mechanics are difficult to operate. You can produce this feature by processing, which increases the partial cost but reduces tool cost by avoiding core strokes in the die. Ribs, External Corners & Metal Savers When designing a part, you need to add ribs to thin walls. Ribs can increase stiffness and add strength to fabricate a more solid part. Often, the addition of ribs can do more to strengthen a component than solid material due to the porosity, and ribs result in an easier part. Incorrect rib construction can result in the concentration of stresses or the creation of unnecessary stresses at the edges of the ribs. You must place the ribs in the correct places for the ejector pins to be placed on the ribs for ejection strength. Ribs should assist the flow of molten metal. Whenever possible, include radii and fillets with ribs to reduce the number of rapid changes at sharp corners and in cross sections. It also allows improved output of the casting. Design into the part an odd number of ribs. This technique eliminates the build-up of stress over to an adjacent rib and reduces the formation of thick intersections. External Corners — Sometimes the die casting can have heavily square outer corners in some places. At desening line locations and die block crossings, the designer must have this type of corner. On other corners, the design should incorporate radii to avoid premature dying failure. It also reduces the risk of damaging the edge of the part during handling and assembly, and it increases the safety of the personnel handling the material. Metal Saver — Ribs have empty spaces between them called metal savers. Metal savers do not serve a functional purpose. But when designing your parts, keep in mind that positioning ribs too close to each other can cause weak metal savers. When designing your parts, evaluate the design for the following design weaknesses. When designing your part, review the specifications in the case of saver to avoid compromising the integrity of the part. Holes & Windows — Holes and windows can be among the most important considerations for design geometry. This element affects the molten metal flow through the component, and the configurations play a crucial role in manufacturability and the final quality of the component. The circumference of holes and windows tend to latch on to die steel during the cooling period, which can have an effect on the ejection of the part from die. Remember, when calculating the draft, the holes and windows require more drafts, compared to inside and outside wall functions. For recommended drafts. referred to nadca product standard publication #402. Draft diagrams are included in Chapter 4, standard S-4A-7-15. Die Casting Lettering, Symbols & Ornamentations Many cast parts require the designer to add letters, logos, trademarks and other identification to the casting. Other castings have date marks to identify the date of manufacture to distinguish one batch from another. Manufacturers use two methods to create these characteristics. The first technique, which is the most common and cost-effective, uses raised letters. In addition, since this method embeds the letters in the cavity, they last longer. The second technique pushes into the letters of the component by forming protruding signs of dying. This technique is more expensive to do in die and it makes the characters susceptible to wear and requires more maintenance. Incorporating Bosss Into Die Cast Design Many parts require managers to act as assembly points and standoffs. When adding this feature, the designer must be very careful to maintain consistent wall thickness. This will require the addition of a hole in the middle of the head. In addition, the inner and outer surface requires a draft. It is difficult for molten metal to flow up a high slim boss feature to fill it to the maximum level. Because of this, die casting designers usually add a generous amount of fillets and ribs (gussets) to assist the flow of the molten metal into these areas and to facilitate the ejection of the part from die. Molded Surface Finish on Die Cast Parts The finish of the tooling will determine the finished surface of a die cast component. A highly polished finish on die will produce good surface casting parts. The surface varnish on most tool ing makes it easy to get a matte finish. Some decorative molded components and other castings and products require the application of an external surface finish. So designers and manufacturers can plan for as-cast finish from the beginning of die design, parts NADCA finish in five classes and offer the following guidelines: Class 1: Utility Grade — This class does not have any cosmetic requirements for as-cast finishes and allows for cold closed, rubs, porosity, lubricant build and other flaws. The end-use finish can be as-cast, or the customer can a protective coating that anodize (non-decorative) or chromate (yellow, clear). Class 2: Functional Grade — Allows surface defects that can be removed by point polishing or covered by heavy paint. For the final finish, the customer may choose decorative coating, such as varnish, enamel, plating (Al), chemical finish or polished finish. Class 3: Commercial class – Permission for the removal of surface defects by methods agreed by the customer. For the final finish, the design can call for structural parts in high-stress areas. Other options are Plating (Zn), electrostatic painting and transparent colors. Class 4: Consumer grade — This variety has non-offensive defects on the surface. In case of rejection due to ytviviness (flatness), as observed by reflection of light, the client can determine an approach in agreement with the die castern. For end use, the designer can use decorative parts. Class 5: Superior grade – The finish that applies to limited areas of the part has a maximum value expressed in micro-inches on the printout. End use consists of O-Ring Seats or Packing areas. As-cast surface categories do not apply to machined surfaces. The designer must identify finished machined surfaces separately on the design drawing. The customer and the press thrower must approve the final selection. CAD Feature Order To facilitate the development of a die casting design model and to reduce the possibility of malfunctioning, NADCA recommends the use of the following CAD feature order: Basic geometry functions: Put the functions that make up the base geometry of the model at the top of the function tree, including bosses, extrusions, circuits, cuts, shells, lofts and sweeps. Cast holes with core shavings: Holes that you will cast during the manufacturing process, and which you may or may not press or the machine later. Department lines: To place the lines next in the order of the paragraph, including any department line that appears in a component after the draft has been applied. Draft: This feature goes next. Fillets: Add fillets to all geometries, with the exception of certain means cedals. Processing: Finally, add all machine functions at the end of the functional order — suppressed and unsuppressed. Putting processing functions last allow easy creation of as-cast and machined model configurations. This is a great way to identify as-cast features and features that require processing. Sometimes very large fillets, drafts or very tapered components can be included in core geometry functions. Place these design elements at the top of the functional order tree. Focusing on these principles of successful Die Casting Design Summary, along with the functional requirement, the designer must incorporate numerous manufacturability-related factors into the design of a part or product to produce successful castings economically. In order to achieve this overall design goal, have Goals in mind during the design process: To learn more about die die services offered by Premier Die Casting, please contact us online today. Back to top Contact Us for Die Casting Services Services