Number of Cavities and Positions

School of Technology and Management, Politecnico Institute of Leiria

Careful consideration should be given to the number of cavities within a mould. The costs of making the mould must be balanced against the production requirements for the injected part. Good marketing data that can predict the sales volume for the injected part often are the key to successfully determining the proper number of cavities. Also the injection machine model is very deterrent in defining the number of cavities. Single-cavity moulds are frequently used for limited production runs or when the part is very large, so that the size requirement of the injection moulding machine does not become excessive.

When more than one injected part, all the same, is made in the same mould, the mould is called a multi-cavity mould. It can be stated that the more parts produced to specification in a single moulding cycle, the higher the profit. The objective of a multi-cavity mould is to produce multiple identical parts within each mould injection cycle. This is much simpler said than done, even if we assume that steel dimensions and cooling between cavities are identical. Successful multiple cavity moulding requires that the melt conditions introduced to each of the cavities be the same. For instance, a tree or fishbone runner layout will use less material than most runners, but will result in the most imbalanced filling of cavities in multi-cavity moulds when precision moulding is needed. As melt enters this runner system, it will progressively reach each of the gates and cavities along its length. At a relatively fast fill rate, in a mould with non-restrictive gates or cavity wall thicknesses, the cavities closest to the sprue will fill first and then progressively fill the remaining cavities along its length. In doing this, each cavity will be filled at a different pressure, flow rate, and melt temperature. A balanced flow to each of the cavities in a multi-cavity mould will maximize the potential of producing parts within specification and provide the largest process window. When producing a multi-cavity mould, a mould that allows the use of mould inserts to define the cavity and core, might prove valuable. Therefore, when a part needs to be changed, the inserts for one part can be removed and the inserts for the second part put in their place.

Some moulds are built so that the cavities are different from each other, in order to inject different parts with the same mould. These moulds, called family moulds, are usually constructed so that all of the parts of a particular assembly can be moulded at the same time (like model airplanes, etc.). The obvious advantage of family moulds is that only one mould needs to be used to make all of the parts in the assembly. However, there are also some disadvantages. If parts of different shapes are to be made in one mould, the runner system should be sized so that the flow down each leg is the same, thus filling all of the moulds simultaneously. The sizing of runner systems for non-uniform parts is very complicated and results in a slight decrease in part accuracy in moulds where the cavities are different if not properly designed.
Experience has shown that family moulds are most useful when small quantities of an assembly are to be made or if the mould is just for prototyping and separate, multi-cavity moulds making identical parts are to be constructed after the part has been exactly defined and the market established.

When designing a mould for injection moulding, certain information must be defined in order to start the design process, such as process variables and process parameters. Process variables are: part geometry, moulding material, demands on the part, lot size and delivery date. The process parameters are: number of cavities, major mould dimensions, injection machine model, mould costs and cost of injected parts.

These variables and parameters are all dependent of each other, for instance, the number of cavities also depends on: the available production time, product quantity required, machine shot size and plasticizing capacities, shape and size of the mouldings, and mould costs. In spite of these dependencies, the number of cavities is directly dependent of the major mould dimensions, which is determined from the injection machine model. Once the injection machine is chosen, the major mould dimensions are automatically defined, which then define the limit for the number of cavities within the mould.

In order to aid the determination of the number of cavities in the mould, and bearing in mind the limit number of cavities, three simple formulas are presented. Use the minimum value derived from the following formulas to define the number of cavities.

### Product quantity
If the dimensional tolerance of the part is not very critical and a large number of mouldings are required, multi-cavity moulds are preferred. The number of cavities is dependent on the time available to supply a specific lot of parts (tm), the number of parts in the lot (L), the cycle time to produce a single set of parts (tc), the reject factor (K), expressed as K = 1/(1-reject rate).

The relation is:

\[ NC = \frac{L \times K \times tc}{tm} \]

### Shot capacity
The injection machine shot capacity is also a factor in determining the number of cavities. Take 80% of the machine capacity as the shot weight (S) and divide it by the part weight (W) to get the number of cavities.

The relation is:

\[ NC = \frac{S}{W} \]
**Plasticizing capacity**

The injection machine plasticizing capacity is also a factor. Divide the plasticizing capacity \( P \) of the machine by the estimated number of shots per minute \( X \) and part weight \( W \).

The relation is:

\[
NC = \frac{P}{X \times W}
\]

After the number of cavities has been established, the cavities have to be placed in the mould as ingeniously as possible. In injection moulding machines, the barrel is usually positioned in the central axis of the mould. This establishes the position of the sprue. The cavities then have to be arranged relatively to the central sprue in such a way that the following conditions are met:

- All cavities should be filled at the same time with melt of the same temperature.
- The flow length should be short to keep scrap to a minimum.
- The distance from one cavity to another has to be sufficiently large to provide space for cooling lines and ejector pins and leave an adequate cross section to withstand the forces from injection pressure.
- The sum of all reactive forces should be in the centre of gravity of the platen, in other words, there must be equilibrium of the forces in the mould during injection.

Bearing in mind the considerations for cavity layout, three possible solutions may be used. In the table below, is illustrated the possible solutions with the advantages and disadvantages of each one.
### Table 1: Comparison of cavity layouts.

<table>
<thead>
<tr>
<th>Cavity Layout</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular</td>
<td>Equal flow lengths to all cavities, easy demoulding (especially of parts requiring unscrewing device).</td>
<td>Only a limited number of cavities can be accommodated.</td>
</tr>
<tr>
<td>In series</td>
<td>Space for more cavities compared with a circular layout.</td>
<td>Unequal flow lengths to individual cavities (a uniform filling is only possible with corrected channel diameters).</td>
</tr>
<tr>
<td>Symmetrical</td>
<td>Equal flow lengths to all cavities without gate correction.</td>
<td>Large runner volume, much scrap, rapid cooling of melt. To avoid this, a hot manifold or an insulated runner system must be used.</td>
</tr>
</tbody>
</table>

### Bibliography

*How to Make Injection Molds; 3rd edition*; G. Menges, W. Michaeli and P. Mohren; Hanser Publishers


*Plastics – Materials and Processing; 3rd edition*; A. Brent Strong; Pearson Prentice Hall