7 Step-by-Step Design

7.1 Predesign Requirements

Before starting a mould design, the designer should be in possession of the following information (some items of which have been mentioned previously).

- An unambiguous fully detailed component drawing
- Specifications of the moulding material including grade and colour
- The moulding machine specifications
- All the estimating details including any sketches
- Tool specifications as follows:
    - Number of impressions
    - Type of mould e.g., two-plate, three-plate, split, side core, hot runner, etc.
    - Type of runner system
    - Type of gate
    - Method of de-gating
    - Use of robotics
    - Estimated cycle time

7.2 Golden Rules

1. Never start a mould design without all the necessary information.
2. If an established design works well, don't embark on a totally new design if you can base your design on the established one.
3. The simpler the design the more reliable and efficient it will be.
4. Always sketch two or three alternative approaches to the design before committing yourself to the first one you think of.
5. Draw a sufficient number of views so that the design can be understood fully.

7.3 Step-by-Step Design

It is very difficult to explain in words alone how a mould is designed; we will therefore follow through a step-by-step design of a typical mould tool from start to finish which illustrates the procedure. In order to do this, we will consider a moulding that has to be produced on an eight-impression basis.

Note that in Chapter 9 the two-plate tool design is discussed in more detail.

7.4 Design Example

For the sake of simplicity and clarity this example will be fairly basic; nevertheless, the principles involved are the same for any mould design (see Table 7.1).

Table 7.1 Specifications for design example

Component	Flanged housing
Type of tool	Two-plate
Gate type	Edge or sub gate
No. of impressions	8
Material	Polyacetal
Cycle required	20 seconds
Production rate required	50,000 per month

A component drawing has been supplied as shown in Figure 7.1. It consists of series concentric diameters with a hole through the middle.

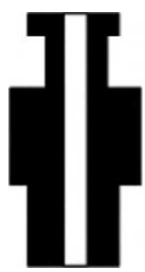

Figure 7.1 Flanged housing

The dimensions of the part have been omitted in this example but we will assume that we can mould them to the drawing tolerances on an eight-impression tool.

Among the first considerations we have to make is 'Have we ever moulded a similar shape before?' If we have, we should look up the design and find out whether the tool ran satisfactorily. We would then be able to use this design as basis for our new component. We will assume in this case that we cannot find a similar design and will have to design the whole tool from scratch.

Assuming we have all the information required to hand (as listed above) we can make a start. The first things we have to consider are the following:

- Where should the split line be located?
- Where will we gate it and what type of gate is required?
- Where we are going to eject it and how?
- Will venting be required?

7.4.1 STEP 1: The Split Line

This is always a crucial stage in the design process. If we get this wrong the repercussions will be severe in production. In fact there are only two possible places where this component can be split: at **B** or **C** in Figure 7.2. If we tried to split the component at **A** or **D** it would be undercut in the tool and it could not de moulded in a two-plate tool because it could not be ejected from the cavity.

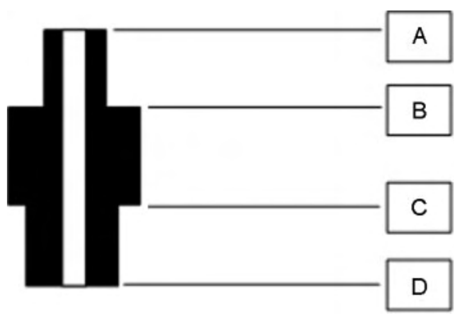

Figure 7.2 Possible split line positions

We could split the tool at position C as shown in Figure 7.3. If we were to select this position for the split line, this ***would*** work as the moulding will shrink away from the cavity walls and on to the pin that forms the central hole.

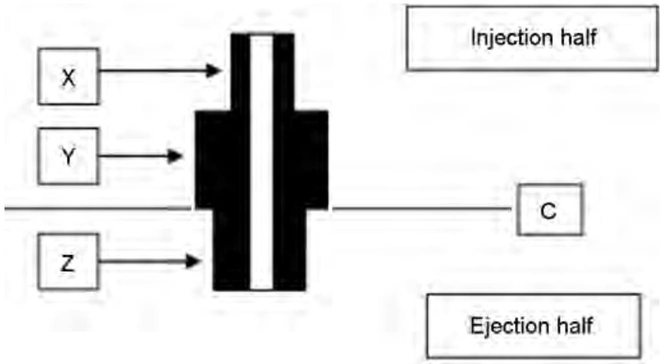

Figure 7.3 Part split at C

However, this is not the best method, especially from the point of view of accuracy and toolmaking, because the majority of the cavity form (**X+Y**) would be in the injection half or fixed half of the mould and it would be better if the majority of the form were in the ejection side of the tool. It is desirable to have as much cavity form in the ejection half as possible because the majority of the toolmaking work will be on this side of the tool. This is because the ejector system is also in this half of the tool and it would be sensible to have as much of the form as possible to be machined in this half in the same operation. This minimises the matching up of cavity forms in the two separate halves of the tool. If we split the tool at **B**, we will have achieved this with **X** in the injection half and **Y+Z** in the ejection half.

This is a good procedure to follow in general as, apart from toolmaking considerations, the greater the amount of component in the ejection half; the more likely it will be that the component stays on this side of the tool when it opens. This is clearly essential, as the component must stay on the ejection half of the tool after it opens in order to be ejected.

Therefore, we will split the tool at **B** as shown in Figure 7.4.

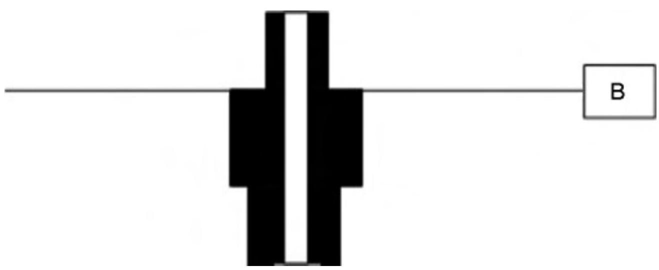

Figure 7.4 Mould split at B

7.4.2 STEP 2: Gating

If the relationship between the hole and all the diameters were important and subject to close concentricity tolerances, a two-plate tool might not be the best choice. This is because gating this part from the side might lead to differential shrinkage and warpage due to the unequal melt flow length. If this were important, a three-plate tool or hot runner tool would be preferred as the part could be gated at the top, providing more equal melt flow lengths.

In this example, this is not the case and we may therefore gate the part on the edge of diameter **Y** at the split line as shown in Figure 7.5 or with a sub gate shown in Figure 7.6.

As we require over half a million parts per year, sub gating is the obvious choice as the parts will be automatically de-gated.

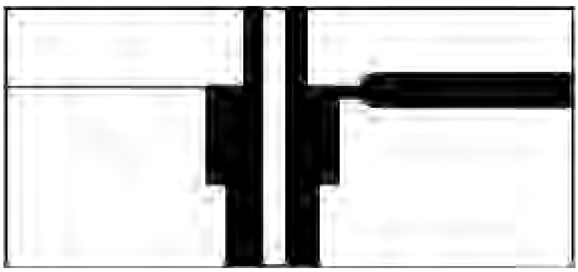

Figure 7.5 Edge gating

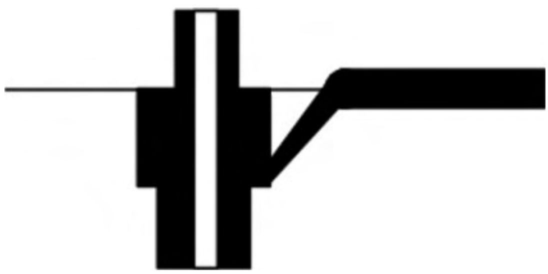

Figure 7.6 Sub gating

7.4.3 STEP 3: Ejection

We could eject this part with pins, with a stripper plate or with sleeve ejectors. It is always preferable to avoid pin gating if the options of stripping or sleeve ejection exist, for three reasons:

1. The ejection area of pins is smaller than in the other methods and we would achieve far greater ejection support with stripping or sleeve ejection. This eliminates the tendency of pins to hob or embed themselves in to the part.

2. For an eight-impression tool we would need 32 rather slender pins for ejection, which entails more toolmaking work and alignment.

3. The relatively slender pins may tend to deflect in the tool during ejection, causing premature wear and breakage.

This leaves the choice between stripper plate or sleeve ejection, so how do we choose between them? Basically the governing factor is the diameter of the part being stripped. Generally, smaller diameters should be sleeve-ejected and larger diameters stripped. In the opinion of the author, the cut-off point should be around 30 mm diameter. This is around the maximum comfortable size for toolmaking and for working with standard mould components.

In this case the diameter of the base of the moulding is 15 mm and therefore we will opt for sleeve ejection (Figure 7.7).

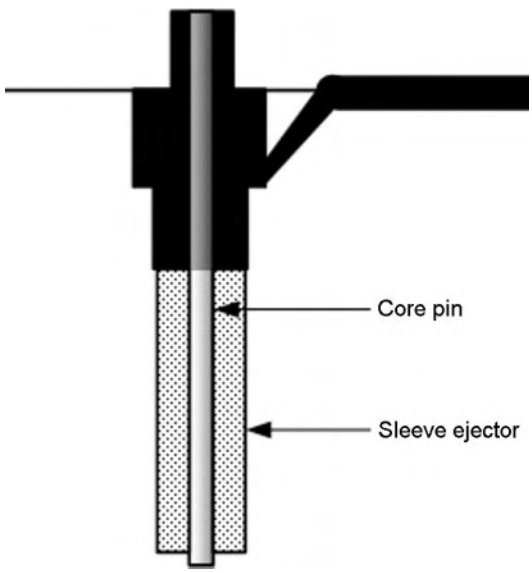

Figure 7.7 Sleeve ejector

7.4.4 STEP 4: Cavity Inserts

We can machine the impressions straight into a plate, but this has two disadvantages:

1. A plate that has cavities sunk directly into it may suffer form warping or distortion during the hardening process.

2. If a cavity suffers any damage during production, it can be very difficult to repair it.

It is therefore common practice to use cavity inserts to avoid these problems since:

(a) It is easier for the toolmaker to work on the inserts, as they are smaller.

(b) If any damage occurs it is much easier to replace an insert.

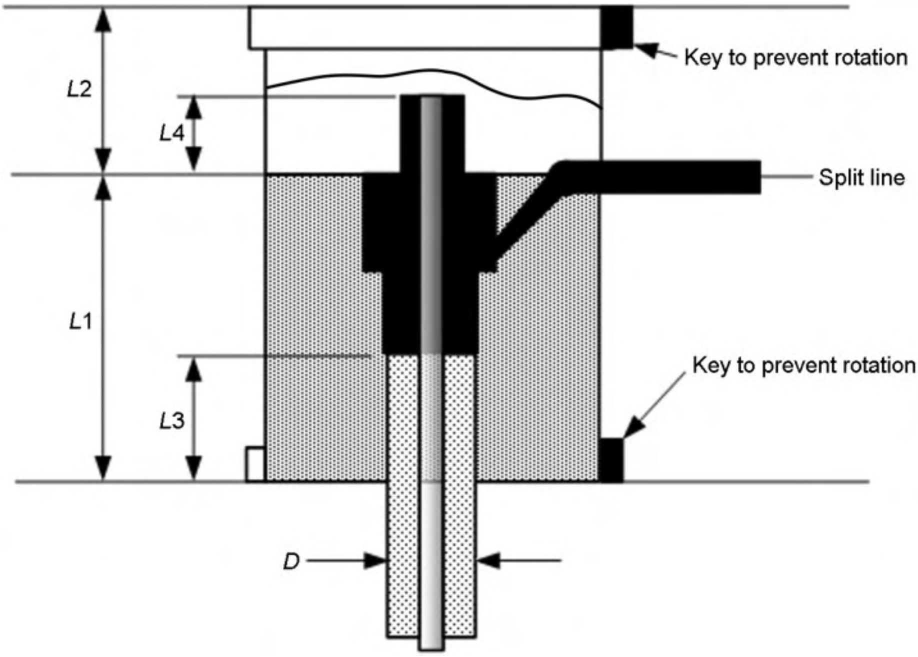

Figure 7.8 Designing the cavity insert

The diameter of the inserts should be large enough where possible to ensure that the whole of the sub gate form lies inside the insert as shown. This makes the spark machining of the sub gate easier and eliminates join lines that may prevent the gate exiting cleanly during ejection. As a rule of thumb, the following guidelines are suggested (see Figure 7.8). Further refinements come with experience.

The length of the lower insert L_1 should be the depth of the part below the split line + 1.5–2 times the length of the sleeve ejector diameter for adequate sliding location L_3.

The length of the upper insert L_2 should be 1.5–2 times the height of the form in the insert L_4.

Note that the lengths L_1 and L_2 automatically determine the plate thickness for the fixed half and ejection half of the tool. However, plates are only available in standard thicknesses from suppliers such as DME, DMS and Hasco. Therefore, the nearest standard plate sizes should be selected to determine final cavity depths.

7.4.5 STEP 5: Venting

If we look at Figure 7.8 we can try to visualise the flow of the melt into the cavity form. The material is initially directed down towards the bottom of the part and will then fill the cavity upwards from this point.

Any air in the cavity will be forced upwards and escape via the split line of the tool until the melt reaches the split line. Once the melt continues beyond this point, it seems that there is no exit path by which the air can escape. This means that the air may become trapped in the fixed half cavity and result in burning of the moulding.

Note that the actual fill pattern will depend on the gate size, speed of injection, injection pressure, tool temperature, and so on. To accurately simulate the most likely fill pattern a computer simulation is preferable.

In this case, however, we have identified the possibility of air entrapment and, if this possibility exists, we should do something about it before the event, and to counter this it is necessary to provide for a route for the air to escape.

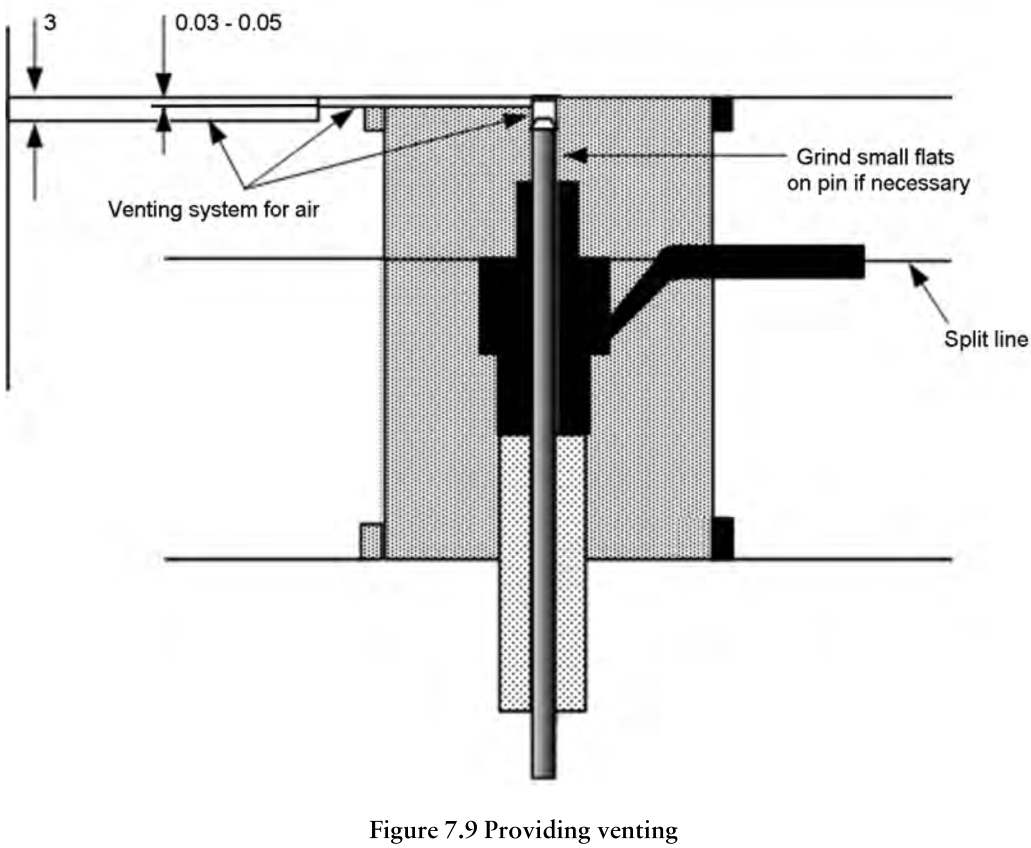

Figure 7.9 Providing venting

The solution kills two birds with one stone. If we extend the core pin upwards into the fixed half cavity insert, we will give extra support to the pin, preventing any tendency for it to deflect because of nonsymmetrical melt pressures. By extending the locating hole for the pin upwards to the top of the insert we also provide an escape route for the air. This ensures the air will exhaust along the top of the fixed half cavity and the cavity retaining plate.

This method works well for moderate injection speeds, but extra provisions will have to be made if high injection speeds are used. This can be achieved by grinding a flat channel approximately 0.03–0.05 mm deep along the cavity retaining plate as shown. The width of the channel is not critical and can be any width within reason. As soon as possible this vent should be opened up to allow the air to exhaust and expand into a larger space, because the air and gases being forced out of the cavity are very hot and can reach very high temperatures. By opening up the vent, the gases will be allowed to expand rapidly and thereby cool rapidly.

With high injection speeds it may also be necessary to grind small flats on the core pin where it locates in the top insert to allow the gases to escape more easily.

7.4.6 STEP 6: Water Cooling

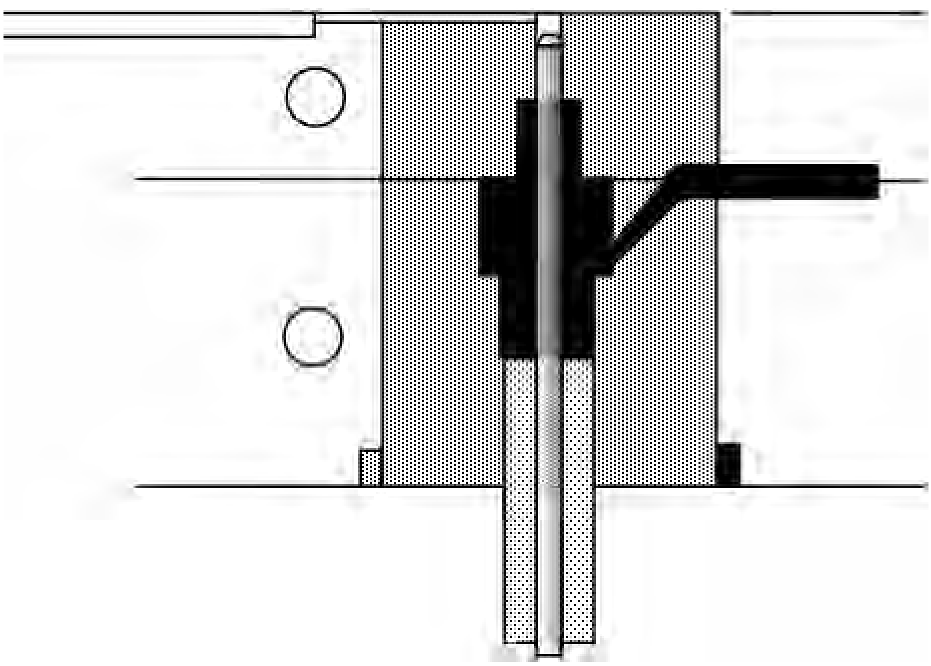

Figure 7.10 Adding water cooling

Temperature control is essential for all mould tools (see Chapter 11). We need to cool the moulding as soon as possible so that we keep moulding cycles within acceptable limits.

In this case we have two options: incorporating cooling channels into the cavity inserts if possible, or putting water channels through the mould plates next to the cavity inserts. Since locating cooling channels into the cavity inserts would be difficult in this case, we will use cooling channels through the mould plates (Figure 7.10).

7.4.7 STEP 7: Impression Centres

We are now in a position to start looking at the impression centres. The first stage is to establish the type of sprue bush we will be using. One is selected from a standard parts catalogue (Chapter 18) and from this we can establish the centre distance of the impressions. The minimum distance between the cavity insert and the sprue bush is around 10 mm (Figure 7.11).

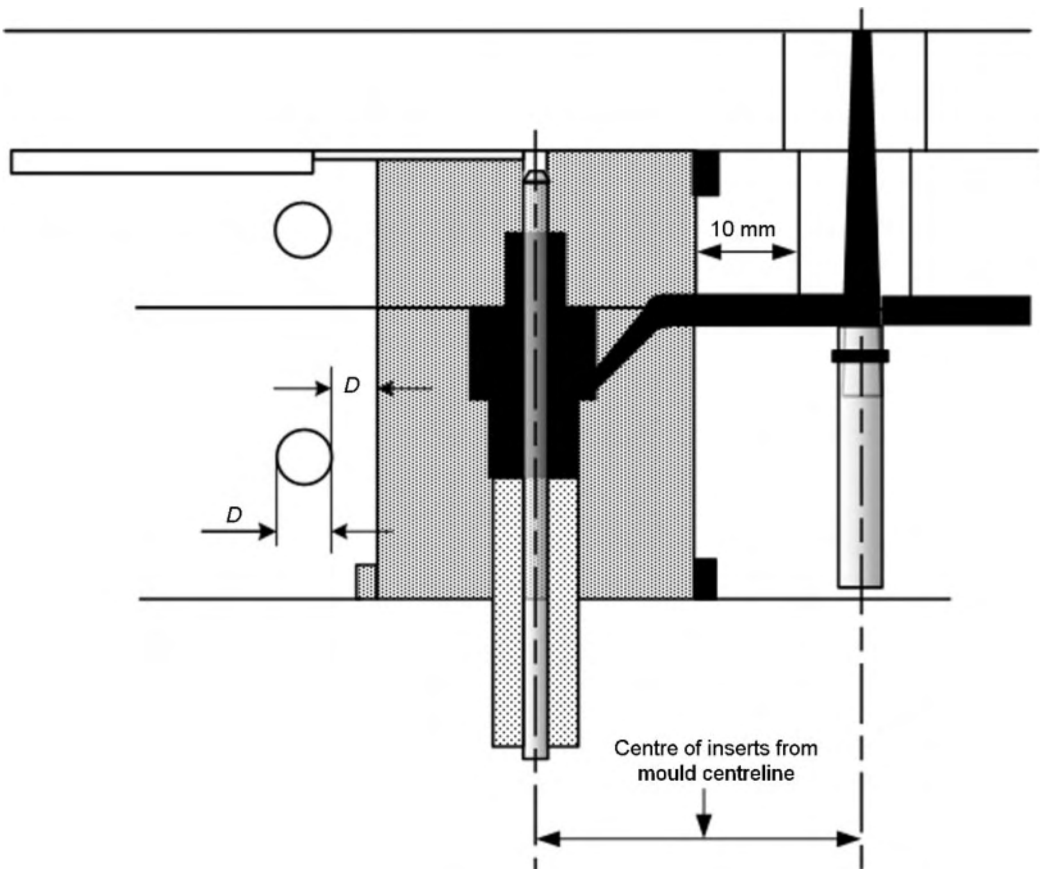

Figure 7.11 Establishing impression centres

7.4.8 STEP 8: Mould Layout

We have now reached the stage where we can determine the rest of the mould layout in plan view. The first stage is to lay out the impressions.

The view shown in Figure 7.12 is of the fixed half of the tool. This enables us to draw in the sprue bush and then arrange the cavity inserts around it. Note: the scale has been reduced for this view.

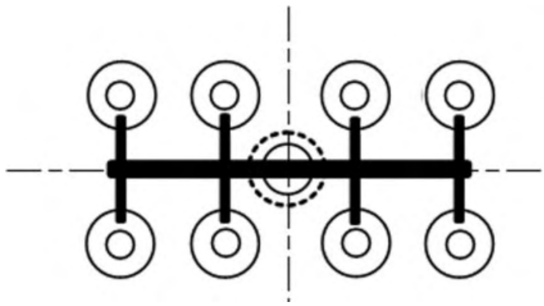

Figure 7.12 Impression layout – fixed half

We can now switch to completing the layout by looking at the ejection half of the tool. Figure 7.13 completes the picture. First the waterways are included as previously determined. Then the limits of the ejector plate can be established just outside the cavity inserts. Next the return pins are drawn in, followed by the guide pillars and screws. Just outside the guide pillars, the outside of the main tool can now be established (based on standard plate sizes). The platen drawing will have to be consulted to determine the positions of the mould fixing holes. A flange is drawn on the outside of the tool to accommodate these.

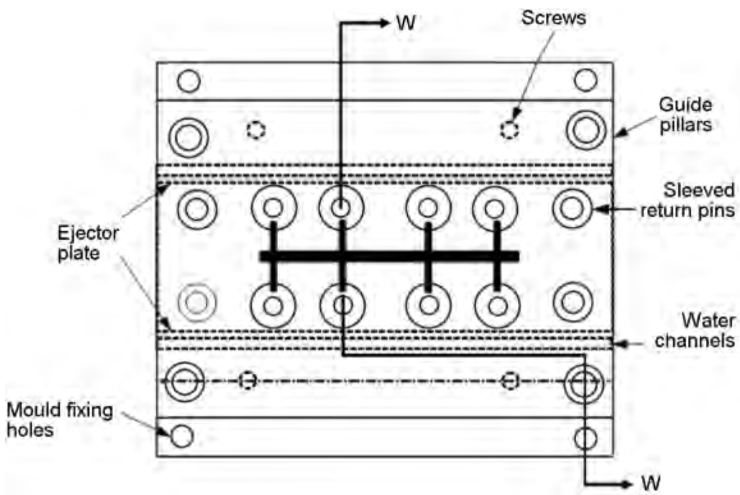

Figure 7.13 Mould layout – ejection half

7.4.9 STEP 9: Main Sectional View

Now that the plan views have been established we can turn our attention to the main sectional view that will complete the basic tool design (Figure 7.14).

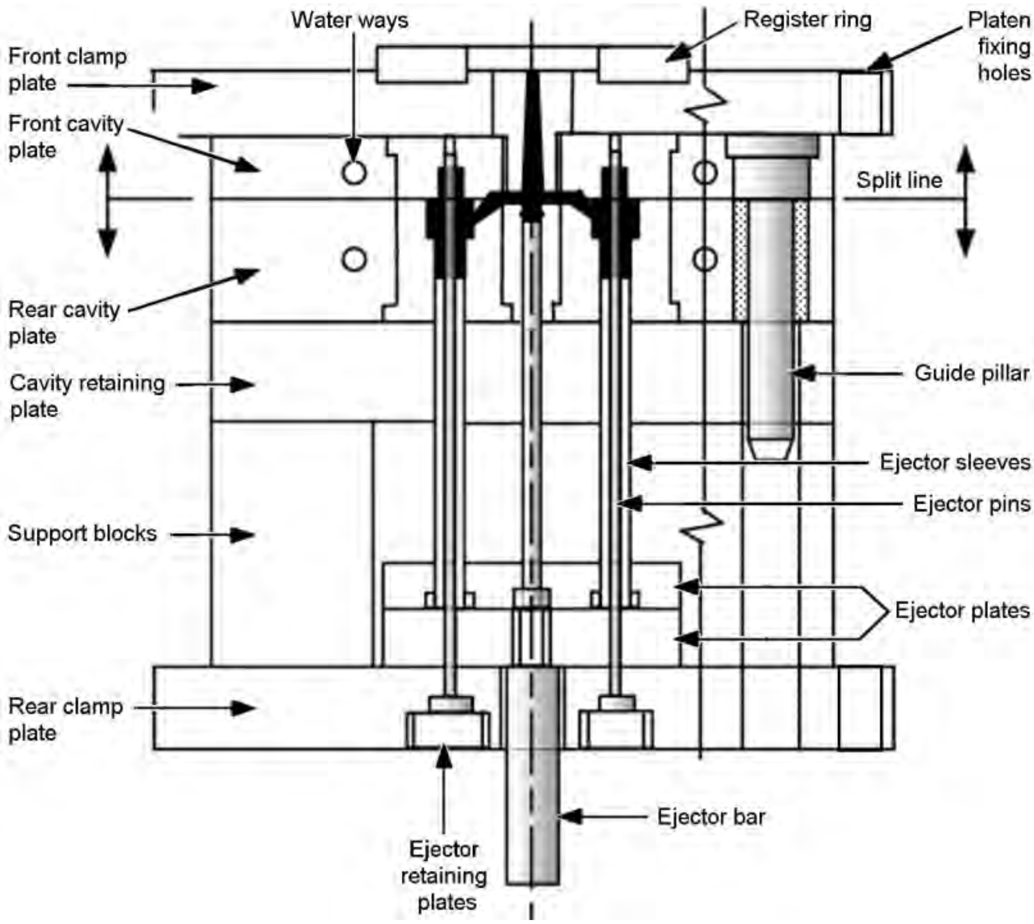

Figure 7.14 Sectional view W-W

This completes the step-by-step design for a basic component but the principles involved are very similar for all mould tools. This design is necessarily a subjective one and other designers may take a different approach, but it has served well for over 2000 designs so far.

Once experience has been gained in this field, designers naturally tend to develop their own personal approach. New designers should take every opportunity to study and understand as many mould general arrangement drawings (GA) as possible. Consult Chapter 9 for more details on two-plate mould refinements.