Alongside the spindle speed, we also have to calculate the appropriate feed rate and cut depth per pass and tool overlap (the cut width per pass). Feed rate denotes the distance covered by the tool per unit time (for example, millimeters per second). For example, if each pass removes 0.5 mm of material, the cut depth per pass is also 0.5 mm. Table 3.2 lists typical manufacturer tool values based on the 3 mm tool we used for our sample calculations.

d	z	v_c		f_z	a_e	a_p
		min	max			
3	3	250	350	0.05	1.0 · d	1.0 · d

Tab. 3.2 The basic parameters for our sample 3 mm tool

The cut-width value a_e denotes the maximum depth to which the tool can engage with the material during a single face-milling pass, and is often given as a factor of the diameter of the end mill. These values vary for different raw materials. The tool table 3.2 is based on has a maximum value of 3 mm. The cutting-depth value a_p denotes the maximum depth the mill can cut during a single pass—in this case, also a maximum of 3 mm. In other words, this particular tool can remove material with a maximum square area of 3 × 3 mm per pass.

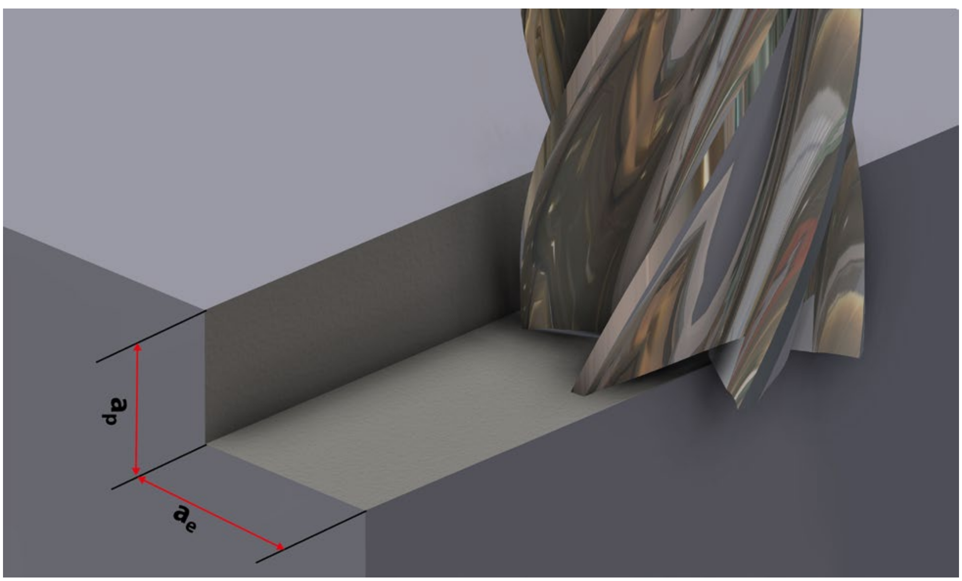

Fig. 3.4 Cut width and depth

The feed rate is based on the feed-per-tooth value f_z. This denotes the cut depth produced by a single tooth during one revolution—in this case 0.05 mm. If you are using a three-tooth mill, this value has to be multiplied by three to account for the three cuts it makes per revolution.

As we already know the speed range from our previous calculation, the feed rate can be derived from the feed-per-tooth value using the following formula:

$$f = n \cdot f_z \cdot z \frac{\text{mm}}{\text{min}}$$

Our example gives us the following values:

$$f_{min} = 26000 \cdot 0,05 \cdot 3 \frac{\text{mm}}{\text{min}} = 3900 \frac{\text{mm}}{\text{min}}$$

$$f_{max} = 37000 \cdot 0,05 \cdot 3 \frac{\text{mm}}{\text{min}} = 5550 \frac{\text{mm}}{\text{min}}$$

We now have all the basic values we need. At minimum spindle speed, you can remove 3×3 mm of material at a feed rate of 3,900 mm/min, which increases to 5,550 mm/min at maximum spindle speed.

If the resulting spindle speed range exerts too much load on the machine, select a smaller cut-per-pass value in both directions. If that doesn't have the desired effect, you can select a lower feed-per-tooth value. This means that the mill cuts less deeply into the material and operates outside of the range specified by the manufacturer. Further reducing the feed-per-tooth setting utilizes less of the cutting edge, which in turn means the tool has to cut more often to cover the same distance and blunts much faster as a result. The force exerted on the tool is also distributed over a smaller area, which contributes to faster wear.

Figure 3.5 shows how the forces aggregate at the tip of the cutting edge when you apply the same load to a smaller area. This means that more force is applied to less of the tool's surface.

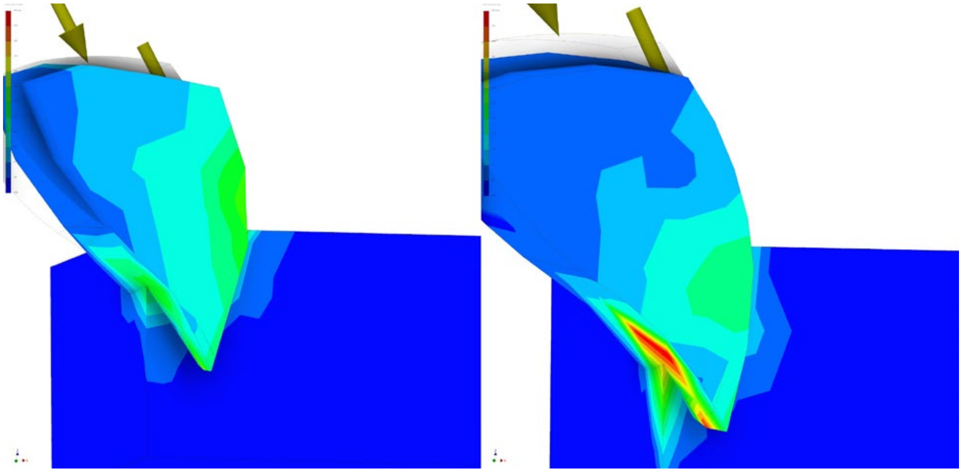

Fig. 3.5 An FEA simulation based on different feed-per-tooth values

However, you cannot simply continue to reduce feed rate or spindle speed. If the load is still too great at too low a setting, you can reduce the cut width or the cut depth per pass to compensate. This means you have to make more passes, but the cutting speed remains the same.

Impossible Settings

In some cases, it is simply impossible to make optimum settings—for example, if the spindle cannot rotate fast or slow enough, or if the machine isn't rigid enough to support the required feed rate. In such cases, the only alternative is to use a different tool or carry on with the tools you have using less-than-ideal settings.

Even if you have to work out of range, it is useful to know the optimum parameters so you can at least tweak things in the appropriate direction. Always record the settings for combinations of tools and materials that work well together, but make a note of poor tools, tricky materials, and settings that produce poor results too. Keeping a record will help you to avoid repeating your mistakes, and makes purchasing suitable materials and tools easier as you progress.

3.1.5 Cooling and Lubrication

The forces involved in milling produce a lot of heat. If you exceed the recommended cutting speed or continue using a blunt or clogged cutter, you run the risk of overheating or breaking the tool. It is essential to ensure that cutting tools generate as little heat as possible.

Heat generation can be reduced passively by using coated tools, or actively by applying suitable coolants or lubricants while you work. The processes described