

CNC Milling for Makers



Christian Rattat began his career on a Commodore Amiga 2000 in the 1980s and has been developing software ever since. He works for large corporations in Microsoft and Unix environments, and also builds and implements software for microcontroller-based applications. In his free time, he builds and flies multirotors—a hobby that gives him plenty of excuses to experiment with CNC routers and 3D printers.

Christian Rattat

CNC Milling for Makers

Basics – Techniques – Applications

Translated by Jeremy Clout



dpunkt.verlag

CNC Milling for Makers
Christian Rattat
christian@rattat.net

English translation: Jeremy Cloot
Editor: Dr. Michael Barabas
Copy-Editing: Patricia Pane
Layout: Ulrich Borstelmann
Production: Susanne Bröckelmann
Cover design: Helmut Kraus, www.exclam.de
Printer: Media-Print Informationstechnologie, Paderborn, Germany

This book is a co-publication by dpunkt.verlag and Rocky Nook

ISBN 978-3-86490-472-1 (dpunkt.verlag)
ISBN 978-1-68198-302-8 (Rocky Nook)

Copyright of the English edition © 2017 dpunkt.verlag GmbH
Copyright of the original German edition © 2016 dpunkt.verlag GmbH
Wieblinger Weg 17
69123 Heidelberg

Rocky Nook Inc.
1010 B Street, Suite 350
San Rafael, CA 94901
USA

Distributed in the U.S. by Ingram Publisher Services
Distributed in the UK and Europe by Publishers Group UK and dpunkt.verlag GmbH

All rights reserved. No part of the material protected by this copyright notice may be reproduced or utilized in any form, electronic or mechanical, including photocopying, recording, or by any information storage and retrieval system, without written permission of the publisher.

Many of the designations in this book used by manufacturers and sellers to distinguish their products are claimed as trademarks of their respective companies. Where those designations appear in this book, and dpunkt.verlag/Rocky Nook was aware of a trademark claim, the designations have been printed in caps or initial caps. All product names and services identified throughout this book are used in editorial fashion only and for the benefit of such companies with no intention of infringement of the trademark. They are not intended to convey endorsement or other affiliation with this book.

While reasonable care has been exercised in the preparation of this book, the publisher and author assume no responsibility for errors or omissions, or for damages resulting from the use of the information contained herein or from the use of the discs or programs that may accompany it.

This book is printed on acid-free paper.
Printed in Germany

5 4 3 2 1 0

Content Overview

1	Introduction	1
2	Your Own CNC Router	43
3	Milling Tools	139
4	2D Milling in Practice	157
5	Machine Upgrades	193
6	Advanced Milling Techniques	233
7	Looking Ahead	265
	Glossary	269
	Index	277

Table of Contents

1	Introduction	1
1.1	What Is Milling?	2
1.2	Gantry Milling for Hobbyists	6
1.2.1	The Machine Table	8
1.2.2	The Milling Motor	9
1.2.3	Feed Rate	12
1.2.4	All About Milling Tools.	15
1.2.5	Features and Add-ons	21
1.3	Raw Materials.	31
1.4	CNC Milling	31
1.4.1	CAD	32
1.4.2	CAM.	35
1.4.3	Machine Control Systems	38
1.5	Alternatives to Milling.	39
1.6	Personal Safety and Risks	40
1.6.1	Motor and Cutters	40
1.6.2	Electricity	41
1.6.3	Fire Hazards	41
1.6.4	Hazardous Materials.	41
1.6.5	Safety in the Workplace.	42
2	Your Own CNC Router	43
2.1	Choosing the Machine for this Book	45
2.1.1	Inventables X-Carve	48
2.1.2	Stepcraft-2/600	49
2.1.3	Other Models	51
2.1.4	Making a Decision	52
2.2	Assembling the Kit	53
2.3	Step 1: Assembling the X-Z Connector	56

2.4	Step 2: Mounting the Z Axis	68
2.5	Step 3: Assembling the X Lead Screw and the Sides of the Gantry . . .	74
2.6	Step 4: Assembling the Gantry	82
2.7	Step 5: Assembling the Main Frame	89
2.8	Step 6: Final Mechanical Assembly	99
2.9	Step 7: Installing the Electronics	102
2.10	Step 8: Assemble the Machine Table (5 minutes)	110
2.11	Step 9: Installation and Startup (30 minutes)	111
	2.11.1 Grease the Rails and the Lead Screws	111
	2.11.2 Install the Software	112
	2.11.3 Prepare to Align the Lead Screws	113
	2.11.4 Connect Your Machine	113
	2.11.5 Align the Lead Screws	115
2.12	Alternative Software	117
	2.12.1 Installing the Hardware	118
	2.12.2 Installing and Configuring UCCNC	121
	2.12.3 Running a Test Job	126
2.13	Your First Workpiece	131
3	Milling Tools	139
3.1	Milling Tools	140
	3.1.1 Teeth and Flutes	141
	3.1.2 Cutting Length and Shank Types	142
	3.1.3 End Mill Materials	143
	3.1.4 Cutting Speed and Feed Rate	144
	3.1.5 Cooling and Lubrication	148
3.2	Which Tool for Which Material?	151
	3.2.1 Solid Carbide End Mills	151
	3.2.2 Unusual Types of Cutters	153
	3.2.3 Long Mills	154
	3.2.4 Forming Cutters	154
3.3	Upcut and Downcut Milling	155
3.4	Tool Maintenance	155

4	2D Milling in Practice	157
4.1	Clamping the Workpiece	160
4.2	Softwood	162
4.2.1	Example #1: MDF	163
4.2.2	Example #2: Poplar Plywood (using inappropriate parameters)	164
4.2.3	Example #3: Poplar Plywood (using appropriate parameters)	166
4.3	Hardwood	168
4.3.1	Example #1: Solid Beech (right-hand spiral)	168
4.3.2	Example #2: Solid Beech (left-hand spiral)	169
4.4	Perspex and Polycarbonates	170
4.4.1	Example #1: Perspex	171
4.5	Extruded Foam	173
4.5.1	Example #1: PVC	173
4.6	Engraving Plastic	175
4.6.1	Example #1: Engraving Plastic	176
4.7	Aluminum	177
4.7.1	Example #1: Aluminum	178
4.7.2	Example #2: Facing an Aluminum Blank	181
4.8	Brass	183
4.8.1	Example #1: Brass	184
4.9	Aluminum Dibond	186
4.9.1	Example #1: Aluminum Dibond	187
4.10	Glass	189
5	Machine Upgrades	193
5.1	Enclosure	194
5.2	Vacuum System	199
5.2.1	Centrifugal (Cyclone) Separator	200
5.2.2	Vacuum and Connectors	201
5.2.3	Dust Extraction in the Enclosure and at the Tool	202
5.3	Aluminum T-slot Machine table	206
5.4	Clamps and Clamping Blocks	207

5.5	Machine Vise	212
5.6	Tool Length Sensor	214
5.6.1	Fitting a Tool Length Sensor	215
5.6.2	Configuring the Tool Length Sensor in WinPC-NC	216
5.6.3	Using and Adjusting the Sensor	219
5.6.4	Using the tool length sensor with UCCNC	220
5.7	Underwater Milling Using a Water Bath	225
5.8	Additional Emergency Stop Button	229
6	Advanced Milling Techniques	233
6.1	Isolation Milling	234
6.1.1	How to Configure Eagle	237
6.1.2	Milling a PCB Using WinPC-NC USB	242
6.2	Underwater Milling	245
6.3	2.5D Milling	249
6.3.1	CAD	250
6.3.2	CAM	253
6.3.3	Milling	259
6.3.4	Slots, Pockets, and Facing	262
7	Looking Ahead	265
7.1	Replacement Parts	266
	Glossary	269
	Index	277

Foreword by Stepcraft

Dear Reader,

The invention of Numerical Control (NC) technology in the 1950s and the Computer Numerical Control (CNC) technology that followed in the 1970s have made it increasingly simple for industrial companies to quickly produce runs of identical, high-quality, complex parts. Today, the availability of CNC technology to small firms and private users opens up a whole new world of technical and creative possibilities.

In spite of improvements in technology, the knowledge required to produce CAD/CAM models and transform them into components using software commands often appears too much to master, especially for beginners.

In this book, Christian Rattat gives you all the help you need purchasing your own machine, setting it up, and creating your own objects. He combines firm background knowledge with a wealth of tips and tricks on machines and machining, creating the ideal jumping-off point for your own CNC exploits. His enthusiasm is infectious and proves that CNC technology isn't rocket science. Above all, alongside the countless possibilities CNC machining provides, the thing it offers more than anything else is fun!

Have a great time discovering CNC for yourself.

Markus Wedel and Peter Urban



Stepcraft GmbH & Co. KG
An der Beile 2
58708 Menden
Germany
Phone: +49 (0) 23 73 – 179 1160
info@stepcraft-systems.com
www.stepcraft-systems.com

Author's Foreword

Milling and lathing are the cream of the techniques available to today's model makers and hobbyists. They make it possible to produce metal, wood, plastic, and composite components of sufficient strength and precision for today's demanding projects. Combined with computer-based control and 2D or 3D models, they provide an almost limitless world of creative possibilities.

CNC technology has shifted the knowledge required to work milling machines toward the use of preprogrammed applications and other computer-based skills. Increasingly powerful software and hardware have shifted the emphasis from the "how" of machining to the "what." Once you have produced a working model of a component, producing the finished part is often a trivial final step. Milling a part takes only a few minutes and components can be duplicated as often as you like at a constant level of quality.

The two greatest hurdles that prevent many people from acquiring their own CNC machine are the cost and fear of the complex technology involved:

- Do I have the right skills?
- Where do I start?
- Which programs do I need?
- What gear do I need?
- How does the process of turning an idea into a finished part work?
- How much does all this cost?
- Which tools and settings do I need for which materials?

If you can't answer these questions, it can be hard to know where to start. Websites and online forums only really help if you have a good idea of which answers you are looking for. The huge range of information and opinions available makes it virtually impossible for beginners to filter out the input they need and, the deeper you delve into the subject, the more expensive it appears to become. You are sure to find people who claim it's impossible to even start milling without linear guides, a high-frequency spindle, a ball screw, and a whole bunch of other things too, and you might just end up thinking you'd be better off using a nail file. This kind of hocus-pocus is great for people who make and sell milling machines but is based on a flawed approach. Anyone with a little

experience will select such a sophisticated tool based not on its specifications, but rather on what it is actually required to do.

These are the challenges I faced as a model maker, so I began with the part of the process that costs little or nothing: creating models. Using SketchUp, Blender, and other similar programs is a quick and relatively easy way to learn how to create 3D models, and it is essential to become really skilled at modeling if you want to progress from extruding or cutting basic samples to creating top-notch components.

You will only be able to produce adequate models if you properly understand complex geometry, sizing techniques, and other design aids. I gained further insight into the processes involved by getting a friend to create parts from my models using his machine. I quickly learned the principles involved and decided to purchase my own machine. This is a great way to get started, and online forums are full of people who are happy to mill a couple of models for you, usually for a small fee.

It took me a total of about two months to get up and running, and I can now model and mill a lot of the parts I need within an hour or two. Duplicating parts that I have already made often takes just a few minutes.

My experiences have taught me a systematic approach to CNC machining from the ground up. This book addresses all the essential elements of the process and, with a little effort on your part, will help you to quickly start making your own CNC machined parts.

The Book Website

The website that accompanies this book can be found at <http://cncbuch.de/en/>. It includes notes and corrections to the text and, as an appendix to chapter 2, videos explaining how to set up a Stepcraft Series 2 machine.

I will continue to update the site with tips and useful additions, so make sure you check in from time to time.

Thanks

First and foremost, I would like to thank my editor Ursula Zimpfer; Peter Griwatsch, Maik Schmidt, and Wolfgang Lindner for their reviews; and Anke Eltermann, Markus Wedel, and Peter Urban at Stepcraft for their professional and technical support. Thanks also go to Susan Grey for her moral support; to Michael, Miriam, Sabrina, Vanessa, and all the other staff at dpunkt, who made this book possible. And, last but not least, thanks to Alfred for the best schnitzels in the world.

For Reference

Visit <https://www.hoffmann-group.com/US/en/usa/service/downloads> to download the excellent *Machining Handbook* PDF, which includes descriptions of a huge range of machining tools. The section on milling tools begins on page 242.

Upcut and Synchronous Milling

Milling processes also differ according to whether the rotational direction of the tool is the same as or opposed to the feed direction. During upcut milling, the workpiece is fed toward the tool, whereas synchronous (or “downcut”) milling feeds the material away from the tool during the cut.

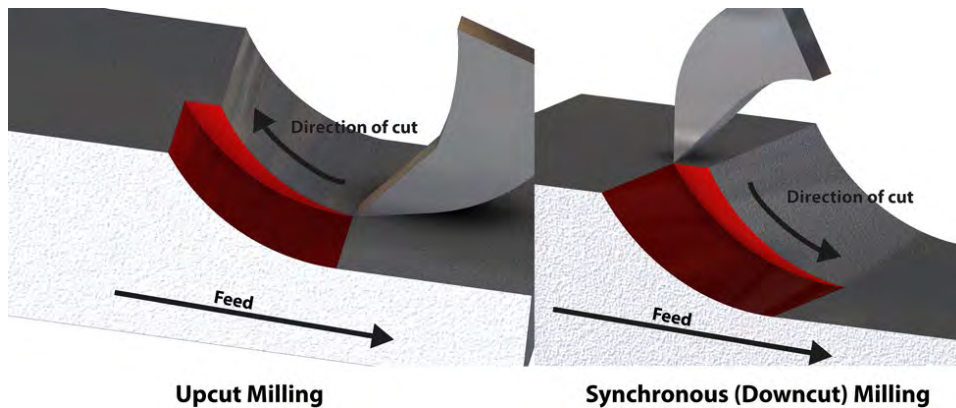


Fig. 1.13 Upcut and synchronous (downcut) milling

Upcut milling forces the workpiece toward the cutting edge, producing increased load and exerting lateral force on the cutter. These forces reach their maximum levels just before the cutting edge exits the workpiece surface. If the chip breaks at this point, the forces involved dissipate immediately, producing excessive compression of the material and an uneven cut. This quick alternation of load often causes vibrations in the tool, resulting in so-called chatter marks.

During downcut milling, the tool and workpiece move in the same direction. The force exerted by the cutter is greatest at the moment it hits the surface and decreases in the course of the cut. Lateral force is produced when the cutter enters the workpiece—i.e., the moment when the cutting edge is more or less perpendicular to the direction of feed. The cutter is under less load when it exits the cut. This reduces vibration but requires the lead screw to be virtually play-free. If this is not the case, the tool will drag the workpiece during the cut until

the play is exhausted and the tool once again enters the workpiece with a jolt. This negates the potential advantages of the downcut process.

Types of Milling Bits

Milling bits (or “cutters”) are generally classed as “roughing” or “finishing.” Finishing bits remove less material and provide a higher-quality finish, while roughing bits are used to quickly remove large quantities of material. These processes are equivalent to the results produced by the rough and fine files used in industrial manufacturing.

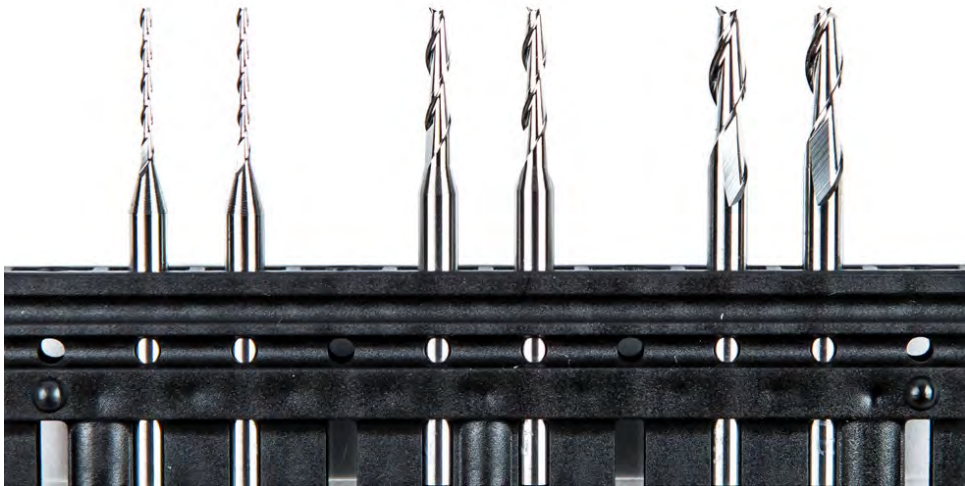


Fig. 1.14 1 mm, 2 mm, and 3 mm solid carbide fishtail bits

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	$1.0 \cdot d$	$1.0 \cdot d$

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

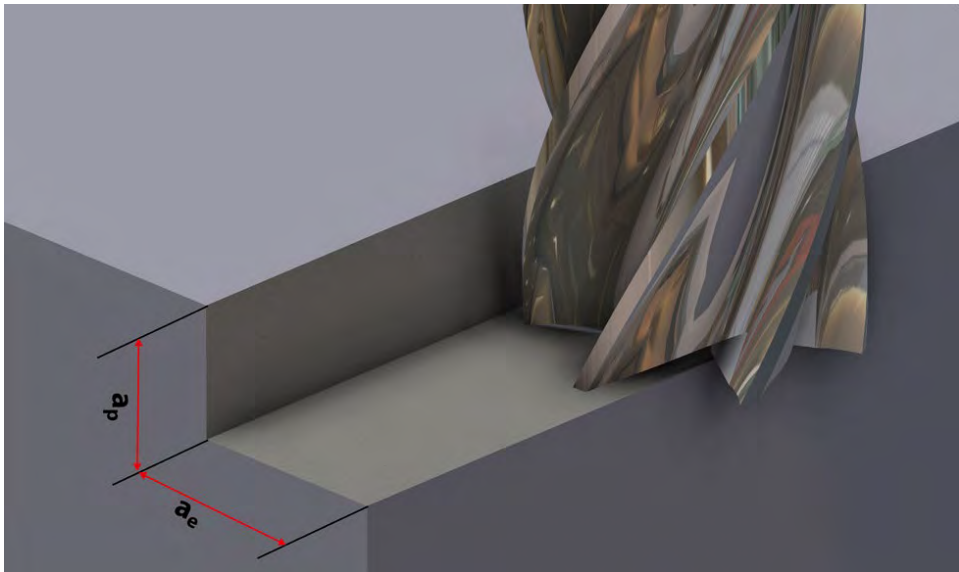


Fig. 3.4 Cut width and depth

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.

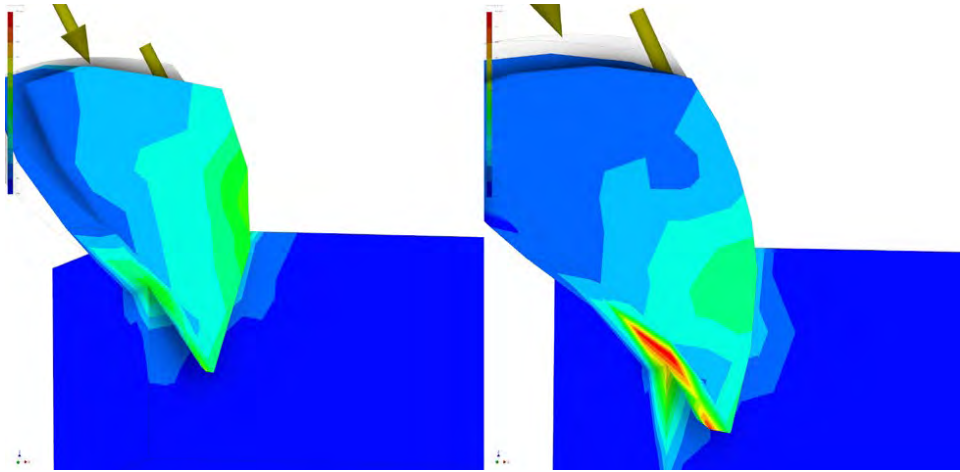


Fig. 3.5 An FEA simulation based on different feed-per-tooth values

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

6

Advanced Milling Techniques

This chapter explains in detail how to use a gantry router to perform:

- PCB Isolation milling
- Underwater milling
- 2.5D milling using CAM software



6.1 Isolation Milling

Most hobbyists who make their own printed circuit boards (PCBs) create them using etching techniques. The simplest way to do this is to take a copper-plated board, mask the areas you don't want to etch, and place the board in an etching solution of ferric chloride or sodium persulfate. The chemical then etches away the unwanted parts of the copper plating. Both of these chemicals represent a health and environmental risk, and usually have to be disposed of professionally by a toxic waste specialist.

If you don't require the high-precision results that the etching process offers, you can use your CNC router to cut and drill PCBs. This approach also enables you to produce two-sided PCBs. A great place to start is using the Eagle software package (www.autodesk.com/products/eagle/overview), which is free for private use. If you need to create larger PCBs (the free version only works for designs up to 80×100 mm), you can upgrade to the fully featured subscription-based Standard or Premium versions, or simply switch to a different application. With a couple of simple tweaks, Eagle can be used to produce output that you can mill directly using WinPC-NC.

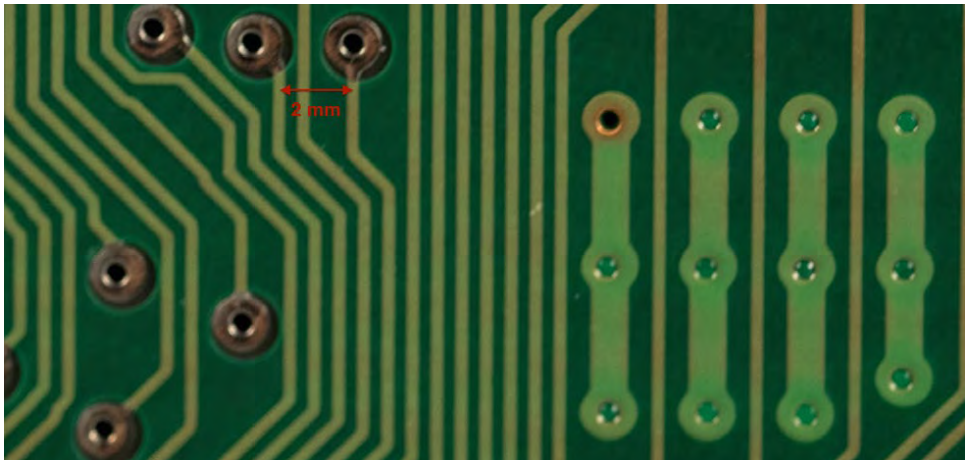


Fig. 6.1 One of the challenges involved in milling a circuit board

The real challenge involved in milling PCBs is getting your router to produce sufficiently fine structures. Integrated circuits often require traces that run between two pins, and standard PCB layouts leave just 2.54 mm (or sometimes 1.27 mm) between pins. Unfortunately, this doesn't mean you have 2.54 mm of space to play with. Soldering pins to the circuit board requires 0.8 mm holes with an additional copper surround, which leaves an effective distance of just 1.5 mm between pins, which in turn has to be milled to produce a trace and two isolating surfaces. In other words, you have to set up your machine to effectively remove copper strips that are a maximum of 0.1–0.2 mm wide.

It is obvious you can't use a conventional end mill to cut such narrow traces. Furthermore, a 0.1 mm mill used at 20,000 rpm provides a cutting speed of just 6 m/min, whereas copper requires a cutting speed closer to 100 m/min. Even at 100,000 rpm, you would only reach a cutting speed of 31 m/min, which is simply not enough.

To get going, we need to make a compromise or two. The copper layer on a standard PCB is 0.035 mm thick, and the carrier layer beneath should remain unharmed if possible. If the board flexes due to uneven clamping, you won't be able to cut at a consistent depth, and setting the workpiece Z zero point is critical, as you will be cutting to a maximum final depth of between 0.1 mm and 0.3 mm.

The tool of choice for milling PCBs is an engraving bit or "graver." Gravers have V-shaped tips and are available with tip angles of 10 degrees upward and thicknesses of 0.1 mm and more. The width of the tip dictates the minimum width of the traces you can cut, while the angle of the tip determines how much wider the cut becomes the deeper you plunge into the material.



Fig. 6.2 A 60° 0.2 mm graver

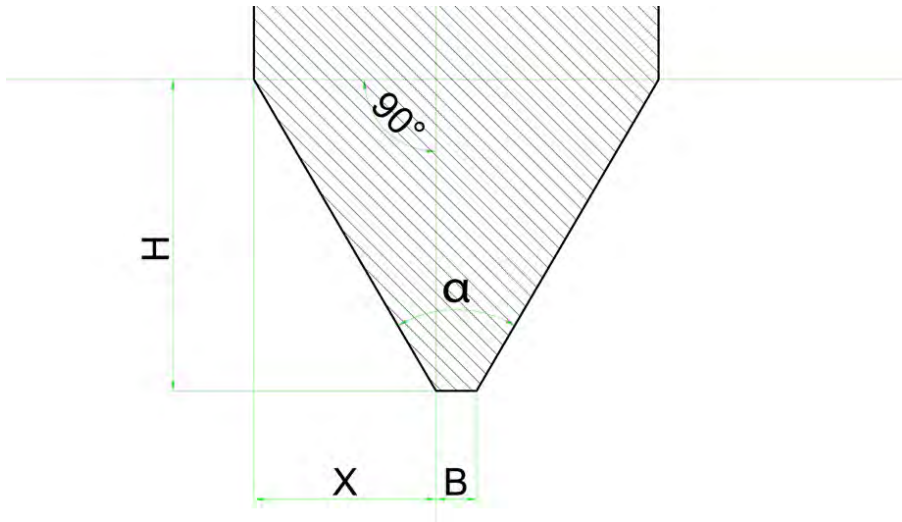


Fig. 6.3 The geometry of the tip of an engraving bit

To calculate the width of the cut, you need to know the opposite side, X , of the right-angled triangle formed by half the included angle of the tool tip, α (in this case, 60 degrees), and the adjacent side H . The cut width is then equal to $2 \times X$ plus the width of the tip.

$$B_{\text{Fräsbahn}} = B_{\text{Spitze}} + 2 \cdot H \cdot \tan \frac{\alpha}{2}$$

A 60-degree graver with a 0.1 mm tip used at a cut depth of 0.15 mm will produce the following cut width:

$$B_{\text{Fräsbahn}} = 0,1 \text{ mm} + 2 \cdot 0,15 \text{ mm} \cdot \tan \frac{60^\circ \cdot \pi}{180} = 0,27 \text{ mm}$$

To work out the tan, the angle usually has to be converted from degrees to radians by multiplying it by $\pi/180$, as these are the units used by most spreadsheets and pocket calculators.

The resulting value equates to the width of the cut at the surface of the board. If the copper layer is 0.035 mm thick, the surface width of the cut will be reduced to just 0.23 mm.