Basic die related design principles

Stefan Fredriksson - SweCast

Parting

A casting has to have some kind of a line that splits the component and creates a contact surface between two or more mould parts. The placement of this line depends on the geometrical shapes and what kind of tolerances the different surfaces has. Often the surfaces that are placed in the parting line become poor and needs to be machined afterwards.

To be able to get the component out of the mould or the tool, it should be design with drafts and some flat areas that allows the component to be knocked out of when finished.

There are two ways of a parting line. Either you have a straight parting line or you have a broken parting line. Always aim for a straight parting line to keep the cost of the tool as low as possible. But sometimes a broken parting line can eliminate cores. Otherwise a broken parting is often more expensive. There is also more cost effective to keep all deep cavities on one side of the tool. Example of a straight and broken parting line is giving below:
In order to decide the parting line, there are some aspects that are good to take under consideration:

- Customer requirements: Often, the customer has its own requirements of the casting design that makes it difficult to place the parting line. Also, the tolerance requirements from customer matters because the surface in the parting line is often poor.
- Tool costs: The benefits of a straight parting line are often cheaper tool costs. In some cases, it’s more economical to have a broken parting line if the number of movable cores can eliminate so that the costs of tools is reduced.
- Machining: Often, the component needs some machining afterwards, therefore it is of relevance where to place the parting line. As said before, the surface in parting line has lower quality than the other surfaces. Sometimes the surfaces can not be machining because the requirements. Some surfaces can also be hard to reach. Moreover, you have to machining where the inlet and vents are placed.
- Metal flow and solidification: The inlet has to be in the parting line. The filling process is very important, and the placement of the inlet decides whether the casting is filled properly or not. When high pressure die casting, the feeder has as purpose to continue pressing melt into the casting to prevent the casting to shrink during its solidification. This means that the location of the inlet is also very important.
- Cores: The location of the cores decides where to put the parting line.
- Knockout: Depending on where the parting line is located, it demands more or less force to knockout the component when pressure die casting. Also, when gravity casting the knockout affects the parting line.

All these aspects are important, but the most important is the metal flow, because adverse mould fills often result in defects, for example air and oxides. After the tool or component is designed it’s often difficult and very expensive to do any changes.

*Image 3. Two different design of a casting, the one on the right is better regarding the parting line.*
Drafts

To avoid the casting from get caught in the mould or tool when knocked out, the casting must be provided with drafts.

Image 4. The function of drafts

To get the most suitable casting design, drafts should be introduced as early as possible in the process. The draft starts from the parting line and there are ways to calculate the drafts depending on if it's an inside, or outside wall or and hole. In the formula, there is also a constant depending on the material.

Image 5. Formulas to calculate the drafts.

\[ D = \frac{\sqrt{L}}{C}; \quad \alpha = \frac{D}{0,01746 \cdot L} = \frac{1}{0,01746 \cdot C \cdot \sqrt{L}} \]

Where C is the constant that is depending on the where the draft is placed and which material is used. The draft angle is usually signified with \( \alpha \).

Table 1 Constant (C) to calculate draft angle

<table>
<thead>
<tr>
<th>Alloy</th>
<th>Inside wall</th>
<th>Outside wall</th>
<th>Hole</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mg</td>
<td>7,00</td>
<td>14,00</td>
<td>3,33</td>
</tr>
<tr>
<td>Al</td>
<td>6,00</td>
<td>12,00</td>
<td>4,68</td>
</tr>
<tr>
<td>Zn</td>
<td>9,90</td>
<td>19,80</td>
<td>6,75</td>
</tr>
<tr>
<td>Cu</td>
<td>4,90</td>
<td>9,90</td>
<td>3,33</td>
</tr>
</tbody>
</table>
According to the calculations of the draft, the inside draft should be twice as big as the draft for an outside wall. In case of very big or small components these recommendations are not always correct, but they give guidance when to decide draft angle. A general rule is to design the casting with drafts about 1-2 degrees.

### Shrinkage

Shrinkage is caused by the volumetric contraction of the metal when it changes from the liquid state to the solid state. It continues until the metal is at ambient temperature. There are many parameters that affect the shrinkage. Some of them are:

- Alloy cast.
- Metal temperature
- Injection pressure
- Time before ejection
- Die temperature
- The shape of the casting

In the figure above the casting can’t shrink constant throughout the casting because of the design. This will give rise to stresses.

In the tool, you must compensate for the shrink of the component that arises during the solidification and cooling. With some simple formulas the shrinkage can be estimated. The formula under is valid for linear shrinkage.

\[
\varepsilon = \alpha \cdot \Delta T
\]

**Explanations:**

- \(\varepsilon\) = Shrinkage
- \(\alpha\) = Expansion coefficient [°C]
- \(\Delta T\) = Difference in temperature of casting (knockout temperature on casting minus room temperature)
When pressure die casting, apart from the shrinkage from the casting there has to take in consideration the expansion of the tool. The change of volume that occurs in the tools is in fact the expansion of the tools minus the shrinkage of the casting.

\[ \varepsilon = \alpha \cdot \Delta T = \alpha_{\text{tool}} \cdot (T_{\text{tool}} - T_{\text{room temperature}}) - \alpha (T_{\text{casting}} - T_{\text{room temperature}}) \]

**Explanations:**
- \( \varepsilon \) = Total shrinkage
- \( \alpha \) = Expansion coefficient [ºC]
- \( T_{\text{tool}} \) = Equilibrium temperature [ºC]
- \( T_{\text{casting}} \) = Knockout temperature [ºC]

During the development of the casting the shrinkage is often estimated. To make the work easier and more effective there are some values that can be used for general shrinkage.

**Table 2  Shrinkage on different alloys**

<table>
<thead>
<tr>
<th>Material</th>
<th>( \alpha ) [ºC]</th>
<th>Total shrinkage [%]</th>
<th>Normal shrinkage [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Magnesium</td>
<td>( 26 \cdot 10^{-6} )</td>
<td>4,1</td>
<td>0,7³ / 2,0⁴</td>
</tr>
<tr>
<td>Aluminium</td>
<td>( 21 \cdot 10^{-6} )</td>
<td>7,1</td>
<td>0,5³ / 1,3⁴</td>
</tr>
<tr>
<td>Steel</td>
<td>( 10 - 12 \cdot 10^{-6} )</td>
<td>-</td>
<td>1,6³ - 2,6⁴</td>
</tr>
<tr>
<td>Iron</td>
<td>( 10 - 12 \cdot 10^{-6} )</td>
<td>3,2</td>
<td>2,0⁴</td>
</tr>
<tr>
<td>Copper</td>
<td>( 18 \cdot 10^{-6} )</td>
<td>5,3</td>
<td>1,6⁴</td>
</tr>
<tr>
<td>Zink</td>
<td>( 27 \cdot 10^{-6} )</td>
<td>4,1</td>
<td>0,9³ / 2,6⁴</td>
</tr>
</tbody>
</table>

³ Pure metal
⁴ Tool or model shrinkage
³ When pressure die casting
⁴ When sand casting
Mould/tool

The costs of a casting are related to the difficulties of the geometrical shapes. Therefore it is important to design the casting with as simple geometrical shapes as possible to keep the costs down. The most geometrical shapes are today available to cast with exception of deep cavities and deep holes.

![Image 7. Because this component has deep cavities it’s not a good casting in a cost point of view.](image)

By casting deep holes there will be worse dimension accuracy because it requires movable cores and also it will be more expensive. To optimize the casting, it should be design without movable and loos cores.

![Image 8. With or without loose cores](image)

Wall thickness

A condition to receive casting with good properties and few defects is a good mould filling. One way to achieve this is to design the casting so that the whole mould is filled before its starts to solidify. If not the whole mould is filled first, there is a good chance of cold shuts in the casting. To reduce the risk of cold shuts, the casting should be designed without any sharp or unnecessary corners. This will complicate the melts advance in the mould.
Other ways to make the melt flow better thru the mould is to equip the casting with thicker wall thickness or ribs. To avoid problems with cold shuts there are also recommendations on how thin the sections can be designed depending on material and alloy. The table below describes recommendations on minimal wall thickness.

Table 3  Recommendations on wall thickness

<table>
<thead>
<tr>
<th>Material</th>
<th>Recommendation on minimal wall thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>High Pressure die casting</td>
</tr>
<tr>
<td>Aluminium</td>
<td>0,5-1,0</td>
</tr>
<tr>
<td>Magnesium</td>
<td>0,5-1,0</td>
</tr>
<tr>
<td>Zink</td>
<td>0,3-0,5</td>
</tr>
<tr>
<td>Steel</td>
<td>-</td>
</tr>
<tr>
<td>Gray and ductile iron</td>
<td>-</td>
</tr>
</tbody>
</table>

$^1$ For aluminium, these values are also valid in die casting

Image 9. Examples on how to make the melt flow better during the filling process by design the casting without any sharp edges or corners.

Image 10. Uniformity of casting thicknesses
All the time, the development is moving forward and there are today ways to produce even thinner section. For example there is one method called tixomoulding when use magnesium and also there is another method where you can cast in vacuum to be able to produce some thinner sections.

The equation below describes the relation between the distance from the inlet and the minimal thickness for high pressure die casted magnesium.

\[ t_{\text{min}} = 0.8 + (0.004 \text{ to } 0.005) \cdot S \]

\[ t_{\text{min}} = \text{Minimal wall thickness} \]
\[ S = \text{Distance to the inlet} \]

The size and the design of the casting can be seen as a projected area i.e. cross section area in the parting line, see table below.

**Table 4 Minimal wall thickness**

<table>
<thead>
<tr>
<th>Projected area on the casting (cm²)</th>
<th>Minimal wall thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mg</td>
</tr>
<tr>
<td>&lt;25</td>
<td>0.8</td>
</tr>
<tr>
<td>25-100</td>
<td>1.3</td>
</tr>
<tr>
<td>100-500</td>
<td>1.8</td>
</tr>
<tr>
<td>&gt;500</td>
<td>2.5</td>
</tr>
</tbody>
</table>

**Insert**

Take advantage of the possibility to cast details in other materials in a component. This is available in pressure die casting and gravity casting. It is not unusual that is occurs great stress concentrations between the material because they both have different thermal expansion coefficient (α/°C). Some examples of inserts are magnets, thread pegs, lining in engines.
Summary: (10 golden rules for a successfully construction)

1. Avoid any accumulation of material. Design the casting with as even thickness as possible.
2. Avoid sharp edges. Design the casting with radius.
3. Design the casting with drafts.
4. Avoid deep cavities that require cores.
5. Always try to get as few moulds and tool parts as possible.
6. Place the parting line to enable as good mould filling as possible.
7. Avoid cleaning. Try to include these operations in the machining.
8. Aim for a directional solidification.
9. Remember that defects and solidification often decides the strength of the material.
10. In loaded castings, there are often better with compression stresses then tensile stresses.