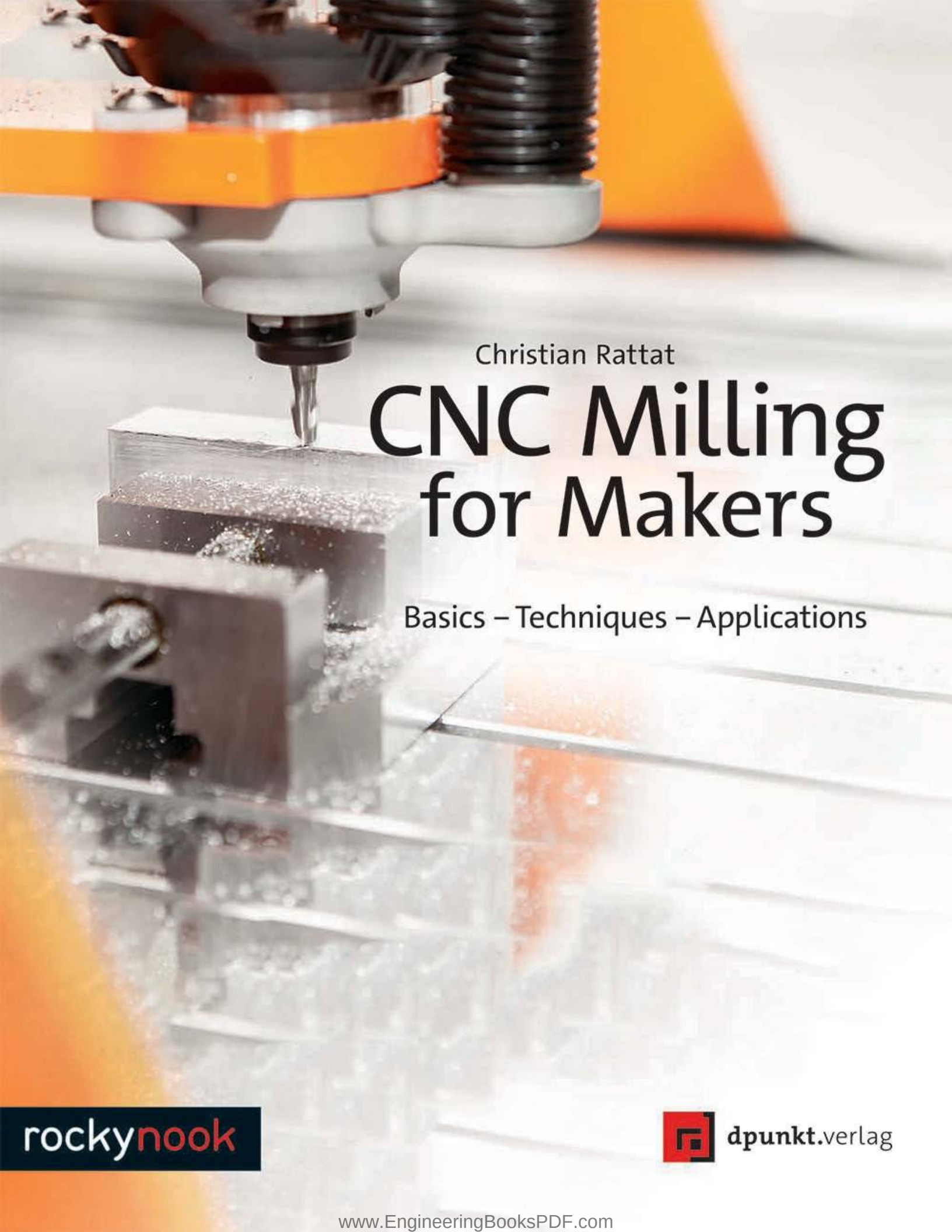




Christian Rattat

CNC Milling for Makers

Basics – Techniques – Applications

rockynook



dpunkt.verlag



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 Clood



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: M.P. Media-Print Informationstechnologie GmbH, 33100 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**
- 2 Your Own CNC Router**
- 3 Milling Tools**
- 4 2D Milling in Practice**
- 5 Machine Upgrades**
- 6 Advanced Milling Techniques**
- 7 Looking Ahead**
- Glossary**

Table of Contents

1 Introduction

- 1.1 What Is Milling?
- 1.2 Gantry Milling for Hobbyists
 - 1.2.1 The Machine Table
 - 1.2.2 The Milling Motor
 - 1.2.3 Feed Rate
 - 1.2.4 All About Milling Tools
 - 1.2.5 Features and Add-ons
- 1.3 Raw Materials
- 1.4 CNC Milling
 - 1.4.1 CAD
 - 1.4.2 CAM
 - 1.4.3 Machine Control Systems
- 1.5 Alternatives to Milling
- 1.6 Personal Safety and Risks
 - 1.6.1 Motor and Cutters
 - 1.6.2 Electricity
 - 1.6.3 Fire Hazards
 - 1.6.4 Hazardous Materials
 - 1.6.5 Safety in the Workplace

2 Your Own CNC Router

- 2.1 Choosing the Machine for this Book

- 2.1.1 Inventables X-Carve
 - 2.1.2 Stepcraft-2/600
 - 2.1.3 Other Models
 - 2.1.4 Making a Decision
- 2.2 Assembling the Kit
- 2.3 Step 1: Assembling the X-Z Connector
- 2.4 Step 2: Mounting the Z Axis
- 2.5 Step 3: Assembling the X Lead Screw and the Sides of the Gantry
- 2.6 Step 4: Assembling the Gantry
- 2.7 Step 5: Assembling the Main Frame
- 2.8 Step 6: Final Mechanical Assembly
- 2.9 Step 7: Installing the Electronics
- 2.10 Step 8: Assemble the Machine Table (5 minutes)
- 2.11 Step 9: Installation and Startup (30 minutes)
 - 2.11.1 Grease the Rails and the Lead Screws
 - 2.11.2 Install the Software
 - 2.11.3 Prepare to Align the Lead Screws
 - 2.11.4 Connect Your Machine
 - 2.11.5 Align the Lead Screws
- 2.12 Alternative Software
 - 2.12.1 Installing the Hardware
 - 2.12.2 Installing and Configuring UCCNC
 - 2.12.3 Running a Test Job
- 2.13 Your First Workpiece

3 Milling Tools

- 3.1 Milling Tools
 - 3.1.1 Teeth and Flutes
 - 3.1.2 Cutting Length and Shank Types
 - 3.1.3 End Mill Materials

- 3.1.4 Cutting Speed and Feed Rate
 - 3.1.5 Cooling and Lubrication
- 3.2 Which Tool for Which Material?
 - 3.2.1 Solid Carbide End Mills
 - 3.2.2 Unusual Types of Cutters
 - 3.2.3 Long Mills
 - 3.2.4 Forming Cutters
- 3.3 Upcut and Downcut Milling
- 3.4 Tool Maintenance
- 4 2D Milling in Practice**
- 4.1 Clamping the Workpiece
- 4.2 Softwood
 - 4.2.1 Example #1: MDF
 - 4.2.2 Example #2: Poplar Plywood (using inappropriate parameters)
 - 4.2.3 Example #3: Poplar Plywood (using appropriate parameters)
- 4.3 Hardwood
 - 4.3.1 Example #1: Solid Beech (right-hand spiral)
 - 4.3.2 Example #2: Solid Beech (left-hand spiral)
- 4.4 Perspex and Polycarbonates
 - 4.4.1 Example #1: Perspex
- 4.5 Extruded Foam
 - 4.5.1 Example #1: PVC
- 4.6 Engraving Plastic
 - 4.6.1 Example #1: Engraving Plastic
- 4.7 Aluminum
 - 4.7.1 Example #1: Aluminum
 - 4.7.2 Example #2: Facing an Aluminum Blank

- 4.8 Brass
 - 4.8.1 Example #1: Brass
- 4.9 Aluminum Dibond
 - 4.9.1 Example #1: Aluminum Dibond
- 4.10 Glass

5 Machine Upgrades

- 5.1 Enclosure
- 5.2 Vacuum System
 - 5.2.1 Centrifugal (Cyclone) Separator
 - 5.2.2 Vacuum and Connectors
 - 5.2.3 Dust Extraction in the Enclosure and at the Tool
- 5.3 Aluminum T-slot Machine table
- 5.4 Clamps and Clamping Blocks
- 5.5 Machine Vise
- 5.6 Tool Length Sensor
 - 5.6.1 Fitting a Tool Length Sensor
 - 5.6.2 Configuring the Tool Length Sensor in WinPC-NC
 - 5.6.3 Using and Adjusting the Sensor
 - 5.6.4 Using the tool length sensor with UCCNC
- 5.7 Underwater Milling Using a Water Bath
- 5.8 Additional Emergency Stop Button

6 Advanced Milling Techniques

- 6.1 Isolation Milling
 - 6.1.1 How to Configure Eagle
 - 6.1.2 Milling a PCB Using WinPC-NC USB
- 6.2 Underwater Milling
- 6.3 2.5D Milling
 - 6.3.1 CAD

6.3.2 CAM

6.3.3 Milling

6.3.4 Slots, Pockets, and Facing

7 Looking Ahead

7.1 Replacement Parts

Glossary

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*](mailto:info@stepcraft-systems.com)

[*www.stepcraft-systems.com*](http://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.

1 Introduction

The first milling machine in the world was invented by Eli Whitney around two hundred years ago. It was capable of manual and partially automated milling. With increasing industrialization, milling machines became more powerful and precise, and the first mass-production machines came into use around 1900.



Control systems capable of automatically executing complete programs appeared around the middle of the 20th Century. This meant that machine operators no longer had to painstakingly set up each step by hand, so manufacturing times and sources of production errors decreased rapidly as a result. The advent of modern computers and improved machine technology now means that most people can perform simple

machine-based milling without having to go through specialist training first.

Milling technology has not only become easier to operate, it has also become a lot cheaper too, making it an affordable option for hobbyists and model makers everywhere. It is now possible to home-build components that used to be either expensive or simply impossible to source. Improved technology also makes it possible to create parts with tolerances of just a few hundredths of a millimeter.

So what's to stop you going out and buying a machine right now? Although CNC machining is no longer taboo for beginners without prior knowledge of the skills involved, if you don't take the trouble to familiarize yourself with the hardware (the machine, its accessories, and the raw materials) and the software, you won't be able to make a single usable part. Furthermore, if you don't learn how to operate your machine properly, you run the risk of serious injury or illness. However, if you prepare diligently, you will be able to manufacture your own parts within the space of just a few days.

This book is designed to help you get started and addresses in detail all aspects of hobby-grade milling. It explains how to turn an easily available kit into a dependable milling machine and goes into detail on how to run it and expand its capabilities. It also demonstrates which tools and attachments to use for which materials and, alongside the hardware, explains how to select and use appropriate software. Hands-on examples illustrate the complete process, from initial concept to finished component.

1.1 What Is Milling?

Milling is a nonabrasive manufacturing process that cuts using a geometrically defined edge. This means that the dimensions and angles of the cut are precisely defined and can be duplicated at will. A grinding stone is an example of a geometrically nondefined tool with a random arrangement of cutting surfaces. These are different for every grinding

stone and cannot be readily duplicated.

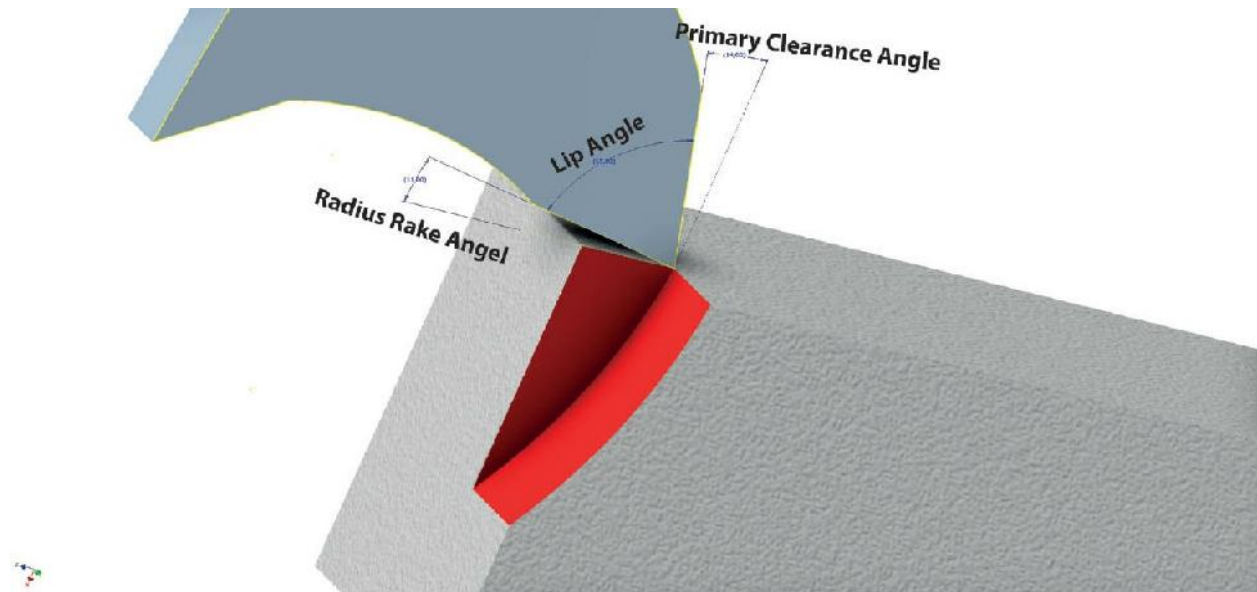


Fig. 1.1 *A geometrically defined cutting edge*

A milling machine (or “router”) is used to mount and guide a rotating tool that has one or more cutting edges, depending on the type of work being done. The workpiece is fixed to the base of the machine (the “machine table”) and the tool is moved until its edge cuts into the workpiece and removes material from its surface. Depending on the type of machine in use, the tool and/or the workpiece can be shifted along and rotated around a variety of axes.

The simplest type of router is a 3-axis machine, which positions the workpiece along the X, Y, and Z spatial axes. Machines exist with up to 15 separate axes, and the more axes available, the more complex the resulting parts can be. A 3-axis machine can only remove material where it has unhindered access to the workpiece—i.e., on the top or side surfaces.



Fig. 1.2 A hobby-grade 3-axis gantry milling machine

Many standard machines can be upgraded to add a fourth axis that gives you access to all sides and the bottom of the workpiece. [Figure 1.3](#) shows an industrial milling machine with a rotating machine table that serves as the fourth axis. Rotating the workpiece enables the operator to remove material from places that are inaccessible using a 3-axis machine. Additional axes also enable the tool to engage with the workpiece at a variety of angles. For example, a combination of rotation and vertical movement can be used to mill threaded parts.



Fig. 1.3 An industrial CNC machine with a rotary machine table (the 4th axis)

The more robust the material of the workpiece, the stronger the machine has to be and the smaller the amounts of material that can be removed during a cut. To remove material, the machine applies force to the workpiece, and an equivalent opposing force is created within the machine. This opposing force can potentially bend or twist the machine itself, which in turn moves the workpiece away from its intended position, causing deviations in the dimensions of the finished part. If these deviations (usually called “deflections”) are too large, the finished part may be unusable.

Engineering machining tolerances are often as little as ± 0.01 mm, sometimes even less. This means that the machine itself has to be stiff enough to prevent flex or torsion-based effects that exceed ± 0.01 mm. These tolerances also apply to the tool and the mechanisms used to clamp the workpiece. To ensure that these tolerances are not exceeded, you have to use either an extremely precisely constructed machine or work slowly to avoid producing excessive force. Machines that work fast using a

lot of force use precision-made steel parts and are accordingly heavy and expensive. Most home users don't have the facilities to accommodate such a machine and usually end up using a simpler gantry-style machine. Many hobbyists even build their own.

1.2 Gantry Milling for Hobbyists

Hobby and model makers seldom require tolerances of a hundredth of a millimeter and are not usually subject to time constraints. Generally, deflections of ± 0.1 mm are acceptable and, by applying appropriate measures, can be improved to a level of 2-3 mm per 100 mm.

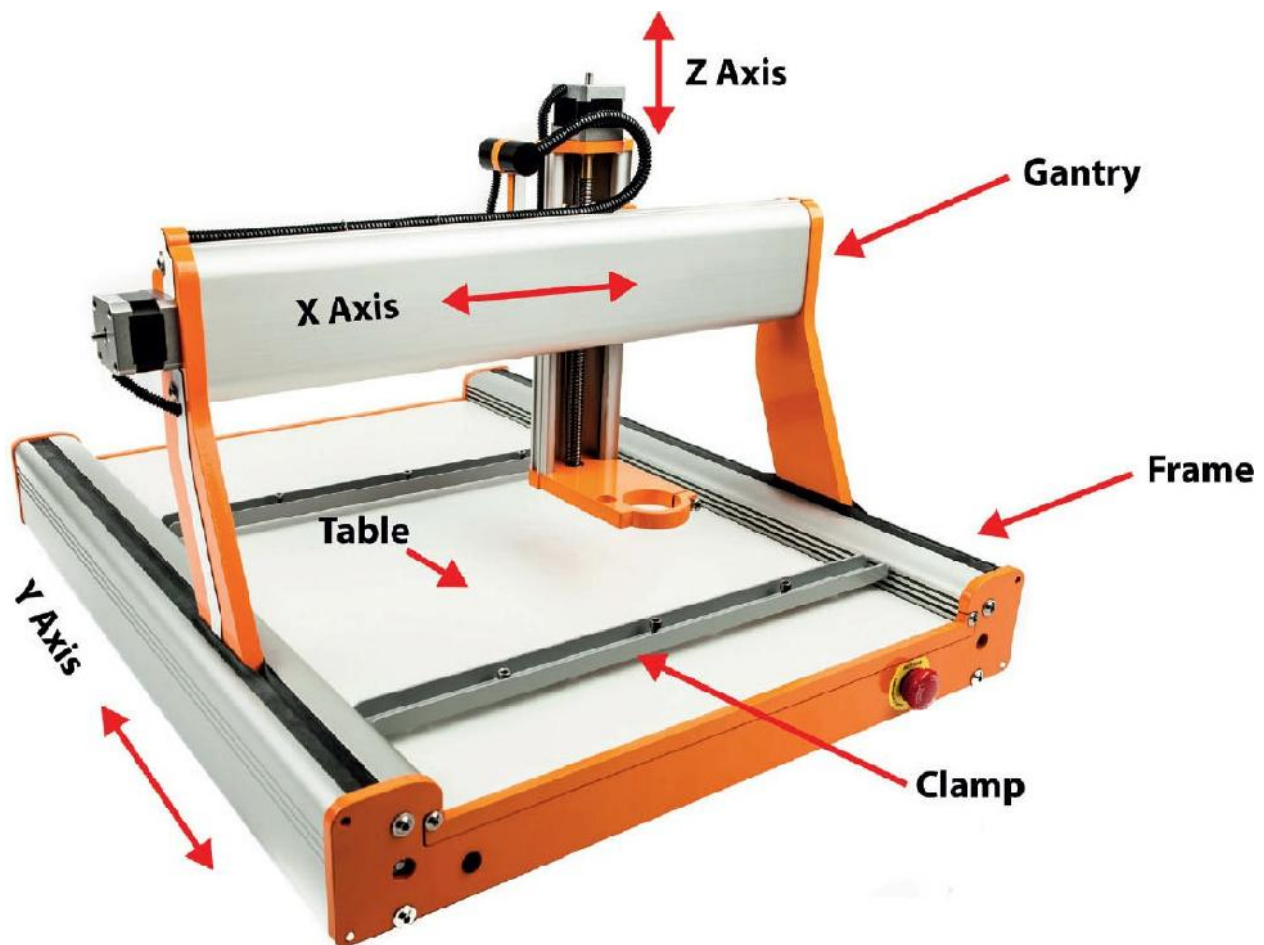


Fig. 1.4 The parts of a gantry router

The machine table in a gantry machine is static and the workpiece is fixed

to it. To keep the tool itself mobile, it is attached to a U-shaped hoop called the “gantry”, which can be moved along the table’s Y axis. A second bracket that moves perpendicular to the gantry is attached to the gantry itself and carries the motor, which can be height adjusted. The frame of a gantry router is often made of extruded aluminum combined with other parts made of solid aluminum, steel, or wood.

The height of the gantry is kept as low as possible because the taller it is, the greater the leverage created by the tool and the less stable the machine becomes. One great advantage of gantry-style machines is that they allow you to work on much larger workpieces than other types of machine. However, the longer a workpiece, the greater the deflections, and the greater the levels of precision and stiffness required of the machine.

Critical forces are produced at the interface between the tool and the workpiece. Applying a tool in the X, Y, and Z directions produces an opposing force in the gantry. The side panels of the gantry produce leverage that exerts force along and across the guides at their lower edges. Applying the tool directly along the Z axis lifts the gantry, thus changing the position of the Z axis itself.

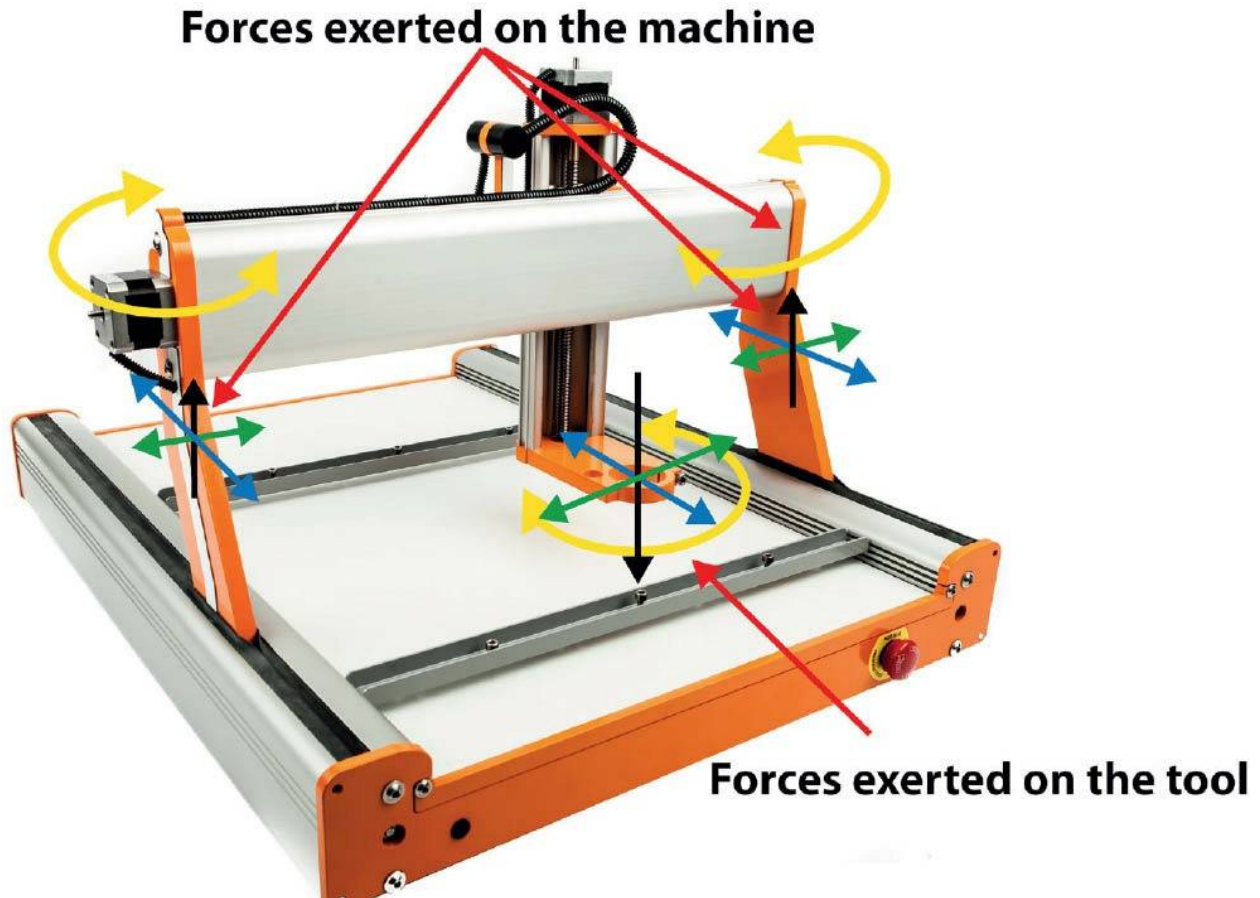


Fig. 1.5 *Forces produced along the X (green), Y (blue), and Z (black) axes. The yellow arrows indicate torsion in the gantry and corresponding changes in the position of the Z axis.*

These forces constantly change direction during a cut and continually move the gantry away from its ideal position. Additional play in the guides or the lead screws exacerbates this effect and once again causes deviations from the planned dimensions. The same applies to the framework of the machine and, if it is not sufficiently stable, it too will flex. Don't think that a 10 mm thick aluminum bracket doesn't bend! The greater the leverage involved, the more likely apparently rigid components are to produce unwanted flex.

If tolerances of 0.01 mm are required, the maximum permissible deflection in any one direction is 0.005 mm. Because a machine is made up of multiple components, deflections of just 0.002 mm in three directions are sufficient to exceed an overall tolerance of 0.05 mm.

Alongside the forces produced by the tool itself, the size of the gantry

and movements in the Z direction can cause issues too. Sudden braking or changes of direction possess sufficient kinetic energy to reduce milling precision. If the machine or its base is not sufficiently stable, the entire construction can begin to vibrate.

1.2.1 The Machine Table

Clamping the workpiece is a critical factor. The workpiece has to be rigidly fixed to the machine table and must be resistant to the forces created by the machining process. The type of clamp depends on the type of table, the raw material, and the size of the workpiece. The workpiece is either fixed directly to the machine table or clamped using a device such as a machine vise.

A machine vise can only be used if your machine is capable of fixing the workpiece with extreme lateral pressure but without flexing. Large or flexible workpieces are often fixed to the table using hold-down clamps. These are available in a large variety of shapes and sizes and are relatively easy to build at home too.



Fig. 1.6 A hold-down clamp

Slotted aluminum plates are a great choice and square nuts, sliding nuts, vises, and claw clamps are just a few of the fixing options you can use with most T-shaped grooves. You will usually need a so-called spoilboard to prevent damage to your machine table when milling holes in a workpiece. A spoilboard is made of cheap, stable but expendable material that is fixed in place beneath the workpiece. Typical materials are:

- Forex (rigid foam)
- Depron (extruded foam)
- Cork
- Rubber
- MDF

Spoilboards can be used multiple times, even if they are already damaged.

The important thing is that the board is sufficiently intact for clamping and supporting the workpiece horizontally. A thick but damaged spoilboard can be milled flat and used again.

A vacuum table is an elegant but expensive alternative clamping tool that consists of a perforated table top and a vacuum pump that sucks the workpiece into place. All you have to do to fix a workpiece is place it on the table and start the vacuum pump. Of course, this technique only works with workpieces that have a flat underside and cannot be used with a spoilboard. You can use a special rubber mat to drill through a workpiece on a vacuum table but, as soon as you drill a hole that is wider than the vacuum channel in the table, the workpiece will no longer be effectively held in place. Because there is no physical clamping involved, vacuum tables are great for working on sensitive raw materials.

The simplest way to fix a workpiece is to use a thick wooden spoilboard and simply screw the workpiece to it or stick it down using double-sided tape.

1.2.2 The Milling Motor

The gantry transports the milling tool to the exact location where machining takes place, while the tool itself is driven by an electric motor. Milling motors are available in a variety of different types, including compressed air-driven models. However, these are seldom found in hobby or home use situations.

A milling motor is often referred to as a spindle, spindle motor, or HF (High Frequency) spindle. The spindle consists of a precisely machined drive shaft and some kind of tool chuck, while its name distinguishes it from a conventional electric motor. Spindles are designed to be compact but to provide very high rotation speeds of up to 100,000 rpm. Hobby-grade spindles provide speeds of up to around 35,000 rpm.



Fig. 1.7 A Kress 800 FME spindle

Compact milling motors that look much like a small electric drill are available from Proxxon, Dremel, Kress, Bosch, Dewalt, and various other manufacturers. Hobby-grade spindles cost up to around US\$250 and are not designed for round-the-clock use. They also tend to wear out faster than other types of spindle, probably due to not being calibrated for high feed rates or cutting speeds. Even if your machine is specified to provide a certain feed, it doesn't necessarily mean that your spindle is up to the job.



Fig. 1.8 A high frequency (HF) spindle

Alongside radial and axial forces, cooling system and open machine designs are factors that adversely affect milling motors too. If too much dust and dirt find their way into the motor (and especially the bearings), it will wear out much faster. To prevent this from happening, milling motors are often built with sealed enclosures and are water cooled. Some open machine designs use airlock systems to keep the motor clean. These use a compressor to produce pressure that constantly blows dust and dirt out through openings in the motor enclosure. This type of motor is much more expensive than conventional models. The most important features of a milling motor/spindle are:

- **Radial Runout:** This is defined in millimeters (for example, < 0.01 mm) and indicates the distance by which the position of the center of the spindle can deviate from the true center of the drive shaft.
- **Input:** This defines the nominal rated input rather than the actual power provided to the tool. Values start at around 300 W for basic

models and can be as much as several kW in high-power machines.

- **Rpm Range:** These are the minimum and maximum rpm (revolutions per minute) values. These values determine the types of tool you can use with the motor in question. The given values are often nominal or “no-load” values, which can turn out much lower in real-world, on-load situations.
- **Weight:** The weight of the motor must comply with the specifications of the machine and mustn't exceed the load-bearing capacity of the gantry.

“More is more” is a concept that only partially applies to milling motors. For instance, high spindle speeds don't always mean high work speeds. The spindle speed is determined by the cutting rate of the tool in use, which is given in units of distance /time (i.e., mm/s or mm/min). The milling speed is determined by the type of material and the shape of the tool. Sticking to the values specified by the manufacturer will increase the lifespan of your tools.

Rotation speeds that are too low can damage milling tools too. The spindle speed has to match the feed rate to ensure that the cutter removes precisely the specified size of chip. If the spindle speed is too high, the chips will be too thick, whereas if it is too low, they will end up too thin. Chips that are too thick put unnecessary strain on the tool, while ones that are too thin only utilize the tip of the cutter. Both scenarios increase tool wear.

The motor speed has to match the tool you are using. For large tools, the motor has to produce appropriately low speeds combined with sufficiently high torque, while smaller tools require higher speeds. Motors that possess a range of torque settings covering the entire range of speeds are usually quite expensive and hobby motors usually represent a technical compromise, so it often makes sense to use different motors with different tools. When selecting a motor, it is important to know in advance which tools and spindle speeds you will be using.

1.2.3 Feed Rate

A cutting tool is moved along the X, Y, and Z axes using threaded lead screws, which work like a nut threaded onto a bolt. If the lead screw is fixed in a stationary position so that the nut doesn't move, rotating the lead screw converts the nut's rotational movement into a linear one.

Lead screws come in a variety of shapes, including, flat, round, trapezoidal and tapered, and are manufactured to a variety of tolerances. High-quality lead screws have threads machined to tolerances of just a few micrometers and you can use the thread pitch to calculate the distance the tool will cover per rotation. The steeper the pitch, the more precisely you will be able to position the tool. Different thread types are used for different jobs. Transmission threads have to move freely and with as little friction as possible, while fastening threads need to produce plenty of friction in order to remain tight.

However well-made they are, there are limits to the degree of precision a lead screw can have. Because hobby-grade gantry routers use the thread pitch to calculate the position of the tool, the tolerances used to manufacture the lead screw will be directly transferred to the workpiece. In contrast, pro-grade machines use custom measuring devices to calculate the actual distance covered by the tool.

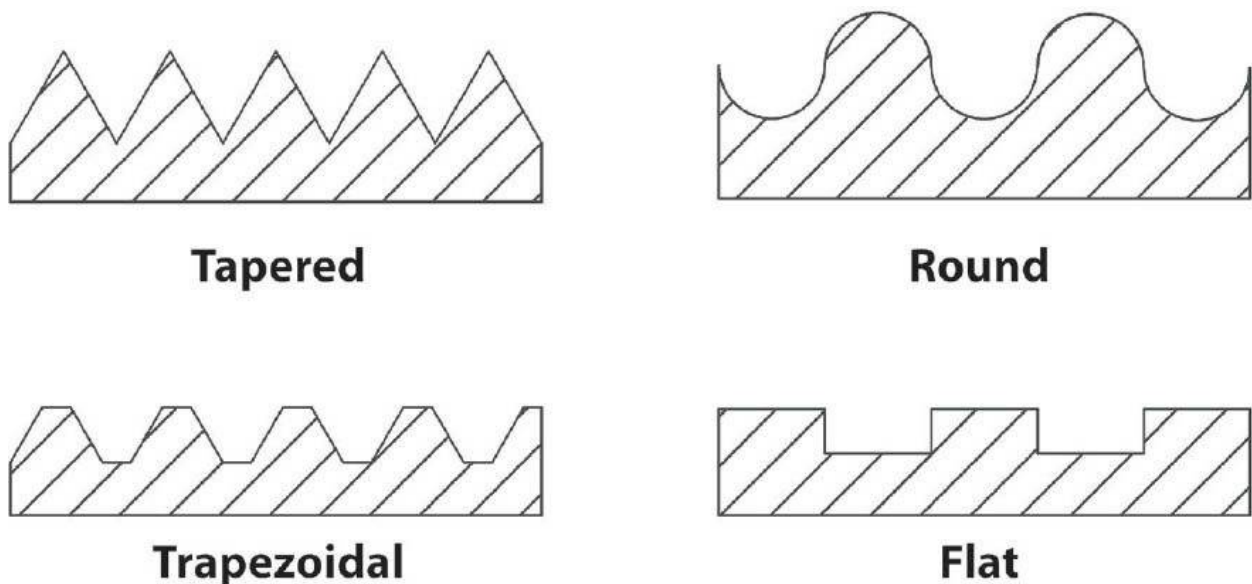


Fig. 1.9 *Types of lead screw thread*

When a bolt or lead screw is rotated within a nut, one side of the screw's thread exerts pressure on one side of the thread within the nut, and it is this pressure that produces feed (i.e., movement) in the mobile part of the machine. At any one moment, the other side of the screw's thread doesn't actually touch the nut. Depending on the screw's quality and diameter, there will be several hundredths of a millimeter of play in the thread. The total amount of play results from the sum of radial and axial play (see [figure 1.10](#)).

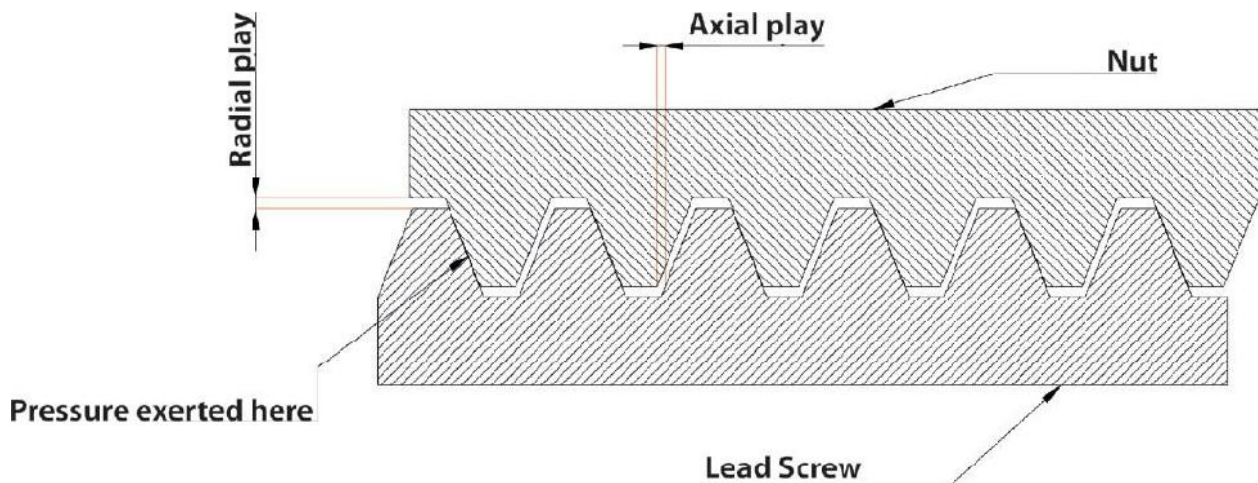


Fig. 1.10 Play in lead screws

If a tool is moved from position X to position Y and back, it will halt before position X at a distance determined by the amount of play in the system. This is because opposite sides of the lead screw's thread are used to produce the forward and backward movements. The offset this creates is called backlash. The machine's control system can use the number of lead screw rotations to calculate the tool's current position but cannot identify backlash. However, because backlash is relatively easy to measure and is usually constant, it can be compensated for in the software settings. Such a setting alters the position of the cutter when the direction of feed is changed.

Some lead screws utilize ball screw mechanisms instead of simple threads. A ball screw uses balls like those used in wheel bearings to exert force on the thread flank and can be set up to completely eliminate backlash. The tiny surface area of the balls produces much less friction

and heat than a conventional thread, making it possible to mill faster and more precisely using less power. However, all the other components of the machine have to be set up appropriately for the ball screw to work effectively. Most hobbyists can produce acceptable results without having to resort to using ball screws.



Fig. 1.11 Roller guides

While lead screws are used to move the tool, the moving parts of the machine (such as the gantry) have to be guided too, often using roller guides like those shown in [figure 1.11](#). Roller guides use two parallel rails and multiple rollers to support the mobile machine component. This type of mechanism is simple to build and relies only on the rails remaining parallel. To ensure that there is no play in the system, the pressure each roller exerts on the rail can be adjusted individually.

This type of linear guide is quite precise but, due to the use of aluminum rails, is not suited to applications that produce high levels of leverage. Machines designed for use in high-load situations usually use flex-free linear guides made of milled steel rather than simple roller guides.

Most gantry machines use a stepper motor to produce feed. Like a conventional motor, a stepper motor uses electromagnetic fields to

produce rotation, and reversing the polarity of the magnets reverses the direction of rotation. A stepper motor can be halted at precisely defined positions within a single revolution. For example, a 1.8° stepper motor can be halted at 200 individual positions within a single revolution. The machine's software knows the stepper motor's specifications, so the thread pitch determines the distance covered by the tool in a single step.

Too much opposing force causes a stepper motor to lose a step—in other words, its electromagnetic field isn't sufficiently powerful to rotate it to the next position. If the control system doesn't detect the lost step, it will assume an incorrect position for the tool. If a 1.8° stepper motor used with a 10 mm thread loses a step, all the following dimensions in the workpiece will be skewed by 0.05 mm. If a total of 10 steps are lost along a single axis, the dimensions of the axis will deviate by 0.5 mm from their intended values. However, you can prevent this kind of error if you know how to set up your machine correctly.

As soon as you begin to add solid metal guides, ball screws, and heavy-grade custom components, a router can quickly end up costing a healthy five-figure sum. However, most hobbyists don't require these kinds of specs for home use and a much cheaper machine will usually suffice. The downside of a cheaper machine is that it cannot effectively mill materials such as stainless steel or titanium, and that most milling processes take much longer to complete.

1.2.4 All About Milling Tools

Different raw materials and different types of jobs require the use of different milling tools, which in turn require the use of varying cutting speeds and feed rates. Tools are available in a variety of shapes and sizes and with cutting edges that differ in number, position, and shape. The shape of the cutting edge is crucial to the results, and the three angles detailed in [figure 1.1](#) determine the size of the chips produced and the ease with which they can be removed. Other subsidiary angles play a less important role.

The lip angle determines the stability of the cutter. Kitchen knives

work on the same principle and the greater the lip angle, the more stable but blunter the blade will be. Steep lip angles are used to mill hard materials and shallower lip angles for softer materials. The lip angle has to be selected in conjunction with appropriate rake and primary clearance angles.

The primary clearance angle ensures that only the part of the tool that removes material makes contact with the workpiece, while the rake angle ensures that the chips produced are effectively removed. The cutter penetrates the surface of the workpiece like a carpenter's plane and removes a chip of material. The shallower the angle of rake, the less curved and the longer the chip will be. Long chips tend to get tangled up in rotating machine parts and are more difficult to remove from the surface of the workpiece. Many cutters are therefore manufactured with a chip breaker (i.e., a steeper angle, a dip, or a curve) to the rear of the cutting edge.

Milling tools are addressed in detail in [chapter 3](#).

Types of Milling

Face and peripheral milling are the two basic machine milling techniques. Face milling is the milling of a surface that is perpendicular to the cutter axis and removes material radially at the end of the cutter's center axis. During peripheral milling, the cutter's axis lies parallel to the surface of the workpiece. Only the orientation of the cutter to the workpiece is important and the orientation of the axes within the workspace is immaterial.

During face milling only a small proportion of the tool's secondary cutting edges come into contact with the workpiece, resulting in a high-quality milled surface. In contrast, peripheral milling uses virtually all of the tool's main cutting edges and is thus more efficient, which increases the lifespan of the tool. However, the large amounts of material this technique removes increases the amount of force involved, resulting in a lower-quality milled surface.

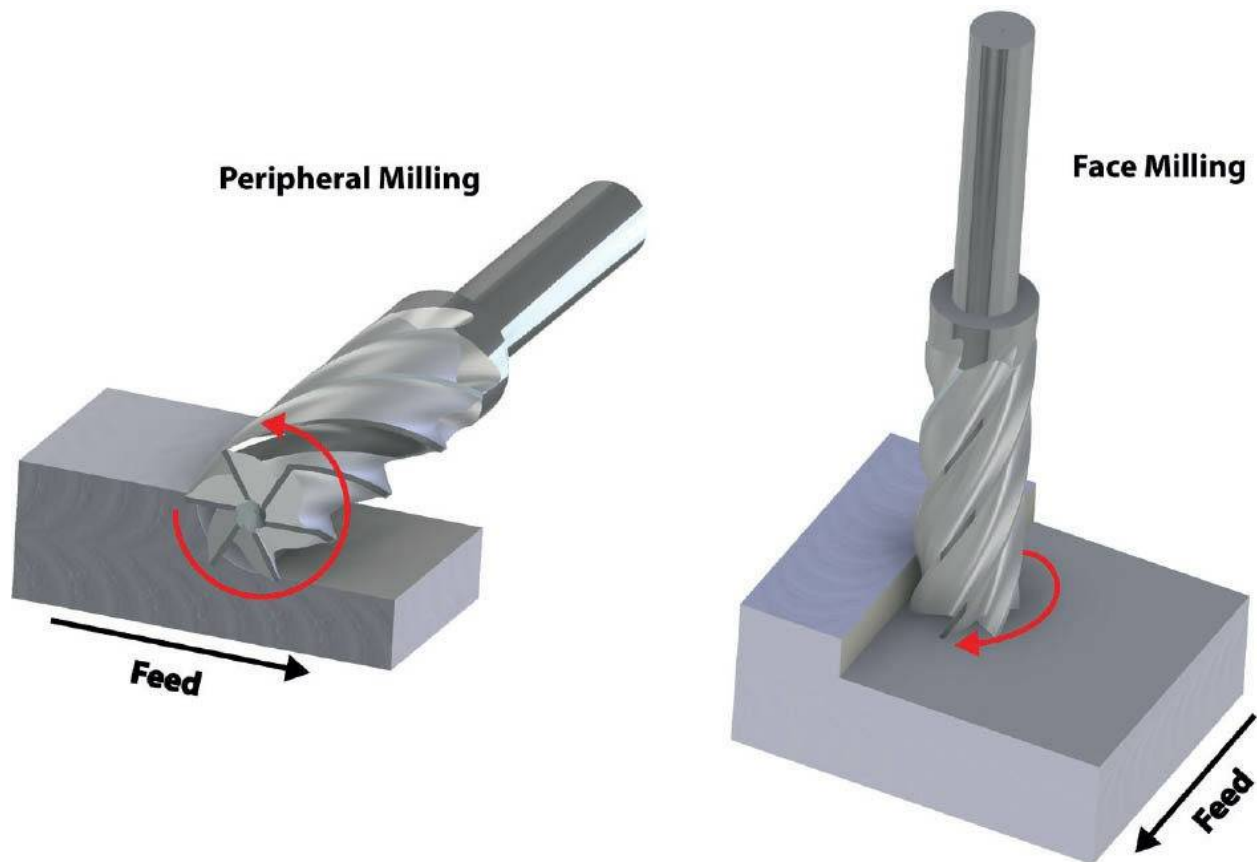


Fig. 1.12 *Peripheral and face milling*

To remove material, the tool is forced into the surface of the workpiece. Because a milling tool can only remove a certain amount of material at one time, the machine has to be told the precise location of the workpiece. The amount of material that can be removed in a single cut depends on the type of material, the type of machine, and the tool you use. The depth of each individual cut (known as cut depth per pass or stepdown) depends on the dimensions of the tool and may have to be repeated to achieve the desired final depth.

For Reference

Visit <https://www.hoffmann-group.com/US/en/hus/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.

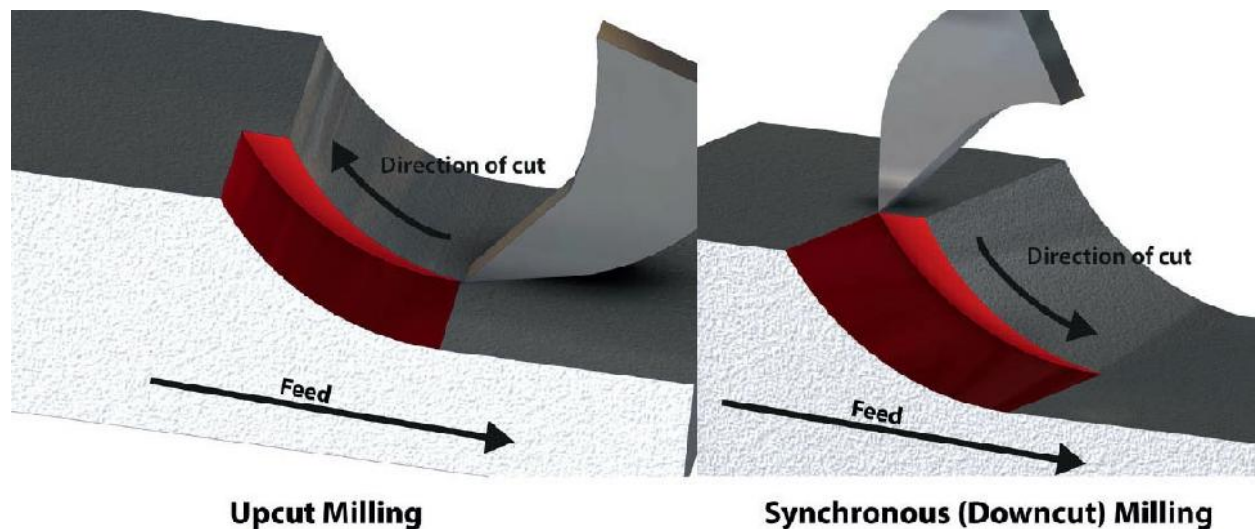


Fig. 1.13 Upcut and synchronous (downcut) milling

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.

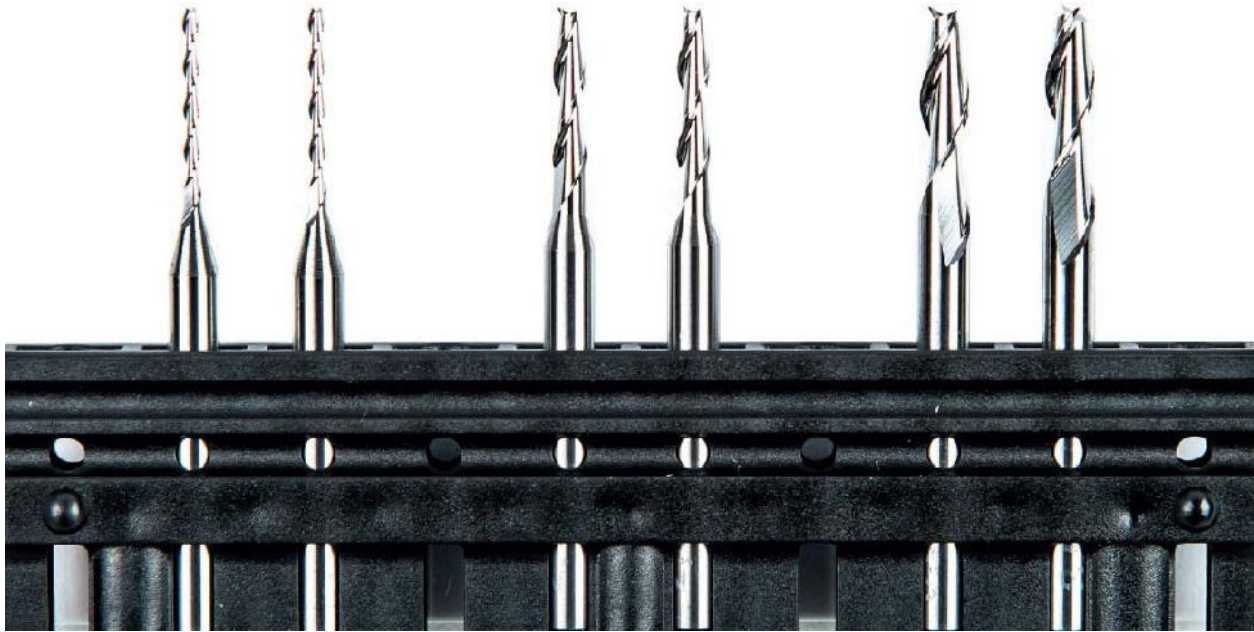


Fig. 1.14 1 mm, 2 mm, and 3 mm solid carbide fishtail bits

Holder Type	Usage
End Mills	
Slot drill	Slots and channels
Slot cutter	Slots
T-slot cutter	T-shaped slots
Plain cutter	Face milling
Angle cutter	Angled cuts
Shell Mills	
Plain cutting head	Face milling
Angle cutter	Guide slots
Profile cutter	Curves and guides
Disc cutter	Slots and separation cuts
End face cutter	Face milling

Table 1.1 An overview of a variety of milling tools according to holder type

Cutters are also available in a variety of designs for various types of job (see [table 1.1](#)). Hobby-grade milling motors usually have only a simple chuck, so hobbyists generally use only end mills. The tool has to fit the chuck perfectly in order to effectively transfer the motor's power to the tool, so tool chucks are available in a variety of sizes too.

Many bits have 1/8-inch (3.175 mm) shanks, while other tools require 3 mm, 4 mm, 6 mm, or 8 mm chucks. It is essential to use the correct sized chuck to ensure that the bit sits tight and is precisely centered. If the bit is loose, it can rotate within the chuck and damage it.

Isolation Milling

CNC machining techniques can be used to create printed circuit boards (PCBs) from copper-coated boards. Building PCBs using etching techniques involves the use of hazardous chemicals that need to be handled appropriately. Instead of etching away all of the copper except the required circuit, milling removes only the copper on both sides of the

circuit, thus isolating it and giving the process its name.

It is not worth acquiring your own machine to create just a few circuit boards per year. Creating a small, single-sided circuit board using the toner transfer method costs about US\$1.50 (excluding the board itself), and you will also need a drill to make the necessary holes. To get started making PCBs, you will need to spend about US\$65. If you need to create circuit boards regularly and don't want to drill all the holes manually, a router is a great alternative option.

Note that the resolution you can attain using milling techniques is lower than that available through etching, which can be used to create traces with a width of 0.08-0.1 mm, whereas milling only allows traces measuring 0.2-0.3 mm. Techniques offering finer resolution are too expensive for general use and require the use of a motor capable of 50,000 rpm or more. If you already own a milling machine or are thinking of acquiring one for other reasons, making simple one- or two-layered circuit boards is a great additional use for it.



Fig. 1.15 *Using isolation milling techniques to make a circuit board*

In contrast to etching, isolation milling damages the carrier layer in the

circuit board and creates tapered gaps between the circuits that are broader at the surface. If you are milling dense circuits, this increases the risk of copper remnants creating unwanted bridges between traces. The indentations the process creates makes milling boards with three or more layers quite tricky. The narrow recesses in in-between layers tend to produce a capillary effect that attracts solder to through contacts and holes, thus creating unwanted vias between circuits.

Cooling and Lubrication

Milling creates friction and friction produces heat. An overheated cutter loses its hardness, rendering it useless. A common way to reduce friction and heat is to use cutting oil.

The simplest way to apply cutting oil is to spray it from an aerosol at regular intervals while you work. Using small quantities of oil this way is known as minimum quantity lubrication (MQL). However, MQL is not always practical and simply applying more oil doesn't really make sense. In contrast, pro-grade machines often have a built-in lubrication system.

If MQL doesn't provide sufficient cooling to prevent damage to the cutter, you can use a cooling lubricant to provide constant cooling and simultaneous lubrication. Cooling lubricants are applied for the entire duration of a cut using a pump system with an adjustable nozzle, and your machine has to have a system for removing excess lubricant once it has done its job.

Fluid coolants are not suitable for use with wood or other absorbent materials, and the coolant of choice in such situations is compressed air. The poor conductivity of wood exacerbates the issue, making it essential to stick to appropriate cutting speeds. A cutter that gets too hot can even set fire to a wooden workpiece.

1.2.5 Features and Add-ons

A basic gantry machine covers a range of applications, but some accessories are essential for certain types of job. If you are on a budget, consider carefully just how large your machine needs to be before making

a purchase.

One way to save money is to adapt a single machine to cover a range of duties. A gantry machine can be adapted for use as a

- 3D printer
- Laser cutter (only with air filtration due to noxious gases)
- Plotter
- Heat cutter
- Drag knife

The heart of every milling machine is the 3D positioning unit. Each type of tool has to be matched up with the machine's positioning system, although a milling bit set up to rotate at a constant speed can be ignored. In the case of 3D printing, the machine's control systems have to constantly coordinate the tool position and feed with ventilation settings and the extruder temperature.

Note that a gantry machine cannot be used to emulate any 3D machine, especially when the tool in use is subject to heavy loads. Multipurpose machines always involve compromises in their general specifications as well as their compatibility with accessories and control software.

Machine Sizes

If you are on a budget and find that the components you make only exceed certain dimensions every once in a while, it is probably not worth purchasing a machine with dimensions to suit the exceptions. You will usually be able to find a workaround, either by milling your part bit by bit or by making it out of several smaller pieces. You can, of course, have large parts made by a specialist milling shop, but it is up to you to decide which approach is more economical.

Smaller machines are generally more stable than larger ones. The shorter your lead screws and the smaller the machine's framework, the

less likely they are to produce unwanted flex. A smaller machine also takes up less space and is easier to transport.

Dust Removal

When you are machining materials that produce hazardous dust, a dust removal or underwater milling system is a must, not an option. Using a home vacuum cleaner is not sufficient and will probably make the situation worse by simply blowing the dust out the back and distributing it all around your workshop. Furthermore, milling produces sparks that mix with oxygen in the air stream and can cause fires inside the vacuum cleaner.

Contrary to popular opinion, the HEPA (High Efficiency Particulate Arrestance) filters often used to filter pollen in home vacuum cleaners are of no use for milling. Pollen particles measure between several hundredths and several tenths of a millimeter, while the dust particles produced by milling processes can be as small as just a few nanometers, which is small enough to pass straight through a HEPA filter.

A purpose-built separator that eliminates fire hazards and filters out coarse waste is a good solution, but a home vacuum is not suitable for use with a milling machine that is in regular use. The answer here is a purpose-built industrial vacuum that is certified for specific types of dust and for continuous operation.

Purpose-built sleeves that surround the tool are an effective way to collect flying dust and chips, and any excess dust can be removed using a brush while you work.

Installation

It is essential to set up your machine in a stable horizontal position. A camping table or a couple of beer crates are definitely not the right base for a mill! A solid wood tabletop doesn't cost much (either from a DIY store or a woodworkers' shop). Your tabletop needs to be mounted on a solid substructure that is not only stable but that also prevents vibrations. An unstable foundation causes inaccurate milling and can also make a

water bath slop over.

T-Slot Machine Table

A T-slot plate is a great alternative to a smooth machine table and hold-down clamps for all shapes and sizes of workpiece. Cheaper models are made of aluminum and are often available as part of a push-fit system. A 1/2-inch solid aluminum plate with a finely finished surface is a good starting point.

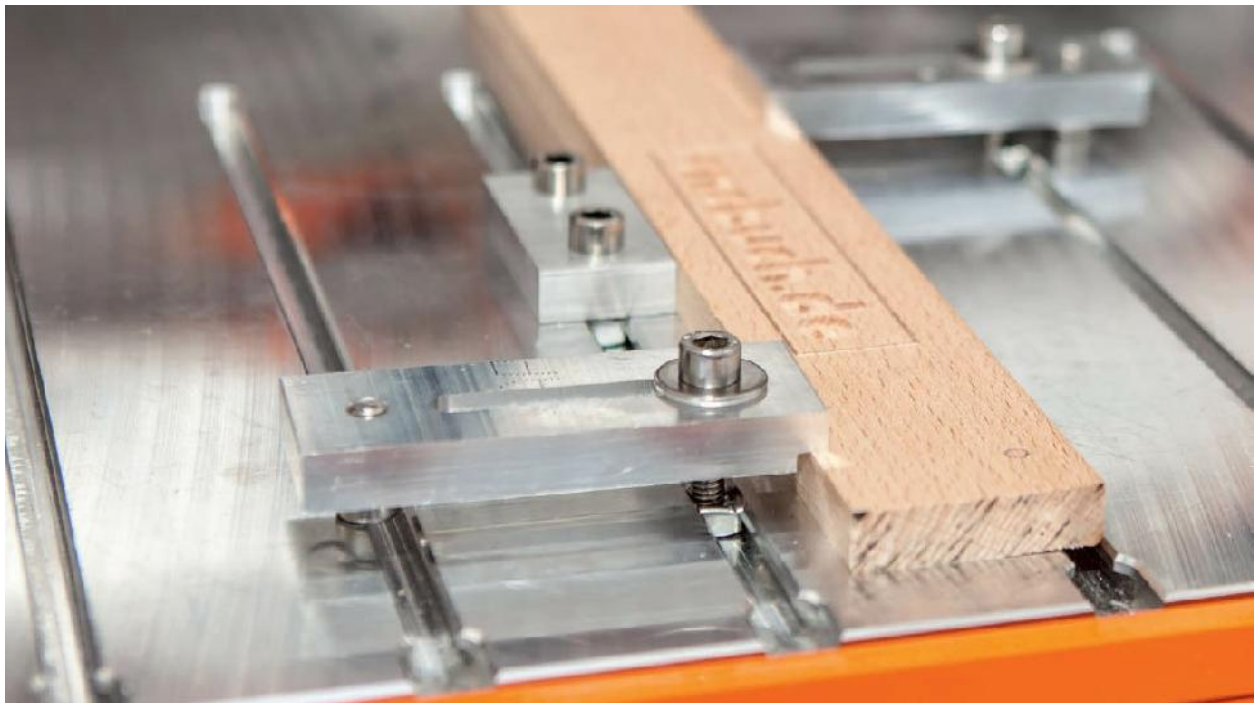


Fig. 1.16 A T-slot machine machine table with a home-made clamp

The effectiveness of a T-slot table depends on its manufacturing tolerances (for example, ± 0.01 mm over 300 mm of travel). Whether these values are relevant or not depends on the degree of precision provided by the machine itself.

Clamping Devices

The workpiece has to be firmly fixed in place whether you use a wood or metal machine table. Various clamping devices are available, depending on the material you are working on. For example, clamping a round

workpiece requires a vise with centering grooves.



Fig. 1.17 A machine vise with centering grooves for holding rounded workpieces

It is relatively simple to make your own clamps using aluminum bars drilled to hold a threaded bolt and a thread with a four-sided nut (see [figure 1.18](#)).



Fig. 1.18 Home-made clamps (left) and a clamping system for holding boards (right)

Tool Length Sensor

A milling machine has to be set up to precisely match the position of the workpiece when it is fixed to the machine table. For jobs with tolerances of just a few hundredths of a millimeter, it is essential to check your setup every time you swap tools or reposition the workpiece. A tool length

sensor is used to quickly measure the exact position of a tool on the Z axis.

The sensor consists of a switch connected to the machine's control system. When a tool is lowered toward the switch, it stops the movement the moment the tool triggers the switch, thus communicating the position of the tool's tip to the machine's software.



Fig. 1.19 A tool length sensor

This provides the software with a zero point on which all further adjustments are based. This point can also be used to adjust the system for different length tools.

The Machine Housing

As well as dust and dirt, a milling machine produces a lot of noise. Cheaper motors especially tend to howl loudly and you will definitely need earmuffs if you use your machine regularly. A purpose-built enclosure enables you to permanently reduce noise levels.

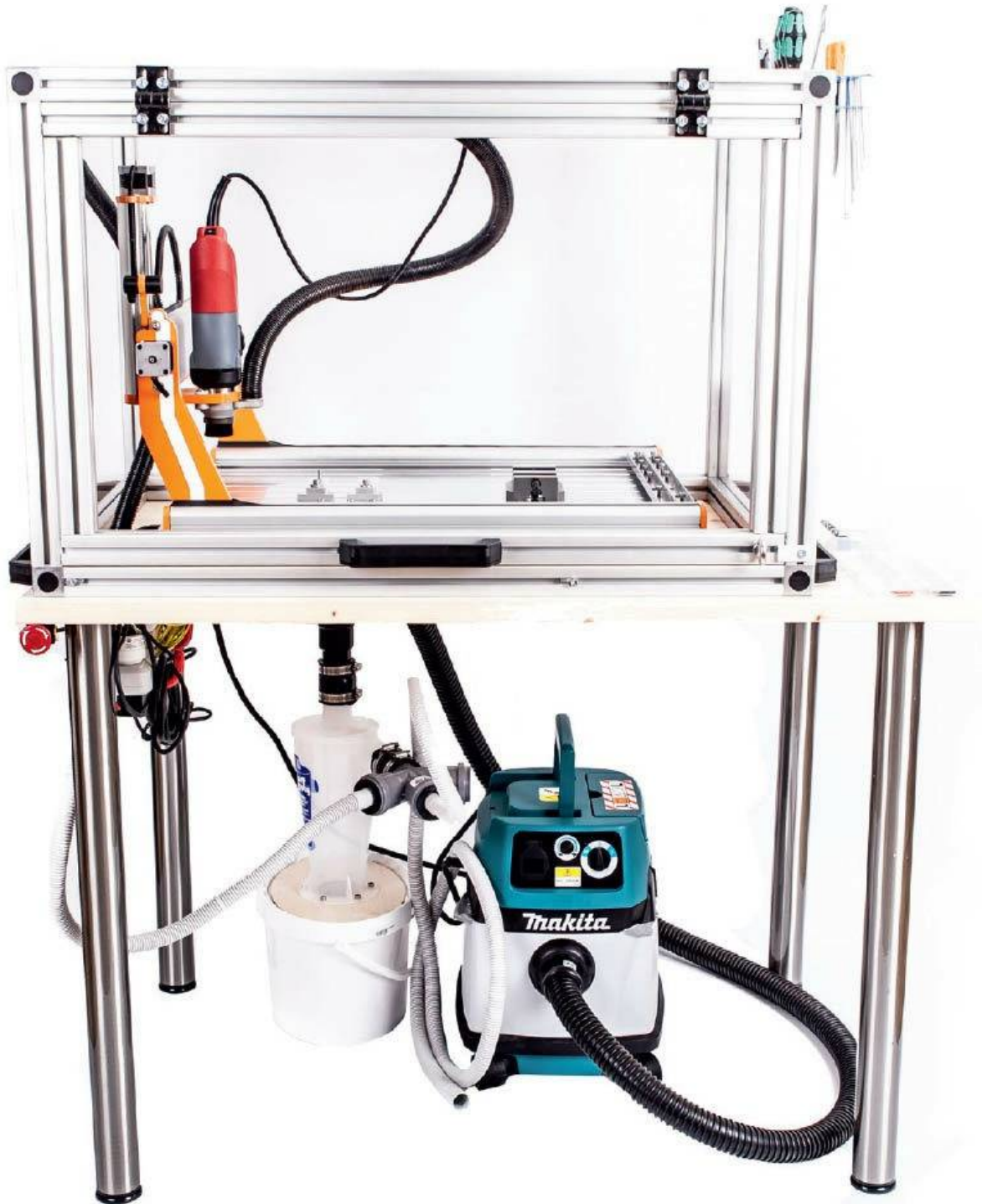


Fig. 1.20 A machine enclosure with an add-on dust removal system

A good enclosure can also prevent you from unintentionally making contact with the machine while it is working and protects you from flying chips and loose tool parts.

Mills/Bits

Be sure to put aside enough money for a good set of cutters. These are often referred to as “bits,” “end mills,” “shell mills,” or simply as “mills.” Milling bits are subject to wear and have a limited lifespan. You will need a selection of different shapes and sizes of bits for most jobs.

Don’t be tempted to buy sets of bits, but instead buy only the ones you really need. Ten or 12 unused bits that lie around gathering dust are simply a waste of money.

Cheap cutting tools and drills are a false economy, and high-quality tools often last 10 or 20 times as long as cheap ones. Don’t be fooled by misleading specifications or terms like “HSS” (high-speed steel), “cobalt steel,” or “titanium.” There seem to be no limits to the claims online sellers can make. A single high-quality cutter can cost anything from a few dollars to more than US\$100. Incredibly cheap prices are usually a sign of poor quality and should be avoided at all costs.

3D Printing

A milling machine with a universal mount can also be used for FDM (Fused Deposition Modeling) 3D printing. Most milling machines are more stable and robust than purpose-built home 3D printers, making it possible to include the material feeder in the printing unit and giving you the option to use flexible filaments.



Fig. 1.21 A milling machine adapted for 3D printing

If you already own a quality milling machine and your printing jobs don't require too large a workspace, it is definitely worth considering the purchase of an accessory 3D printing head.

Rotary Machine Tables and Tailstocks—Adding a Fourth Axis

A 4-axis machine is more complex to use than a simple 3-axis machine.

Most 3-axis machines can be upgraded to include a fourth axis, provided by a unit known as a rotary table.



Fig. 1.22 A 4-axis upgrade with a tailstock (on the left)

Most rotary table upgrades are quite small and offer only limited usage scenarios. The most common uses for a rotary table are the creation of threads and spiral slots or flat surfaces on rounded blanks.

These are specialized jobs, so you need to consider carefully whether you really need a fourth axis for your regular work and whether you are skilled enough to use one properly. A fourth axis also requires the use of more complex control and CAM software, which generates additional costs. You will also need a tailstock to keep the free end of your workpiece steady.

Note that a fourth axis alone doesn't automatically enable 3D milling. It does enable you to reach all the surfaces of your workpiece, but you will still need an additional fifth axis in order to engage with them.

Plotters

A plotter is used to create two-dimensional drawings on paper or any other writable material. A plotter pen is spring-loaded to produce constant pressure on the surface being drawn on. A plotter is also a useful tool for setting up a new machine and for practicing programming and machining with rotating tools.



Fig. 1.23 A spring-loaded plotter pen

Hot Cutters

A hot cutter consists of an electrically heated wire mounted in a U-shaped bracket that is used to cut foam blanks. The wire in a hot cutter is mounted vertically and the machine moves it two-dimensionally in the horizontal plane.

Hot cutters are great for making foam inserts for custom cases. Model makers often use hot cutters to create lightweight parts for ship and airplane models, but are limited to cutting in two dimensions. This makes it virtually impossible to hot cut complex three-dimensional shapes such as a ship's hull or an airplane wing.



Fig. 1.24 A hot cutter mounted on a milling machine

Drag Knives

A drag knife is used to cut films, membranes, and other thin materials in two dimensions. A drag knife's blade rotates freely and automatically follows the direction of drag. A non-rotating blade would only be able to cut in a single direction.



Fig. 1.25 A drag knife with adjustable cutting depth

1.3 Raw Materials

Any material that is hard enough to produce chips when cut is suitable for milling. The shorter the chips a material produces, the easier it is to mill and the simpler it is to remove chips from the work area. If a chip doesn't quickly break off the workpiece, it will create wire that is likely to wrap around the tool or the spindle, thus reducing cutting power. Material that is too soft will simply smear instead of producing chips and will also interfere with the cutter.

The following raw materials are ideal for experimenting with:

- Plywood
- Fiberboard
- Multiplex board
- Laminated wood
- Solid wood
- Acrylic

- Perspex
- PVC
- Styrodur
- Glass
- Carbon fiber board
- Glass fiber board
- Aramid fiber board
- Copper plate
- Dibond
- Aluminum
- Brass

1.4 CNC Milling

A milling machine can be controlled manually or automatically using a control unit and motor-driven feeds. Manual machining uses separate hand wheels and cranks to control feed for each axis.

However, a CNC (*Computerized Numerical Control*) machine can also be controlled manually via its software, so the major difference between traditional and CNC milling is that a computer takes over the basic control functions. In pre-computer times, parts of the milling process were automated using so-called quills to convert the rotational movement of a machine's main drive shaft into variable linear feed along one or more axes.

Numerical Control (NC) was the predecessor of modern CNC processes and was developed during the 1970s, long before modern computers were invented. NC techniques use numerical commands to control simple milling processes.

1.4.1 CAD

Computer control massively reduces preparation and production times and, in conjunction with automatic tool changing (ATC) systems, enables the complete automation of industrial production lines.

The crucial link in the CNC process is the ability to describe to the computer the shape and size of the workpiece and its edges, corners, holes, and threads. The tool required here is a CAD (*Computer-Aided Design*) program that enables the user to precisely model the required design. Although CAD software has come a long way in recent years, it is still quite tricky to reliably model and construct complex workpieces.

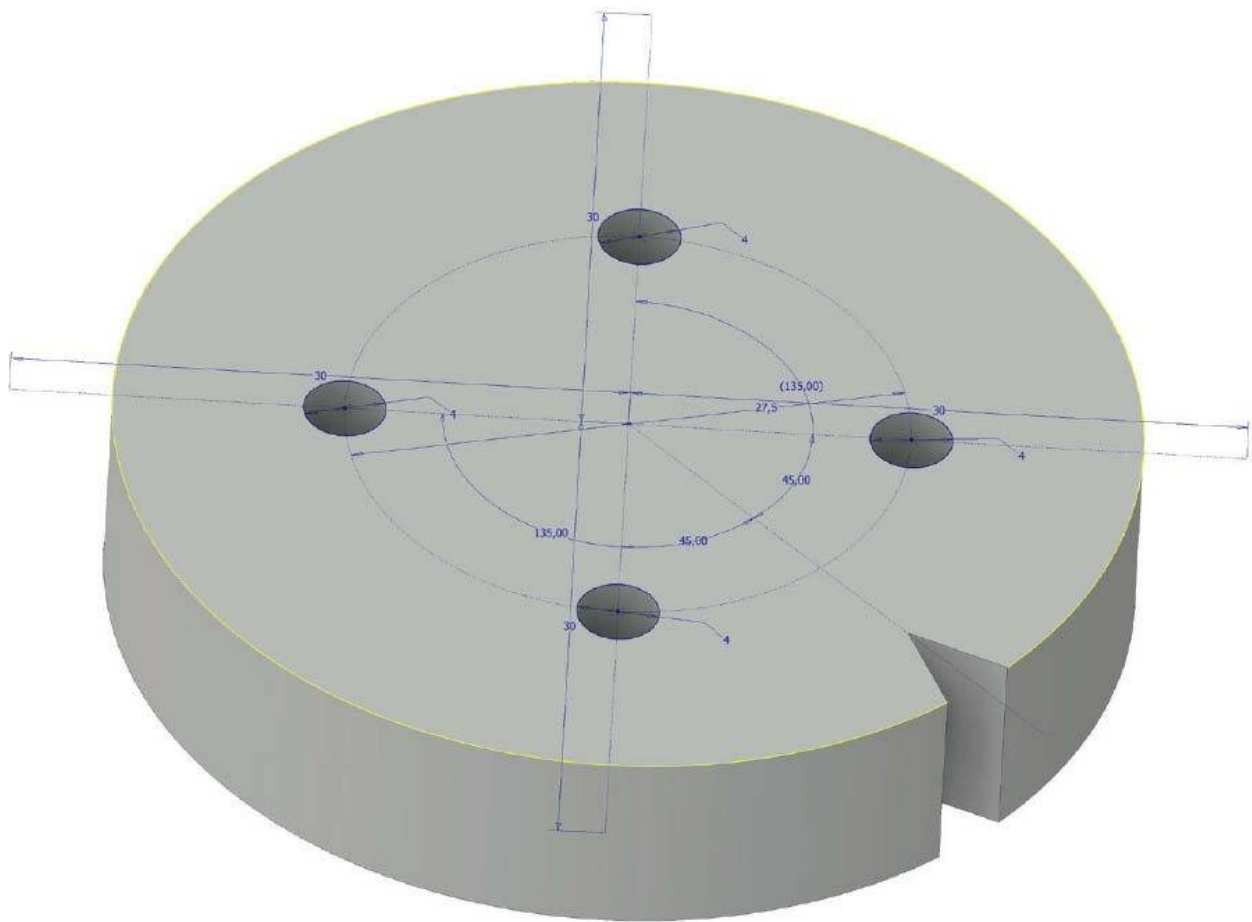


Fig. 1.26 A 3D model of a workpiece made using a CAD program

Computer-generated models are often generated by connecting or intersecting basic geometric objects such as cubes and cylinders. For example, you can model a hole by intersecting an object with a second,

cylindrical object. This type of function is a standard feature of every CAD program, while different CAD applications vary in the number and sophistication of the additional features they offer.

CAD applications include guide functionality to draw lines that are not part of the workpiece itself. These help the designer to position or orientate the various parts of the workpiece. For example, if you want the four holes shown in [figure 1.26](#) to lie within a circle with a diameter of 27.5 mm, you need to define the center of each hole. The blue lines in the diagram are the guides used to do this. It is this type of functionality, along with system integration and built-in industry standards, that make CAD software so complex and expensive.

Alongside traditional 2D and 3D CAD models, milling models for components such as threads can also be constructed using parametric (i.e., mathematical) methods. Recently, digital sculpting techniques developed by the movie and video gaming industries have begun to be used for industrial modeling. Digital sculpting enables free-form modeling on a digital basis. The basic technique uses custom software tools to smooth, flatten, and drag basic shapes into the required form.

Many of today's CAD applications offer all three modeling options, enabling you to combine traditional construction plans and mechanical functionality with artistic design elements. Curves and arches are extremely difficult to produce using conventional design tools, and machining the results nearly always requires 3D milling using at least four axes.

Application	Type	On the Web	Available Versions/Prices
Autodesk 123D design	3D CAD	www.123dapp.com/design	Free or Premium, from US\$9.99/month
Autodesk Fusion 360	3D CAD/CAM	http://www.autodesk.com/products/fusion-360/overview	Free
Autodesk Inventor	3D CAD	www.autodesk.com/products/inventor	Inventor LT: US\$1,750 Inventor: US\$6,900 Subscription: US\$360 Euro/year Students/Educators: Free
Autodesk Mudbox	Sculpting	www.autodesk.com/products/mudbox	Mudbox: US\$850 Subscription: US\$80/year Students/Educators: Free
Autodesk Tinkercad	3D CAD	www.tinkercad.com	Free
Blender	3D CAD & Sculpting	www.blender.org	Free
Cinema 4D	Sculpting Animation	www.maxon.net/en-us	Cinema 4D Studio: US\$4,500
FreeCAD	3D CAD	www.freecadweb.org	Free
ImplicitCAD	Sculpting	www.implicitcad.org	Free
OpenSCAD	Parametric CAD	www.openscad.org	Free
Sculptris	Sculpting	www.pixologic.com/sculptris	Free
Shapsmith	3D CAD	www.shapsmith.net	Free
SketchUp	3D CAD	www.sketchup.com	Make: Free Pro: US\$850
SolidWorks	3D CAD	www.solidworks.com	Standard: US\$5,000 Students: US\$175
ZBrush	Sculpting	www.pixologic.com/zbrush	License: US\$795 Student license: 50% discount

Table 1.2 A selection of well-known CAD programs

3D modeling usually plays only a minor role in hobby machining, with most hobbyists relying on 2D models in conjunction with manual tool and cut-depth choices. However, if the cut depth varies within a single cut or outline, you will need to make a 3D model that includes appropriate data. If a 3D model is constructed so that all required parts can be milled using a 3-axis machine, the resulting process is usually referred to as 2.5D machining.

A CAD model cannot be produced directly by a router, which is a fairly primitive machine that cannot directly interpret 2D or 3D models. Compatibility between the various CAD applications on the market is poor, too, so every CAD model has to be translated into machine-readable form before the actual machining process can begin.

The simplest way to output a 2D model is as a plot—for example, in Hewlett Packard Graphics Language (HPGL) format. However, a plot is two-dimensional so, if your part requires varying cut depths, you need to create a surface model that divides your model up into a series of planes. This eliminates the need for complex three-dimensional descriptions and is a standardized process built in to all common CAD applications.

The most commonly used surface modeling format is STL (Standard Triangulation Language), invented by Charles W. Hull, the founder of 3D Systems. STL is often used in 3D printing processes too. This means you still have the choice between machining or 3D printing your workpiece and it is the subsequent CAM process that determines which route you take.

1.4.2 CAM

Like CAD models, plots and STL files cannot be directly interpreted by a CNC machine. A CNC machine understands only rudimentary commands such as “rotate motor left,” “motor on,” “motor off,” and so on, so an additional step is necessary to convert a CAD model into appropriate machine commands.

A milling tool can only remove material using the tip or the sides of

its cutting edges. Once the specified amount of material has been milled at a certain point, the tool is moved to the next position where milling is due to continue. To produce a finished part, the machine moves the tool over the surface of the workpiece in predefined tracks and removes material layer by layer or progressively from side to side.

The programs used to compute the tracks followed by the tool are collectively called CAM (*Computer-Aided Manufacturing*) software. CAM commands ensure that the tool doesn't collide with the workpiece, that it isn't subjected to loads beyond its specifications, and that it only removes material from the specified places on the workpiece. A CAM program also has to include data that defines the edges of the tool and its required feed parameters.

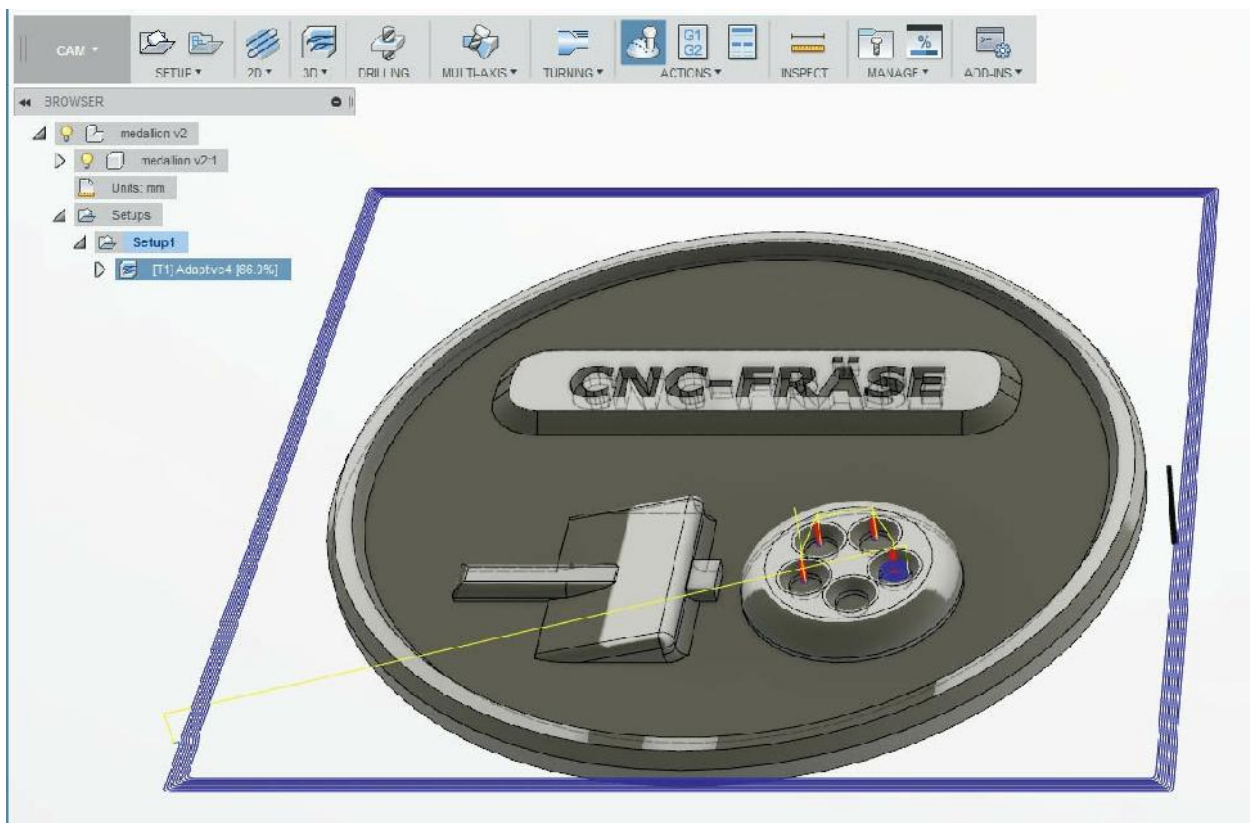


Fig. 1.27 A CAM interface showing the machining parameters for the illustrated workpiece

Like CAD software, CAM applications are available with a wide variety of feature sets and in a broad price range. Simple CAM programs leave it to the user to enter and check the required parameters, while pro-grade

programs enable you to simulate the entire machining process in three dimensions and automatically identify the material and tool(s) being used.

Application	On the Web	Available Versions/Prices
FILOU	www.filou.de	NC16basic: US\$257 NC16expert: US\$907 NC16open: US\$1544
Cut2D	www.vectric.com	Cut2D Desktop: US\$149 Cut2D PRO: US\$449
Cut3D	www.vectric.com	Cut3D: US\$299
Autodesk Inventor HSM	www.autodesk.com	Autodesk Inventor license required HSM Free: Free for 2D and 2.5D HSM and HSM Pro: US\$10,000-\$18,000
Autodesk Fusion 360	http://www.autodesk.com/products/fusion-360/overview	Standard: Free
VCarve	www.vectric.com	VCarve Desktop: \$349 PRO: \$699

Table 1.3 A selection of 2D, 2.5D, and 3D CAM applications

A CAM application produces a machine-level program called G-code that includes all the commands required to control a CNC machine. The illustration on the next page is a G-code excerpt containing the commands required to cut a slot. The code blocks that begin with G01 and end with G03 cause the machine to remove 0.5 mm of material in the appropriate shape. The first G01 command contains the position and speed along the Z axis (both 0.5 mm) and the 24,000 rpm spindle speed. The Z position isn't changed by the following G01 command, which alters the tool's position by 1 mm/second toward the given X and Y positions. The figures in the illustration show some imprecision, although this is limited to less than 1/10 mm.

%

```
G90
T7 M06 S24000
M03
G00 Z-5.0000

G00 X6.5400 Y3.0000
G00 Z-0.5000

G01 Z0.5000 F0.500 S24000
G01 X62.7306 Y3.0072 F1.000
G03 X65.9990 Y6.5966 I-0.2306 J3.4928
G03 X62.6715 Y9.9962 I-3.4997 J-0.0973
G01 X6.3660 Y9.9969
G03 X6.5400 Y3.0000 I0.1335 J-3.4973

G01 Z1.0000 F0.500
G01 X62.7306 Y3.0072 F1.000
G03 X65.9990 Y6.5966 I-0.2306 J3.4928
G03 X62.6715 Y9.9962 I-3.4997 J-0.0973
G01 X6.3660 Y9.9969
G03 X6.5400 Y3.0000 I0.1335 J-3.4973

. . . Further tracks removed

G00 Z-5.0000
G00 X0.0000 Y0.0000

M05
M30
```

G-code sequences are not completely standardized and differ from machine to machine, so you have to use a CAM application that is compatible with your particular machine.

1.4.3 Machine Control Systems

Machine control software directly controls the motors and functions in a CNC machine. The control program interprets the machine program line by line and delivers appropriate commands. This means that the control program itself requires no further data regarding the workpiece. Some control programs also include basic CAD and CAM functionality.

Application	On the Web	Available Versions/Prices
Mach 3	www.machsupport.com	Mach 4: US\$200
UCCNC	www.cncdrive.com	UCCNC with UC100: US\$169
UCCNC	www.stepcraft.us	Custom Stepcraft version UCCNC with UC 100: US\$249
WinPC-NC	www.lewetz.de/en	Light: US\$200 Economy: US\$350 USB with ncUSB-Box: US\$500 Professional with Controller: US\$1,250
WinPC-NC	www.stepcraft.us	Custom Stepcraft versions: WinPC-NC Starter: Free with a new machine WinPC-NC USB: US\$250 when purchased with a new machine

Table 1.4 A selection of CNC machine control applications

Table 1.4 lists a number of widely used machine control applications, some of which are available as cut-price bundles or free with new machines.

Machine control takes place via a normal PC connected to the machine. Many routers use LPT (parallel) ports but, as these are mostly obsolete in the PC world, you will need a parallel-to-USB adapter to make the connection. Timing is key when it comes to controlling a CNC machine, so it is essential that your PC isn't slowed down by other software and that it doesn't accidentally switch into sleep mode during the machining process.

1.5 Alternatives to Milling

The only real alternative to high-precision milling is prototyping, which uses formless raw material. 3D printing is a kind of prototyping.

3D printing processes can be used to produce high-precision, stable parts from metals such as steel and titanium. However, machines for this kind of process are priced in the five- to seven-figure bracket, limiting

their usage to professional and industrial users. Even having a small (several cubic inch) part made by a prototyping service can cost several hundred dollars.

For most hobbyists the alternative is Fused Deposit Modeling (FDM) 3D printing using thermoplastic filaments. The filaments are melted and applied layer by layer using a nozzle, forming a three-dimensional model as they cool. This method enables the construction of complex but lightweight objects. Furthermore, the dimensional tolerances of cooling thermoplastics are not terribly high, and you have to allow for deviations of up to 1/10th of a millimeter.

If you need to make parts out of metal on a budget, the only alternative to milling is using drills, saws, and files. Using these traditional tools, skilled hobbyists can produce workpieces with tolerances as precise as 1/100 mm.

1.6 Personal Safety and Risks

As with all machine tools, using a router involves risk, so you have to learn how to operate your machine properly and observe basic safety procedures in order to work safely. To put you in the picture, the following sections detail the risks of using a hobby-grade mill for model-making.

Warning!

Please note that operating a CNC milling machine involves serious and potentially deadly risks. I am neither a doctor nor a workplace safety expert, so the following sections only go into basic detail on the risks I am personally aware of.

To ensure your own safety and that of others, it is essential that you perform your own due diligence and learn how to use a CNC mill properly, thus reducing the risk of accidents occurring while you work

1.6.1 Motor and Cutters

The most obvious health risk stems from the motor and the tools attached to it. If you need to do anything at all close to the motor, make

sure it is switched off and that it cannot be accidentally switched on. If you touch the switch in error, or someone else plugs in the machine without warning, or if the machine is still connected to the control computer, it can rev up to full power within a couple of seconds and cause serious injury in the process.

If you are using a powerful motor (500W or more) and/or a large diameter cutter, these can easily get tangled up with long hair, clothing, and jewelry. Never wear loose clothing or jewelry (rings or chains) when working with a milling machine. Always tie back long hair or use a hairnet.

If a workpiece isn't properly fastened down, a milling machine can easily rip it out of its clamps and fling it around the workshop. A flying workpiece can be just as dangerous as a bullet or a spinning blade. If you are working on hard or brittle material, a rotating cutting tool will produce chips or splinters that fly around the place and can easily end up in the operator's eyes. Always wear protective goggles or mount your machine in a suitable enclosure.

Cuts are less serious but occur more frequently. New cutters are sharp and can quickly cut to the bone like a scalpel if not handled properly, even when they are not rotating!

1.6.2 Electricity

A milling machine is made largely of metal. To prevent unwanted conduction, many manufacturers use separate sealed power supplies with grounded contacts. If the power supply is built into the machine, make sure that all metal parts are grounded and that all electrics are appropriately safeguarded. When using coolants, lubricants, and underwater milling processes, it is essential to ensure that liquids cannot come into contact with current-carrying components.

Even if your machine has a separate power supply, it will have a 110/230V connection that can easily rub against the moving frame of the machine during use and eventually wear out or get trapped. Here, too, the entire machine could end up conducting electricity and thus become a

deadly health risk.

Be sure to learn how to use your machine correctly and make sure you are aware of all the appropriate safety procedures.

1.6.3 Fire Hazards

CNC machining often produces very high temperatures. Incorrectly used or worn-out cutters especially can get so hot that the chips they produce catch fire.

If you use a vacuum system to remove chips, red hot chips mixing with oxygen-rich air can easily cause a serious fire. Metal chips can create pockets of glowing embers in the collection unit.

If your dust removal system isn't sufficiently powerful, you can end up with a dangerously high dust concentration in the air. If this concentration is high enough and the dust extremely fine, contact with a single spark can cause a dust explosion, which can be strong enough to destroy an entire house.

Freshly machined workpieces can reach temperatures of several hundred degrees, which can cause serious burns.

Make sure you are aware of the risks before you begin work, and always make sure you have an in-date fire extinguisher close by.

1.6.4 Hazardous Materials

The raw material itself is one of the least obvious potential sources of hazards. Dust from composite fibers and certain types of wood can cause cancer or fibrosis. Oak and beech dust are known to be carcinogenic. Note that beech is often present in MDF and other composite wood products.

If you are working with potentially hazardous materials, make sure your dust extraction system is up to the job. If your home vacuum cleaner isn't certified for use with certain classes of dust, it is not sufficient. According to European standards, dust is classed according to maximum concentration of the particles that pass through the extraction system. In

numbers, these are:

- Class L: <1 %
- Class M: <0.1 %
- Class H: <0.005 %

If you are working on carcinogenic material, be sure to use a class H extraction system. Note also that certain types of dust cannot be simply disposed of with domestic refuse. An efficient extraction system is no use if you end up inhaling a cloud of deadly dust when you dispose of your waste.

But don't panic! The guidelines listed here are aimed at people who regularly spend long periods working with machine tools. In many cases, you can prevent the generation of too much dust by using an underwater milling system that binds dust in water and doesn't allow it to get into the air at all.

In addition to your raw materials, lubricants, coolants, and cleaners such as acetone, benzene, and methylated spirit can cause health hazards too. Many coolants and lubricants contain allergenic or poisonous substances that you shouldn't breathe in or get on your skin. Fast-rotating cutters can have an aerosol effect and atomize liquids that can then be unintentionally inhaled. Further information can be found in the OSHA Metalworking Fluids Standards (<https://www.osha.gov/dhs/reports/metalworking/MWFSAC-FinalReportSummary.html>).

Virtually all of the listed substances are hazardous if they come into contact with your skin, eyes, or mucous membranes, and many of them are flammable too.

1.6.5 Safety in the Workplace

The most common cause of accidents is a badly organized workplace. It is surprising how often tripping over a loose cable can cause an entire machine to tip over. Greasy floors are a common hazard too. Supports

that aren't sufficiently stable create a permanent hazard—in other words, a CNC milling machine should under no circumstances whatsoever be used while balanced on a camping table.

Keep your workplace clean and free of all kinds of obstructions. Regularly remove any grease and excess liquids and always clean up any remaining dirt and dust when you finish work. Make sure your workplace lighting is bright and dazzle-free.

Access to Machines

Please ensure that no unauthorized persons—especially children—have access to primed or running machines and that they cannot unintentionally switch them on. Always read your machine's manual and stick to the regulations it stipulates.

2 Your Own CNC Router

Getting set up with your own machine involves either a lot of effort (home-build), a lot of money (purchasing a new machine), or both!



Increasing demands on precision, speed, and the size of workpieces that can be machined makes the choice of hobby-grade machines increasingly narrow. You can always get a commercial shop to machine your parts for you, but this only makes economical sense if one or more of the following conditions are fulfilled:

- You only need to produce a few parts
- Your model is completely finished and you don't need to produce more prototypes to achieve your desired result
- You need to stick to tolerances that your own machine cannot provide
- Your part is made of a material that your own machine cannot process

As well as the machine itself, you need a spindle and a selection of cutters. You will also need a PC, and you have to decide which CAD application you want to use for modeling and which CAM application for converting your models into machine-readable code.

Your answers to the following questions will help you decide which machine you need and to analyze the available options:

- What is your maximum budget?
- How large do the largest workpieces need to be?
- Do you need to make undercuts or threads?
- How many parts do you want to produce per year?
- Which tolerances need to be met?
- Which materials do you want to machine?
- What shapes of material (boards, solids, rounded) do you want to work on?

Your budget is the most important factor of all. Once you have decided how much to spend, don't forget that the sum must include a machine, a spindle, tools, and accessories.

Once you have found a selection of appropriate machines, make sure they are all large enough to cover your needs. Also check whether the accessories you require are available from the manufacturer or from compatible third-party suppliers. Once all these criteria are fulfilled, you need to check whether the machine you are looking at is capable of working with sufficient of precision on your chosen materials.

If you want to work with hard materials such as steel, you need a solid, flex-free machine. This type of machine for home use costs upward of around US\$5,500 without a spindle or other accessories.

You will probably find either no choices at all or several that fit the bill. If you can't find a single machine that matches your specifications, you will have to either increase your budget or rethink your requirements.

If you find that only a few of your planned pieces can't be made using your chosen machine, it can still be cheaper to have these made by someone else and stick to buying a lower-price machine for your own projects. A quick Web search for “CNC machining online” or “CNC milling online” will turn up plenty of commercial milling services.

It costs nothing to ask!

Ask the manufacturer if the machine you are looking at can handle the materials and tools you need, whether it offers appropriate tolerances, and how it performs with regard to wear and repair. The answers and the reception you get will help you decide whether you are comfortable with your choice.

2.1 Choosing the Machine for this Book

The following sections explain why I selected the Stepcraft-2/600 machine as the basis for this book.

The main uses for my machine are building model airplanes and multicopters using wood, aluminum, and carbon fiber laminate. I use these materials to make base plates, motor brackets, transmitter cases, casings, various component holders, and much more. The machine in question should be capable of other things too—specifically, isolation milling.

The plates I use to build transmitter cases and ground stations measure up to 12 × 16 inches (30 × 40 cm). Machining carbon fiber requires the use of a water bath, which uses up at least 2-3 inches (5-6 cm) of the available length and breadth on the machine table. This means the work surface has to measure at least 14 × 18 inches (35 × 45 cm). My workpieces are a maximum of 2 inches (5 cm) tall.

However, because my work involves a fourth axis and machine vises, the overall vertical space requirement is more like 3-4 inches (8-10 cm).

■ **Maximum budget:**

- US\$4,000 including accessories, software, water bath, enclosure, and vacuum system

■ Maximum workpiece size:

- 18 × 15 × 4 inches (45 × 35 × 10 cm), including water bath and fourth axis

■ Undercuts and threads:

- Not initially required but should be available as an option

■ Number of workpieces per year:

- 50-100

■ Required degree of precision:

- At least 0.1 mm

■ Raw materials:

- Wood
- Laminates
- Plastics
- Aluminum
- Brass

■ Raw material types:

- Mainly boards
- Occasionally solids
- Round blanks later on

The first step involves calculating rough prices for the individual components. A few hours' online research resulted in some usable data:

- Vacuum system: An industrial-grade vacuum suitable for my purposes costs around US\$400

- Water bath: A finely finished 20 × 20-inch (50 × 50 cm) 10 mm aluminum plate plus aluminum brackets for the sides and other odds and ends for mounting cost a total of about US\$130
- Enclosure: A complete enclosure measuring about 40 × 32 × 32 inches (100 × 80 × 80 cm) made of aluminum struts, sheets of Perspex, and mounting materials should cost around US\$400. These costs can be reduced by using cheaper Perspex or by building your own enclosure from scratch.
- T-slot machine table: An appropriate sized, high-quality T-slot machine table costs around US\$250
- End mills: 1-3 mm solid carbide mills cost between US\$4 and US\$6 each. A selection of 10-15 mills for each type of raw material should be OK to begin with.
- Machine vise: A small high-precision vise costs around US\$65
- Spindle: A compact model from Dremel, Proxxon, or Kress costs US\$130-250. Pro-grade models start at around US\$650.
- Tool length sensor: This saves a lot of effort and costs around US\$65
- Software: There are a lot of software options, including free open-source CAD/CAM applications, bundled starter versions provided with new machines, and pro-grade software that sells for five-figure sums. At this stage, we are interested in CAM software that translates CAD models into machine-readable code and control software for the machine itself. As we are currently looking at a 3-axis machine, we only need 2.5D software to get started. The cost of the software is analyzed together with the machines we have included on our shortlist.

So far, we have spent around US\$1,600, which leaves around US\$2,200 for the machine itself, which has to be large enough to fulfill all of our other criteria too. You can only calculate the cost of accessories once you

have chosen a basic machine.

Purchasing a kit rather than a ready-built machine can save another US\$400-600. The advantage of a kit over a ready-to-run machine is that you get a feel for the various parts while setting it up, making you better equipped to sort things out if anything breaks. Additionally, you could use the money you save to buy a larger machine or more accessories.

Sections 2.1.1 to 2.1.3 describe the machines that made it onto our shortlist. The prices and details of the available accessories were correct as we went to press.

Kit or Ready-to-Run?

You will need a fair level of technical know-how to put a kit together. If you have two left hands and 10 thumbs, and a friend who says “No problem!” without actually knowing what a milling machine is, you are better off purchasing a ready-to-run model.

2.1.1 Inventables X-Carve

The Inventables X-Carve is a light-duty CNC router that comes in three sizes; 500 mm, 750 mm and 1000 mm. It is sold in kit form, which can take a considerable amount of time to assemble, from eight hours to as much as 20 hours depending on your skill level. The main rails of the X-Carve appear to be made from a standard aluminum extrusion material with metal end brackets that connect each axis. The axes move on small ball-bearing V wheels that use acentric posts for adjustment. Each axis is belt driven, which does not give the precision, accuracy, or rigidity of a lead screw or ball screw-driven machine, but it works well as a hobby machine on woods and plastics. When cutting metal, such as aluminum, the machine must be run slowly. The maximum Z-axis travel height is 65 mm (roughly 2.5 inches). Much of the wiring on the X-Carve is exposed on the outside of the machine, which includes screw terminal blocks that are unprotected from the possibility of something metallic on the bench from accidentally making contact. The system is designed with a spindle mount for the Dewalt DW611 Trim Router, which is commonly found at most large home improvement stores or online. A variable, computer-controlled spindle is not available. The newest version of the X-Carve

does come with an improved power supply and controller, which is still externally connected to the Machine. However, the controller and power supply are now contained in one unit.

The X-Carve comes with an MDF machine table that has a grid of holes with threaded inserts installed from the bottom, which doubles as a spoilboard and clamping system. They also have a measuring grid printed on the table, which will eventually get marked up if not used with an additional spoilboard.

The X-Carve comes with free software called Easel. This is a web-based app that runs from any Windows or Mac computer. The software is designed for beginners, making the basic machine operation machine and setting up jobs simple. However, as your skills progress and your projects get more complicated, you will find that you need to look to other software, such as Fusion 360 from Autodesk or V Carve from Vectric.

The X-Carve product line has been around for a while and there is plenty of information and a range of accessories available at the X-Carve website, where you can also find an active online community.

Inventables X-Carve	
Manufacturer	Inventables
Website	www.inventables.com
Service	Telephone, email and online discussion forum
Usable Size	250mm × 250mm × 65mm (9.84" × 9.84" × 2.56") 500mm × 500mm × 65mm (19.68" × 19.68" × 2.56") 750mm × 750mm × 65mm (29.53" × 29.53" × 2.56")
Maximum Z Height	65mm (2.56")
Maximum Workpiece Size	12.99 × 19.49 × 4.33" (330 × 495 × 110mm)
Repeat Accuracy	~0.075mm to 0.13mm (0.0029" to 0.00051")
Backlash	n/a
Maximum Speed	8,000mm/min
Spindle Holder Size	1.69" (43mm)
Accessories	
Spindle Options (sold separately)	Dewalt DW611 (mount included) Optional mounts for Bosch Colt, Dewalt DW660, and universal
T-Slot Table	n/a
Rotary Table (fourth axis)	n/a
Tool Length Sensor	US\$29
Vacuum System	n/a

Table 2.1 Specifications for the Inventables X-Carve

2.1.2 Stepcraft-2/600

Stepcraft's new Series 2 machines are available in four sizes. The two smaller ones have a maximum workpiece size of just 11.81 inches (30 cm), but the next model up the scale, the 2/600, offers a 15.74 × 23.62-inch (40 × 60 cm) clamping area, which is more than enough for our purposes. For those of you who need even more space, the largest model offers 33.07 × 23.62 inches (84 × 60 cm).

The spindle mount is a standard 1.69-inch (43 mm) model, giving you a wide range of motor options. The axes have roller guides mounted in aluminum rails.

The Stepcraft website offers ticket-based e-mail support and a range of assembly videos to help you get up and running. Operating instructions, an assembly guide, and a UCCNC starter manual are available for download.

The machine itself is robust and fulfills all our technical requirements. According to the manufacturer, a correctly set up and maintained machine can attain repeat accuracy of at least 0.1 mm.

Stepcraft-2/600	
Manufacturer	Stepcraft Inc.
Website	www.stepcraft.us
Support	Support: Telephone, email, web, online tutorials Phone: (203) 556-1856, 09:00 am – 05:00 pm www.stepcraftsupport.com Email: info@stepcraft.us
Usable Area	17 × 26.77" (432 × 680 mm)
Maximum Z Height	6.88" (175 mm)
Maximum Workpiece Size	16.54 × 23.62 × 5.51" (420 × 600 × 140 mm)
Repeat Accuracy	±0.0019" (±0.05 mm)
Backlash	±0.0031" (±0.08 mm)
Programmable Resolution	0.0002" (0.005 mm)
Maximum Speed	3,000 mm/min
Spindle Holder Size	1.69" (43 mm)
Tool Diameters	1.0-8.0 mm
rpm Range	5,000-29,000
PC Interface	Parallel port or USB (US\$25 extra)
Control Software	UCCNC (included)
Price	Kit: US\$1,699 Ready to run: US\$2,148

Accessories	
Spindle Options	Proxxon drill/grinder IBS/E, 5,000-22,000rpm: US\$99 Kress 800 FME, 10,000-29,000rpm: US\$269 STEPCRAFT HF-500, 5,000-20,000rpm: US\$680 Dremel 3000, 10,000-33,000rpm: US\$99 Dremel 4000, 5,000-35,000rpm: US\$140
T-Slot Table	US\$399
Rotary Table with Tailstock and fourth Axis Module	US\$670
Tool Length Sensor	US\$50
Vacuum System	US\$95
USB Full Version (4 axes)	US\$270
Automatic Tool Changer	US\$630

Table 2.2 *Specifications for the Stepcraft-2/600*

2.1.3 Other Models

Aside from the Stepcraft machines, I was unable to find any other machines on the US market that fulfilled all my requirements, although the Inventables X-Carve series definitely comes close. Things look different if we include other non-US manufacturers, and one interesting option is the Kompass H 1000 model from CNC1 (www.cnc1.eu), which is built in the Slovak Republic. Unfortunately, the H 1000 is too expensive for our planned budget, and some of the accessories aren't available directly from the manufacturer. You can, however, reduce the price by buying the individual components separately in the Kompass online store and putting them together yourself.

Machines from Probotix, Tormach, CNC-Shark, Isel, and CNC STEP are way out of budget, and the only other alternatives for our purposes are value machines from China. Many of these offer solid frames, HF spindles, ball screw drives, and linear rails at highly competitive prices. However, according to many of the online forums I visited, the “China experience” also includes the risk of unsafe electrics

and mechanical shortcomings—things no one should really have to deal with. Generally speaking, manufacturers with no US office will not be able to offer satisfactory service and support, especially when it comes to returning or repairing a machine.

2.1.4 Making a Decision

Thanks to my college studies and a friend who has a Chinese-made machine, I have a fair amount of experience programming and operating CNC milling machines. I have been using Autodesk Inventor for 3D modeling for a while now, and I have built an Ultimaker 3D printer from a kit. I also develop my own hardware, build custom multicopters, and have spent more than 20 years as a professional software developer. All in all, I reckon my chances of understanding a milling machine are pretty good, but I am nevertheless reluctant to purchase a machine outside of my home country.

If I were on an unlimited budget and didn't need to consider my readers' needs, I would probably go for a machine with a built-in cooling/lubrication system that can process steel and other similar materials with 1/100 mm accuracy. However, in the real world, I don't have that kind of budget and no one is going to buy a book that explains how to pursue a hobby using a machine that costs as much as a mid-range car. It is not my intention to get you to order a machine from overseas, and I don't really want to deal with complaints from readers who are not satisfied with the service offered by a particular manufacturer.

To make sure we spoke the same language and to ensure that the machine adhered to local laws, I definitely wanted to buy a machine from a local supplier, ideally direct from the manufacturer (or its subsidiary). This also meant I could fall back on a warranty if necessary. The Stepcraft machines are the only ones available in the US that fulfill my specifications but, in order to clarify the decision process, I have compared the Stepcraft with the Inventables X-Carve, as this fulfills at least some of my requirements.

The weakest feature of the X-Carve machine is its low maximum Z

height of just 65 mm. Adding a spoilboard, a water bath, and clamps further reduces the available space and makes using a fourth axis virtually impossible. In theory, you could simply remove the table and raise the machine (or lower the fourth axis and/or water bath), but this would create a two-piece setup that is no longer rigid and would definitely reduce machining accuracy. Furthermore, the brackets at the end of the X-Carve machine's Y axis appear thin enough to flex under excessive load. Put simply, I didn't want to waste good money on a makeshift solution when Stepcraft offers a complete, integrated solution that fulfills all my requirements.

Furthermore, instead of just relying on technical data I decided to take a look around the Stepcraft online forum. I found plenty of entries complaining about one thing and another or asking specific questions, but few that said, "Great—everything worked perfectly straight out of the box." I took time to filter out the time-wasters and came to the conclusion that most of the things that didn't work were due to user errors and not because of shortcomings in the machine. To make doubly sure, I called Stepcraft and arranged an on-site meeting, where I was met by a friendly and professional team. My overall impression was positive, so I decided to order a Stepcraft-2/600. The basic costs added up to about 60 per cent of my original budget, leaving me enough money to acquire additional accessories later on once the machine was up and running.

The costs for the individual components are listed in [table 2.3](#).

Component	Price
Stepcraft-2/600 kit with full-version USB interface	US\$1,880
Kress 800 FME	US\$269
T-slot table	US\$299
Tool length sensor	US\$50
12 VHM 1 mm, 2mm, and 3mm cutters (fishtail and spiral)	US\$50
0.125" and 0.24" (3.175mm and 6 mm) Kress collets	US\$40
Total	US\$2,588

Table 2.3 The cost of purchasing of my Stepcraft-2/600 machine and accessories

2.2 Assembling the Kit

My machine turned up ten days after I ordered it. It was well packed in a box measuring $30.7 \times 21.6 \times 4.72$ inches ($78 \times 55 \times 12$ cm) and was quite heavy. I removed the T-slot table to take the photo shown in [Fig. 2.1](#).

Assembly for all Stepcraft Series 2 Machines

The assembly steps for all Stepcraft Series 2 machines are virtually identical, and any differences are explained in the boxes located throughout the text.

The photos show a 2/600 machine.



Fig. 2.1 The boxed Stepcraft-2/600 kit

As well as the components themselves, the kit also includes the following printed materials:

- Full-color EasyBuild assembly manual
- Operating instructions

- Stepcraft QuickStart Guide
- First steps manual for UCCNC
- A list of the included parts
- User manual for the Kress FME 800 spindle (with test certificate)



Fig. 2.2 *The documentation included with the kit*

To begin, go through the packing list and make sure all the parts are there. Don't immediately open all the bags and boxes, as this makes it more difficult to find the right pieces later on. You will need a table or workbench that is large enough to accommodate the machine and all the parts or, failing that, one table for the machine and another for the parts. The tools shown in [figure 2.3](#) are the minimum requirements for building the kit. The grease is included and all the other things are easily available at a home improvement store. For additional help, check out the Stepcraft online assembly course at www.stepcraftsupport.com/assemblycourse

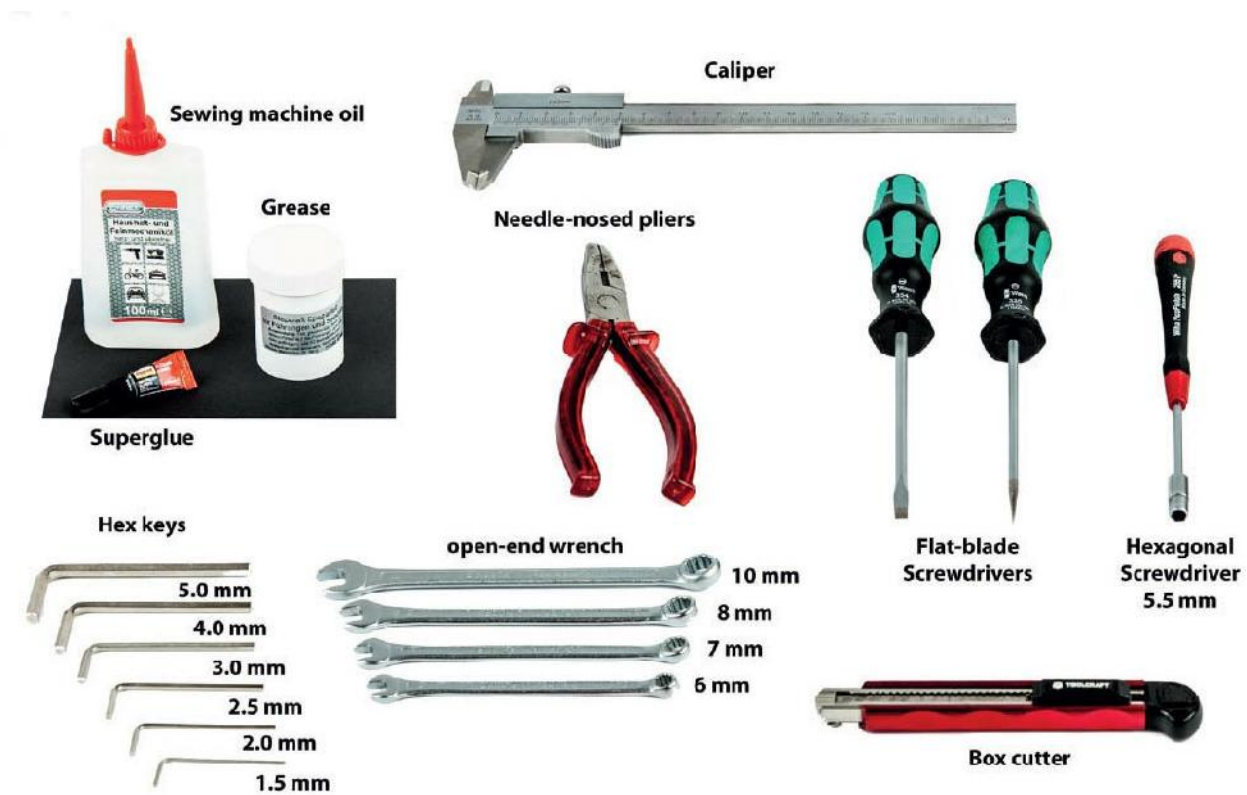


Fig. 2.3 The tools and materials required to build the kit

Once you have prepared the tools and an appropriate space, you can start assembling your kit.

Tools

To prevent damaging the screws and bolts, always use high-quality tools.

Assembling the kit involves first putting together the individual modules and then assembling them to create the finished machine. The entire process takes about five or six hours if you work methodically and allow yourself an occasional coffee break while you check your progress. The approximate duration of each step is given in brackets.

- Step 1: Assemble the X-Z connector (1 hour)
- Step 2: Assemble the Z axis (30 minutes)
- Step 3: Assemble the X lead screw and gantry sides (1 hour)
- Step 4: Assemble the gantry (30 minutes)

- Step 5: Assemble the machine table (1 hour)
- Step 6: Basic machine assembly (15 minutes)
- Step 7: Add the electronics (30 minutes)
- Step 8: Machine table and final assembly (5 minutes)
- Step 9: Checking and starting the machine (30 minutes)

2.3 Step 1: Assembling the X-Z Connector

This part connects the guides that position the spindle in the X and Z directions. The guides are held in the rails by four rollers and are moved using lead screws. [Figure 2.5](#) shows the required parts, which are also illustrated and labeled in the printed assembly guide. Unpack them and place them on your parts table.

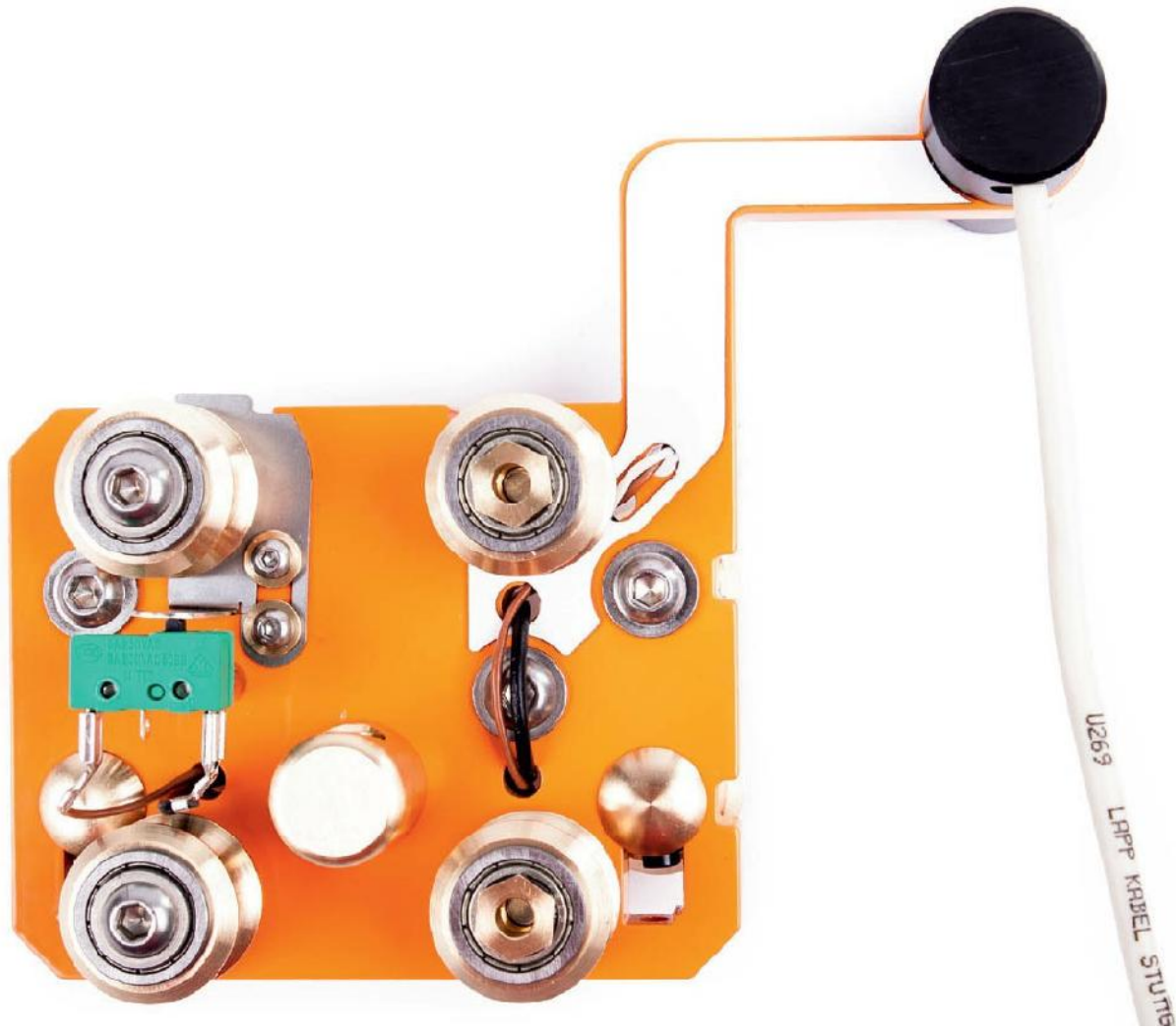


Fig. 2.4 *The assembled X-Z connector*

Rollers

The rollers are packed individually and, to improve the overall precision of the finished machine, are fitted with their own bushes and bearings. Remove only a single roller from the bag as you need it and make sure that its individual parts remain together.

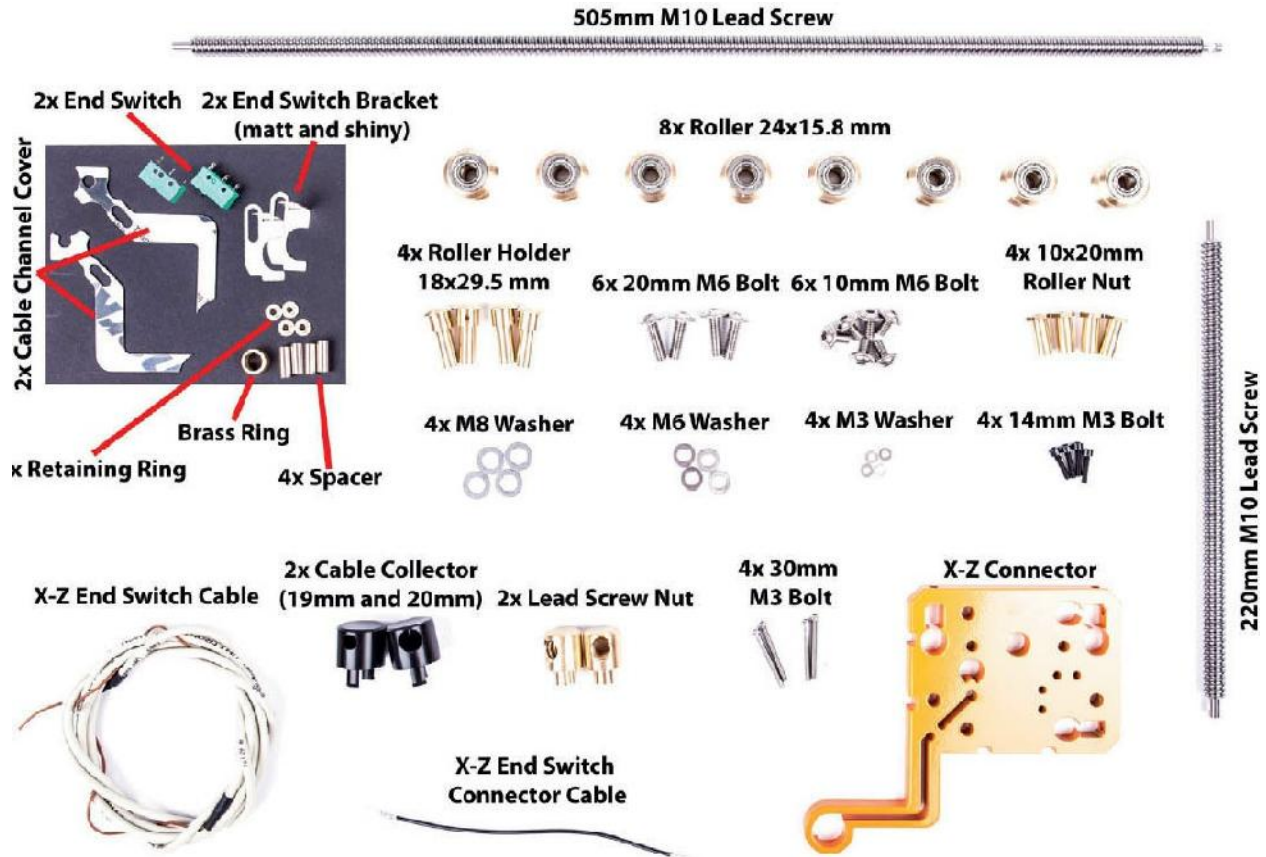


Fig. 2.5 The X-Z connector parts

1. Fix the 20 mm cable collector to the large hole in the X-Z connector and glue the cable channel cover to it as shown in [figure 2.6](#). Make sure all holes in the X-Z connector remain uncovered. The following steps use the cable collector to identify the position and orientation of the connector.

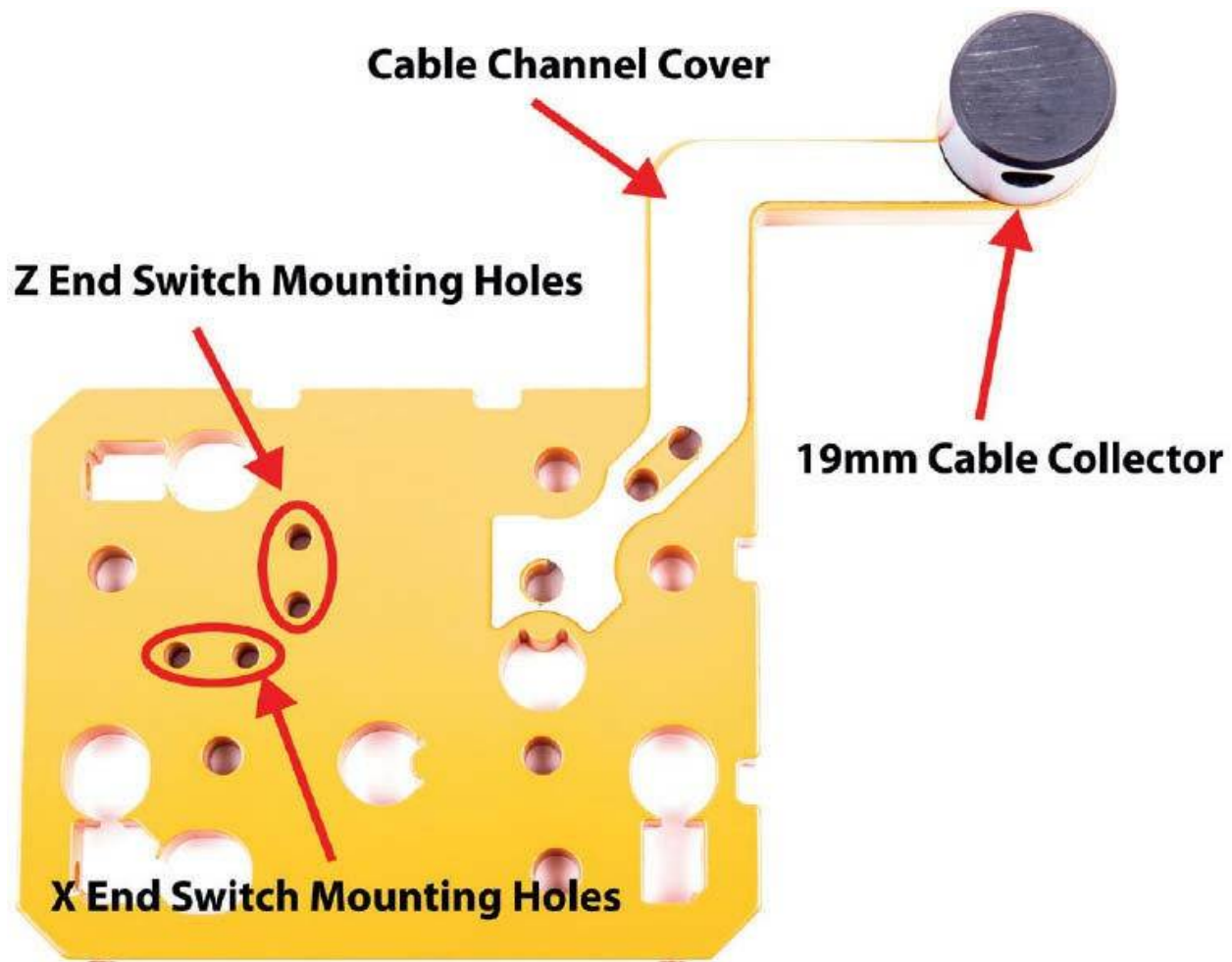


Fig. 2.6 The X-Z connector

2. Take two 30 mm M3 bolts and add a retaining ring to each, making sure that the smaller part of the retaining ring faces away from the bolt's head. This serves as a guide for the end switch bracket.
3. Thread the bolts through the shiny end switch bracket and then the X-Z connector as shown in [figure 2.7](#). The correct holes are shown in [figure 2.6](#).
4. Add a spacer to the other end of each bolt.
5. Attach an end switch to the bolts as shown in [figure 2.7](#) and tighten the bolts. Make sure the black pins on the switch are

positioned correctly and that the retaining rings grip the end switch bracket. The bracket has to move freely once you are done. The plastic switch housing is relatively brittle, so tightening the bolts too much can strip the threads if you are not careful.

6. Add a drop of sewing machine oil to the two places marked in [figure 2.7](#) and move the end switch bracket back and forth a couple of times to spread the oil. Make sure that no oil gets on the other parts and clean up any excess if necessary.

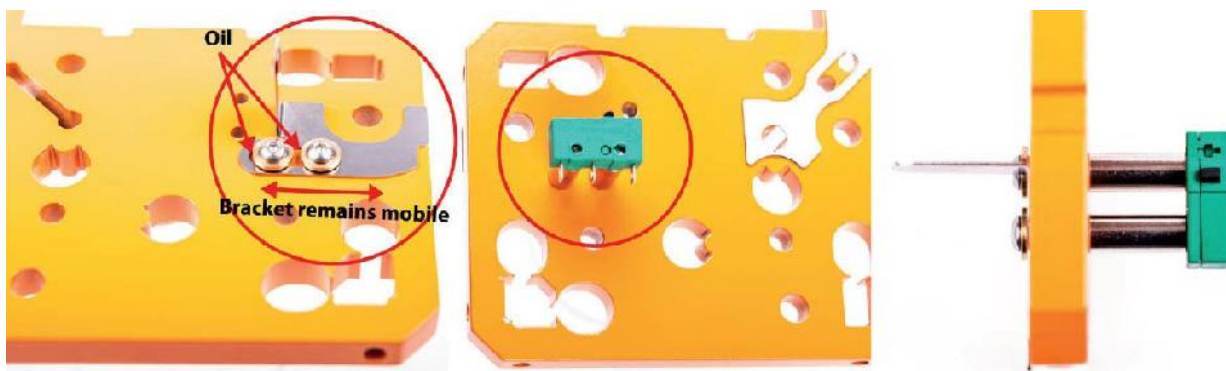


Fig. 2.7 Mounting the X end switch

7. Now mount the Z end switch. This is done exactly the same way as previously described but this time using the remaining (matt) bracket to mount the switch on the other side of the connector. The correct holes and the position of the bracket are shown in [figures 2.6](#) and [2.8](#).
8. Add the two remaining spacers and tighten the bolts.
9. Apply a drop of oil to the retaining rings and move the bracket to spread it around.
10. If you have followed the instructions correctly, the end switches will trigger when you move the brackets inward. The angled tabs on the brackets have to push the black pins on the switches.

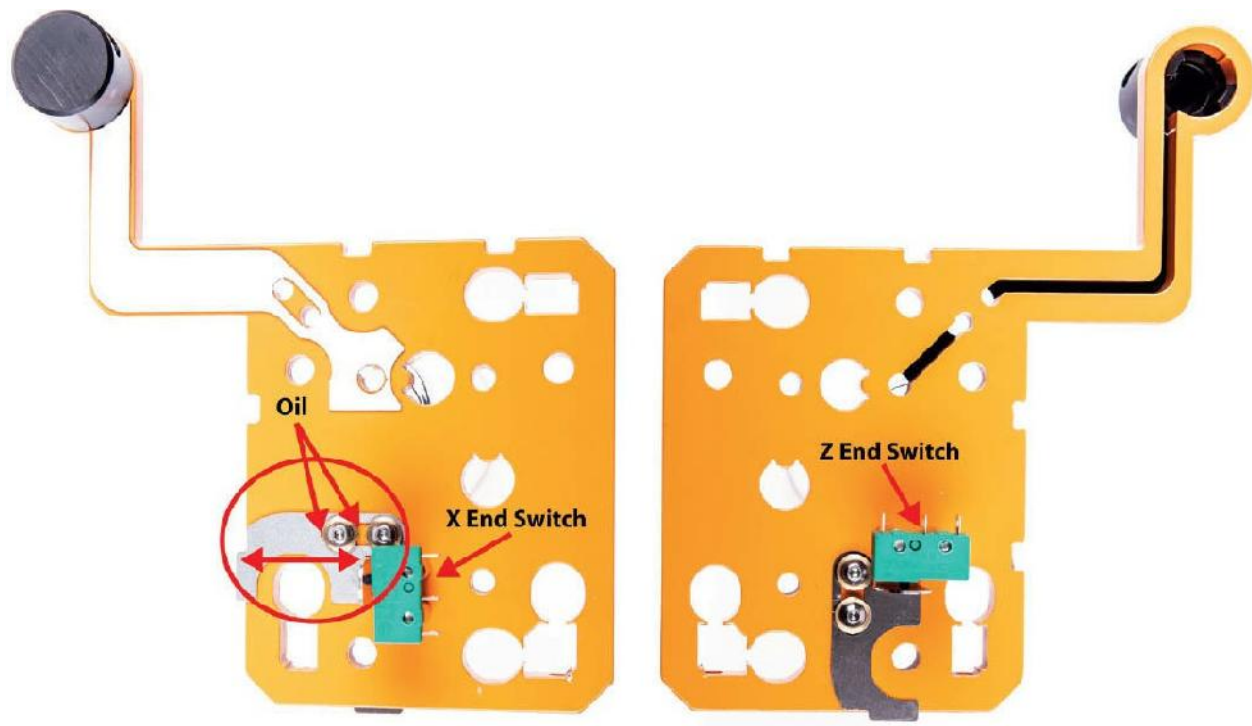


Fig. 2.8 *Mounting the Z end switch*

Please Note

Be sure to tighten the hex bolts firmly and make sure the lead screw nuts are so tight that they cannot be moved or rotated. If the bolts cannot be fully inserted, the manufacturer recommends chamfering the ends.

11. Use 10 mm M6 bolts to attach the lead screw nuts to the X-Z connector. The lead screw nuts are slotted and only fit in one direction in the appropriate holes. You only have to take care to mount them on the correct side of the X-Z connector (see [figure 2.9](#)).

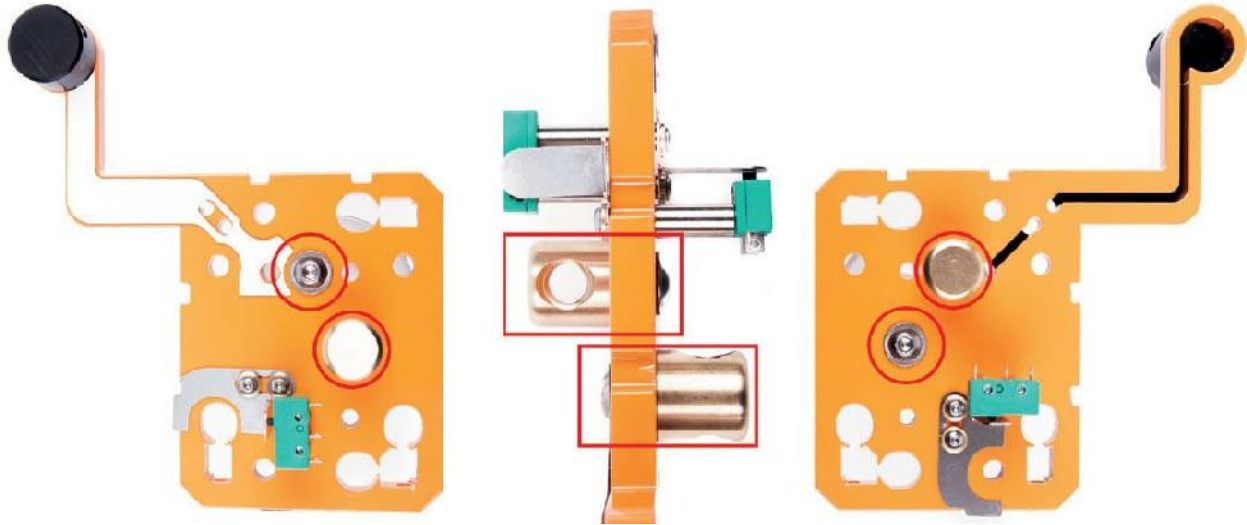


Fig. 2.9 *The mounted lead screw nuts*

- 12.** Fit the connector cables to the end switches. The end switches are connected in series so that the circuit breaks when one of them is triggered. This saves extra cabling but makes it impossible for the control software to identify which switch has been triggered.
- 13.** Connect the cable to the outer pin on the switch on the same side as the black trigger pin.
- 14.** Thread the cable as shown in [figure 2.10](#) and, once again, connect the other end of the cable to the outer pin near the black pin.

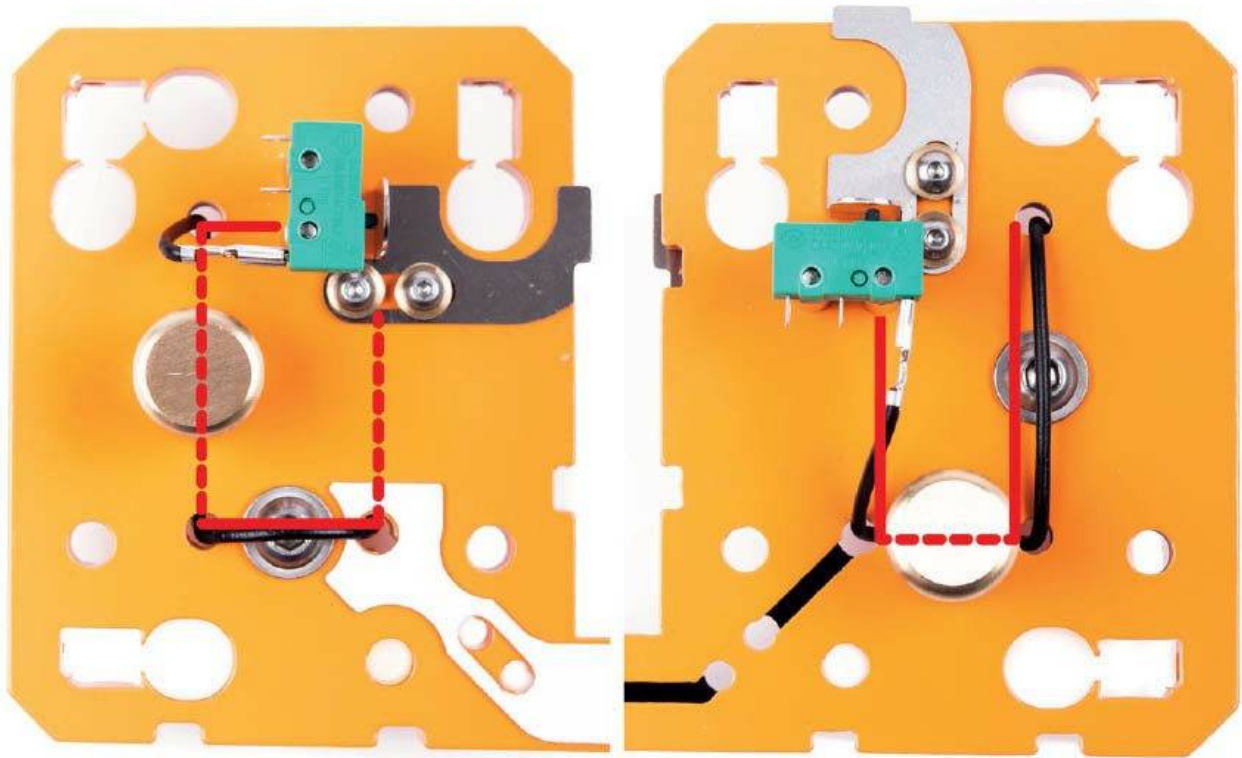


Fig. 2.10 Connecting the end switches

15. Attach the long connector cable to the switches. Thread the end with the flat connectors through the cable collector from the outside. This is easier if you remove the cable collector first.
16. Thread both wires through the cable duct and the holes shown in [figure 2.10](#).

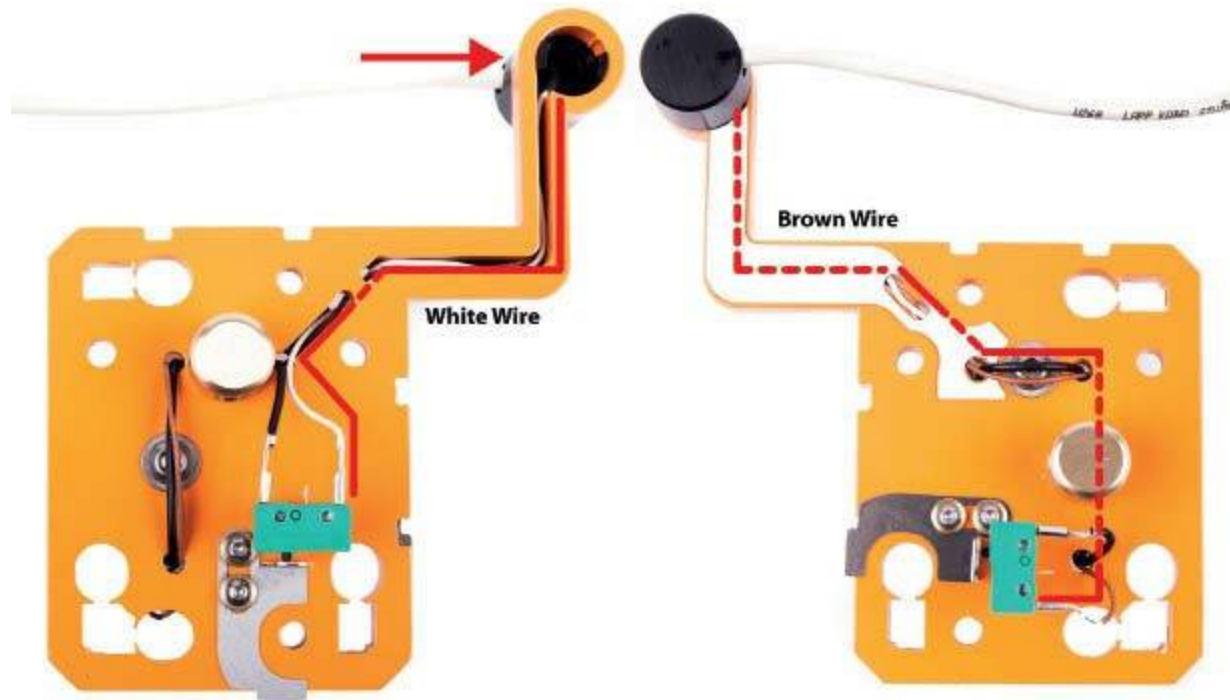


Fig. 2.11 *Fitting the Connector cable*

- 17.** Connect the white wire to the free outer pin on the end switch. Thread the brown wire through the two other holes, as shown in [figure 2.11](#), and connect it to the other free pin.
- 18.** Glue the second cable channel cover to the back of the X-Z connector and fit the other cable collector. The cable duct should now be sealed on both sides.

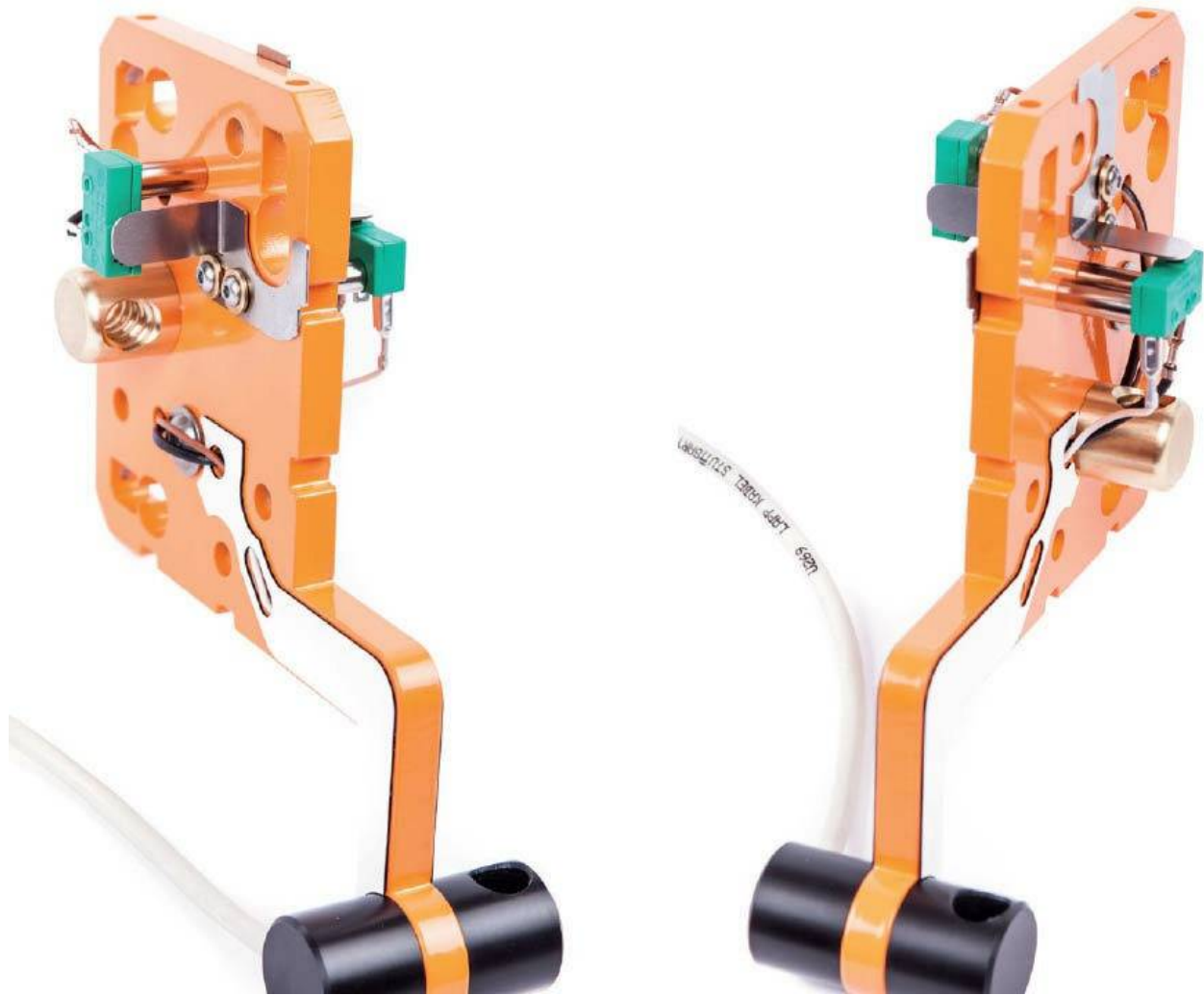


Fig. 2.12 The finished end switch and lead screw nut assembly

Take a break and check your work so far. The next step is to fix the roller holders in place. From each set of four, two are fixed and two remain adjustable.

19. Prepare the four roller holders by inserting a 14 mm M3 bolt and an M3 washer into each as shown in [figure 2.13](#). Make sure that the bolts protrude several millimeters from the mounts.



Fig. 2.13 *Prepare the roller holders*

- 20.** Insert two of the prepared roller holders in the lower two holes as shown in [figure 2.14](#) (photo #1). Use the end switch bracket as a positioning guide. Make doubly sure that the washers are positioned between the bolt head and the X-Z connector. Tighten the M3 bolts a little to ensure that the roller holders don't fall out of their holes during the steps that follow.
- 21.** Turn the X-Z connector over to reveal the long ends of the roller holders. Now add two M8 washers (photo #2) and two rollers (photo #3) to the mounts and fix them tightly in place using 10 mm M6 bolts (photo #4). The rollers should still rotate but shouldn't have any lateral play.

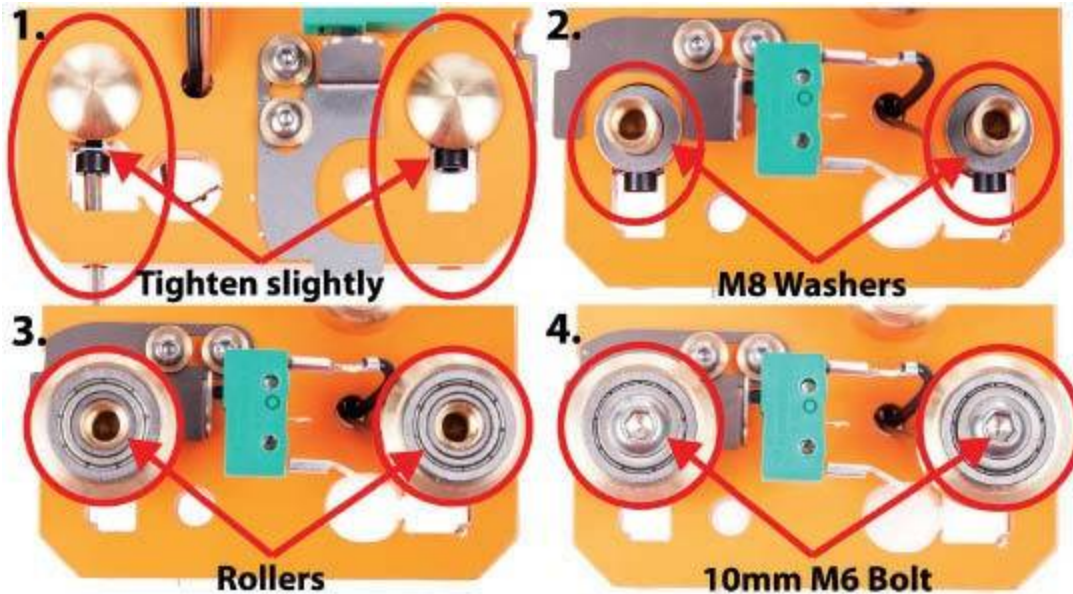


Fig. 2.14 Mounting the first two rollers

22. Mount the next two rollers on the same side of the X-Z connector using 20 mm M6 bolts, as shown in [figure 2.15](#) (photo #1).
23. Mount two rollers and two M6 washers on the bolts and fix them in place using roller nuts. Be sure to use a hex key and an open end wrench to tighten them down *securely*. The rollers should rotate freely but have no discernible play.



Fig. 2.15 Mounting the remaining rollers on the front of the X-Z connector

The other four rollers are mounted the same way on the other side of the X-Z connector, with two mounted freely and the other two tightened firmly. If the lower right-hand roller touches the end switch cable, carefully bend the flat connector out of the way, as shown in [figure 2.16](#).

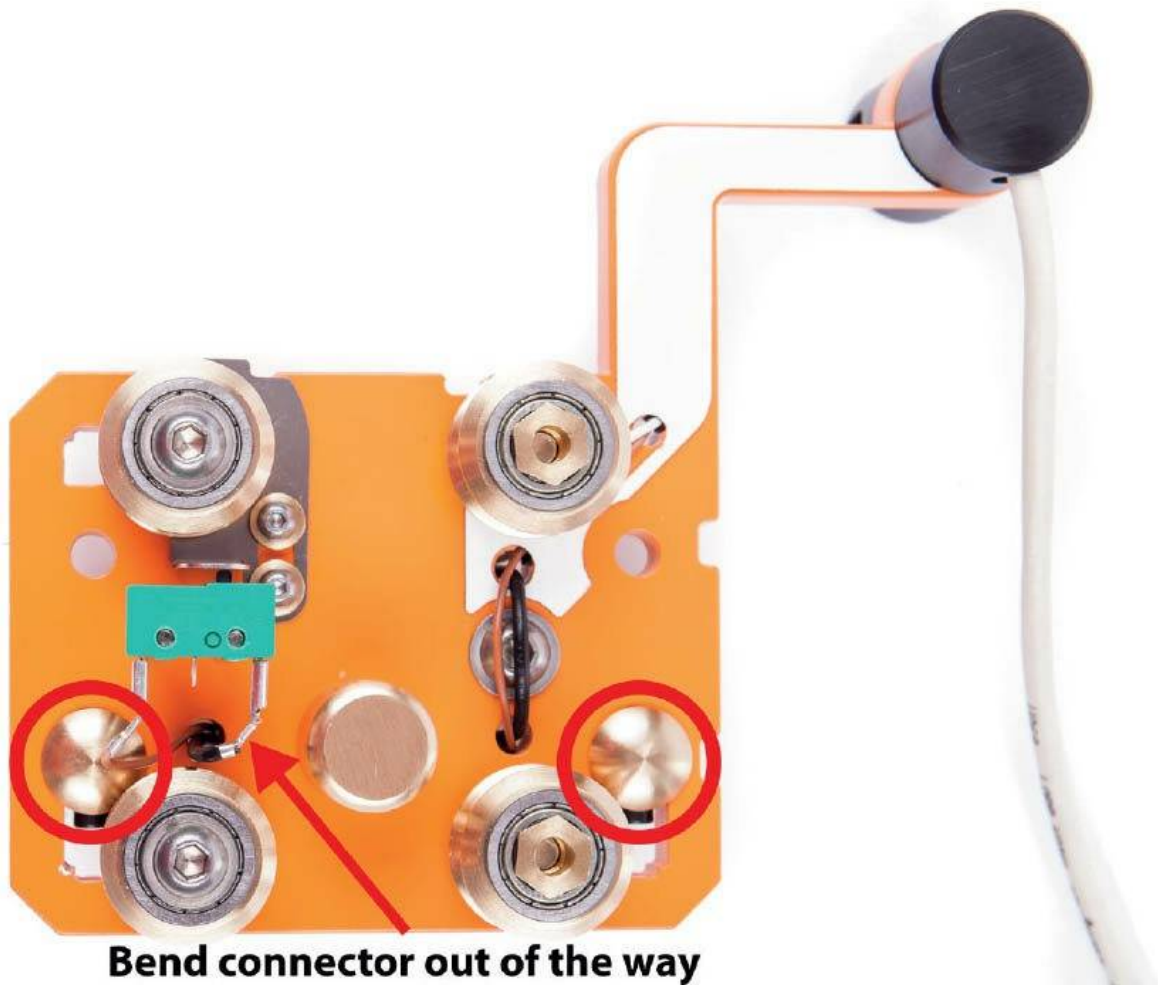


Fig. 2.16 The next two roller holders

24. Place the last two roller holders in the holes indicated in [figure 2.16](#) and gently tighten the M3 bolts.
25. Thread two rollers and two M8 washers onto the other end of the roller holders and fasten them with two 10 mm M6 bolts, as shown in [figure 2.17](#) (photo #1).
26. Thread two 20 mm M6 bolts through the holes shown in [figure 2.17](#) (photo #2), place two M6 washers and two rollers on the bolts and fix them in place using roller nuts (photo #3).

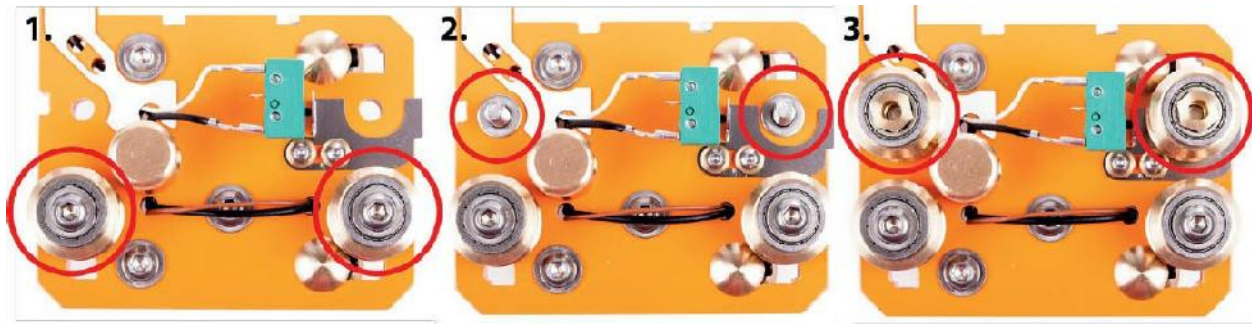


Fig. 2.17 *Mounting the second set of four rollers*

27. Remove the cable collector and thread the brass ring and the cable collector onto the connector cable. Insert the brass ring into the cable collector, as shown in [figure 2.18](#), and reattach the holder to the X-Z connector.

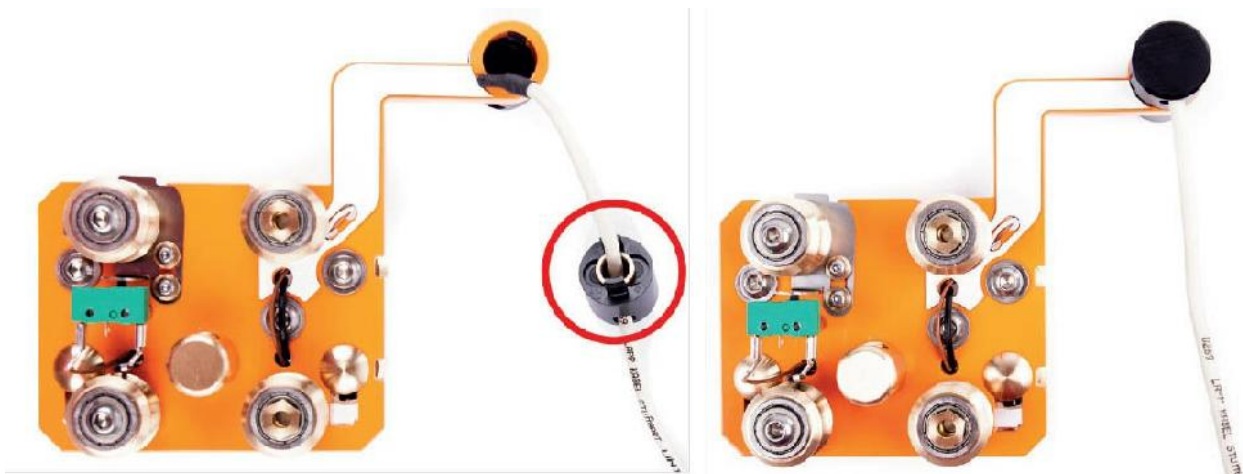


Fig. 2.18 *Mounting the brass ring in the cable collector*

You have now finished the trickiest part of the X-Z connector assembly. Take a break and check your work before going on to align the lead screw nuts.

Please Note

Never use force to align the lead screws. Even the slightest bend in a lead screw will negatively affect your machine's functionality and precision.

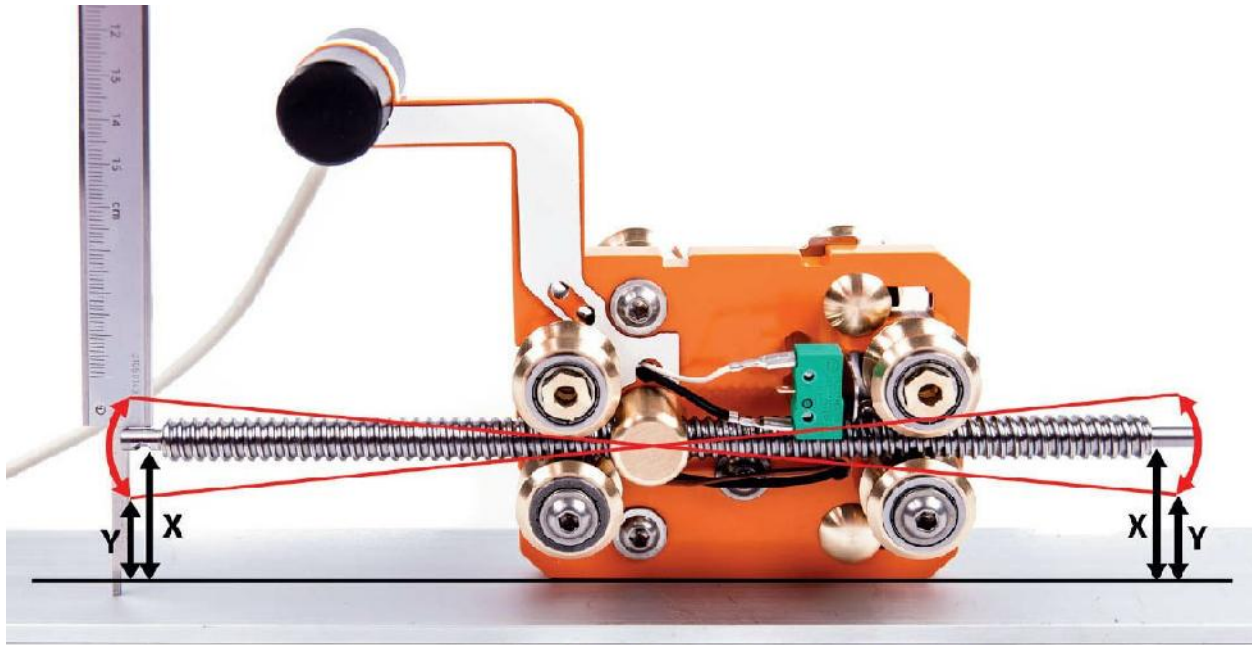


Fig. 2.19 Aligning the lead screw nut for the Z lead screw

28. Screw the shorter M10 lead screw into its nut (see [figure 2.19](#)) until equal lengths protrude from both sides. You now have to rotate the lead screw nut until the distance X is the same at both ends of the lead screw. The distance Y should also be the same if you press each end of the lead screw downward.
29. To adjust the position of the lead screw nut, loosen its bolt and then retighten it until the nut can be rotated using slight pressure. Place the X-Z connector on a flat surface and rotate the lead screw nut until the X and Y measurements match at both ends of the lead screw. Use a caliper to check your measurements. Once the lead screw nut is aligned tighten its bolt firmly, check that you haven't shifted its position during the tightening process and then, once you are sure everything is OK, remove the lead screw.

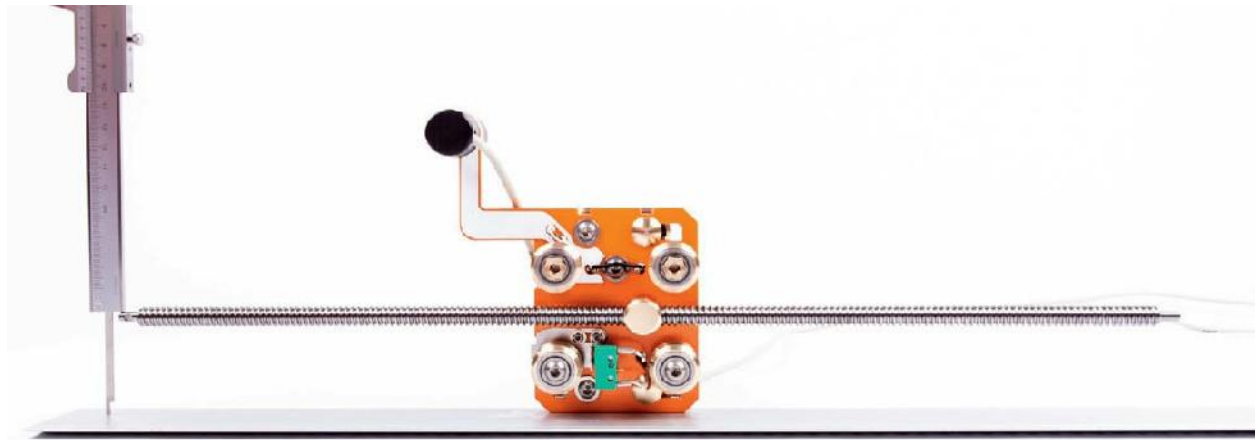


Fig. 2.20 Aligning the X lead screw nut

- 30.** Align the longer X lead screw the same way and remove the lead screw once it is set up correctly.

The X-Z connector is now ready for final assembly and we can move on to mounting the Z axis.

2.4 Step 2: Mounting the Z Axis

The Z axis consists of a guide rail, two motor mounts, the X-Z connector, and the Z motor. The Z axis determines the height of the spindle above the machine table and is mounted so that it moves back and forth along the X axis.



Fig. 2.21 The parts required to assemble the Z axis

1. Begin by fixing the thrust bearing to the motor using superglue. Take care to apply glue only to the outer ring on the bearing and make sure no glue runs onto the motor or the shaft. Thread the thrust bearing over the shaft into the opening in the motor casing, then hold it in place for a few seconds until the glue sets.
2. Apply a drop of 3-IN-ONE oil to the running surface of the thrust bearing as shown by the red arrow on the right in [figure 2.22](#).

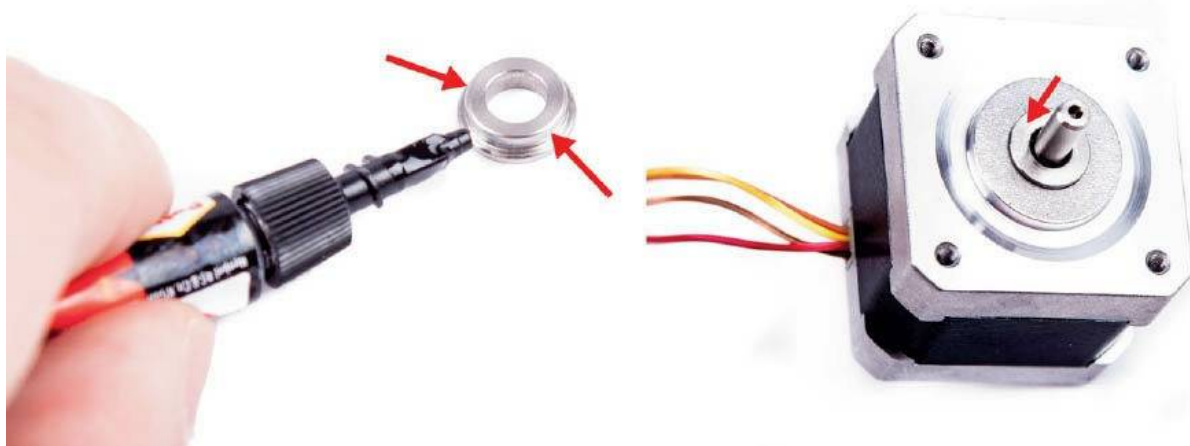


Fig. 2.22 *Inserting the thrust bearing*

3. Thread the shaft connector onto the motor shaft and insert an M4 grub screw, making sure the grub screw is positioned on the flattened side of the shaft. Make sure the shaft connector lies absolutely flush with the thrust bearing and tighten the grub screw securely. If you can move the shaft connector along the shaft's axis, it is either not flush or not sufficiently tight.
4. Insert the 220 mm lead screw into the shaft connector, making sure the hole in the lead screw is lined up with the hole in the connector. Insert an M4 grub screw into the hole and tighten it as securely as possible.

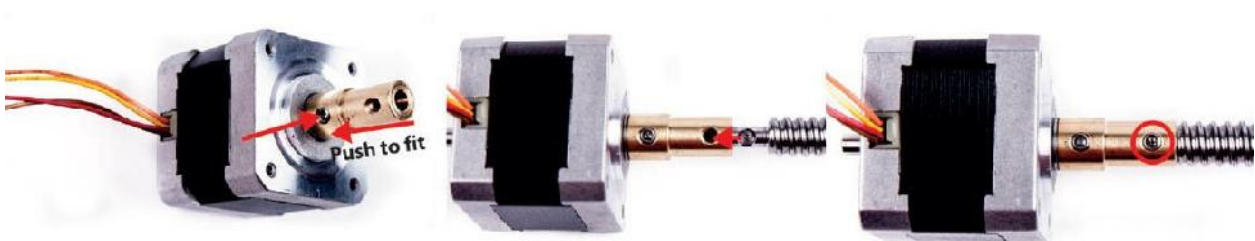


Fig. 2.23 *Attaching the Z lead screw to the Z motor*

Model Differences

In the Stepcraft-2/300, the cable conduit is 160 mm (6.3") long. In all other models, it measures 220 mm (8.7").

5. Take the corrugated cable conduit and thread the motor cable

through it.

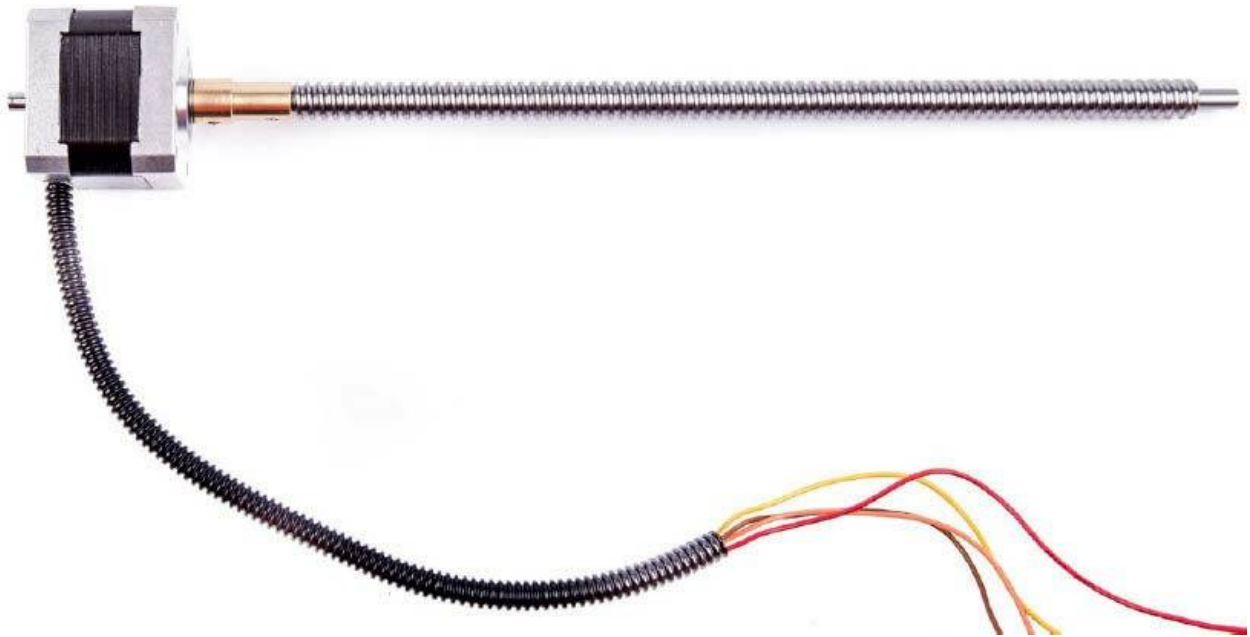


Fig. 2.24 Corrugated tube used as a cable conduit

6. Place the Z motor holder over the shaft as shown in [figure 2.25](#) and thread the cable through the hole in the motor holder. Make sure the motor and the motor holder are positioned correctly and fasten the motor in place using four 12 mm M3 bolts.



Fig. 2.25 The Z motor mounted on its holder

7. Attach the lead screw motor holder to the linear rail using three 25 mm M6 bolts, but don't tighten them completely just yet. Make sure that the guide slot on the linear rail meets the small peg on the lead screw motor holder.
8. Adjust the position of the lead screw motor holder so that there is a small overhang all around its edge (see [figure 2.26](#)), then tighten the bolts.

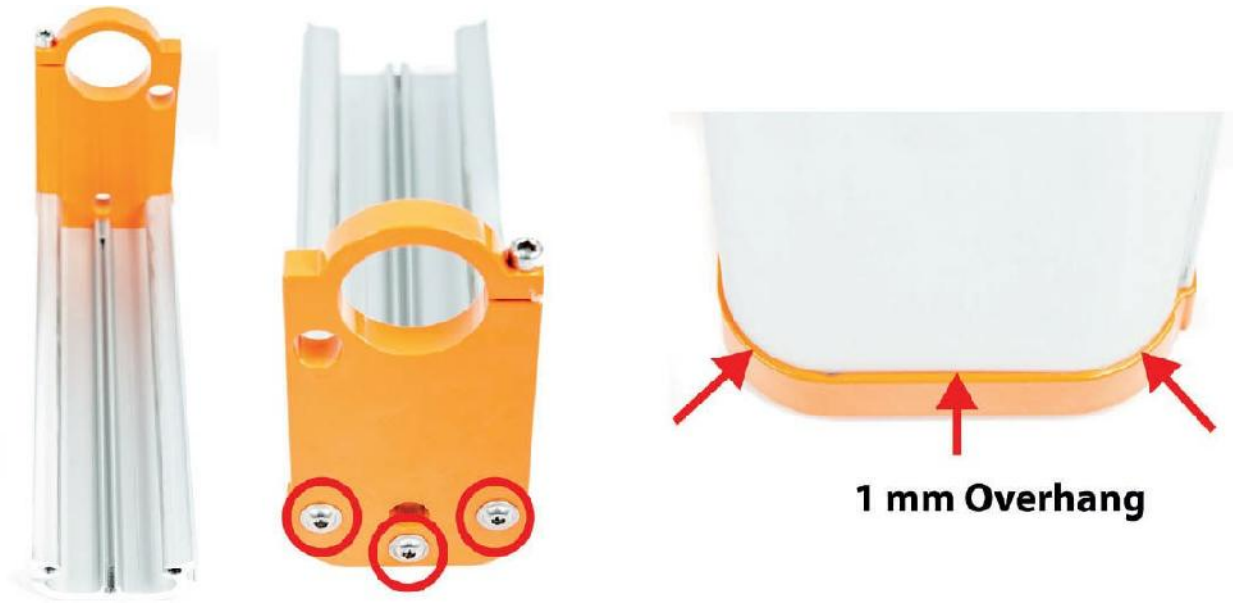


Fig. 2.26 *Attaching the lead screw motor holder*

9. On the X-Z connector, loosen the roller holders indicated in [figure 2.27](#) so that the rollers have sufficient play (1-2 mm) to be inserted into the linear rail. For this step, it is important that all the rollers have a little play.
10. Insert the X-Z connector completely into the rail and retighten the M3 bolts bit by bit while moving the X-Z connector back and forth until the rollers no longer have any play. The rollers will click against the rail if there is still play in the system. Perform this step carefully and, if you are not sure you got it right, loosen the screws and try again, tightening the bolts a quarter turn at a time to ensure that there is sufficient pressure on the rollers.

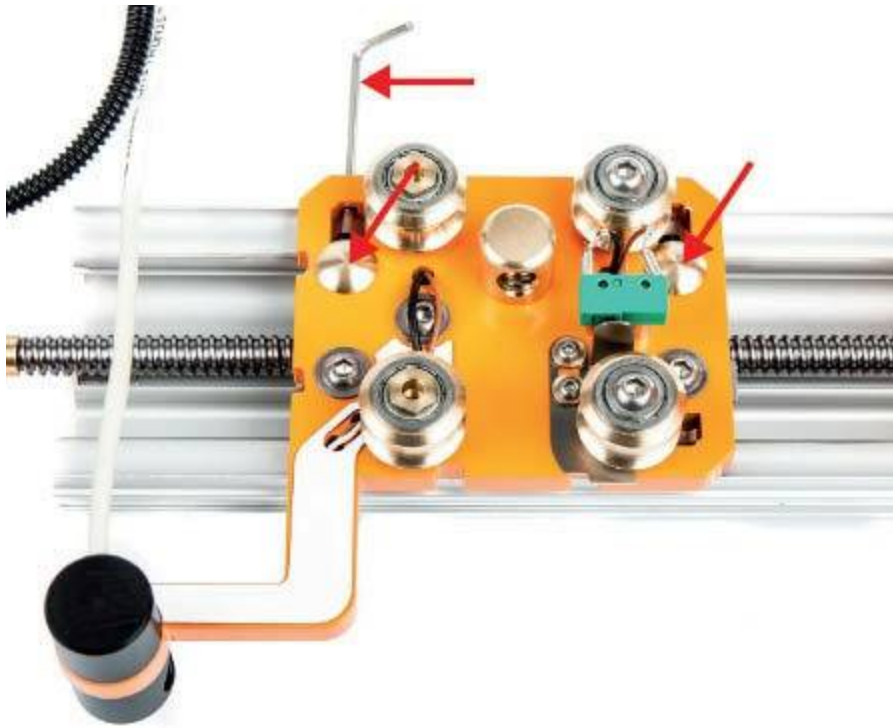


Fig. 2.27 *Fitting and adjusting the X-Z connector*

- 11.** Mount the assembled Z motor and screw its lead screw into the nut on the X-Z connector until the X-Z connector sits in the center of the lead screw.
- 12.** Insert the remaining bearing into the lead screw motor holder, as shown in [figure 2.28](#).
- 13.** Move the Z motor/X-Z connector assembly along the rail until the end of the lead screw fits into the hole in the bearing. Fasten the Z motor holder to the rail using two 20 mm M6 bolts.

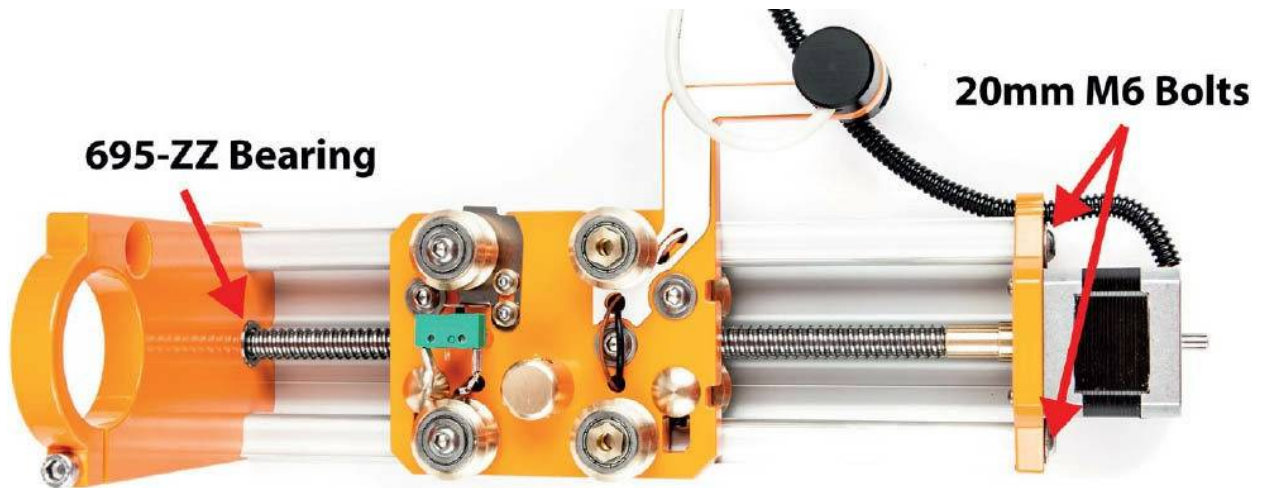


Fig. 2.28 *The finished Z axis assembly*

14. We now need to organize the cables. Remove both cable collectors and thread the cable from the Z motor through the as-yet unused cable collector. Now thread the Z motor cable through the hole in the X-Z connector, through the brass ring, and then through the second cable collector (the one near the end switch cable), as shown in [figure 2.29](#). Now fit both cable collectors back onto the X-Z connector.
15. Use the tube clip to secure the short tube from the Z motor in the slot on the side of the rail. The distance between the motor holder and the clip should be about 2" (50 mm).
16. Cut the 10 mm corrugated conduit tube to the correct length for your machine and thread all the Z-axis cables through it. This is easier to do if you first wrap a piece of tape around the cable ends. Now thread the tube into the hole in the cable collector.

Model Differences

The required length of the 10 mm corrugated conduit tube varies from machine to machine:

Stepcraft-2/300: 310 mm (12.2")
 Stepcraft-2/420: 400 mm (15.7")
 Stepcraft-2/600: 520 mm (20.5")
 Stepcraft-2/840: 700 mm (27.6")



Fig. 2.29 *Organizing the cabling*

That's it! The Z axis assembly is now complete and we can move on to the gantry.

2.5 Step 3: Assembling the X Lead Screw and the Sides of the Gantry

The gantry is made up of the Z axis, the X axis, two side panels, and the Y axis rail.

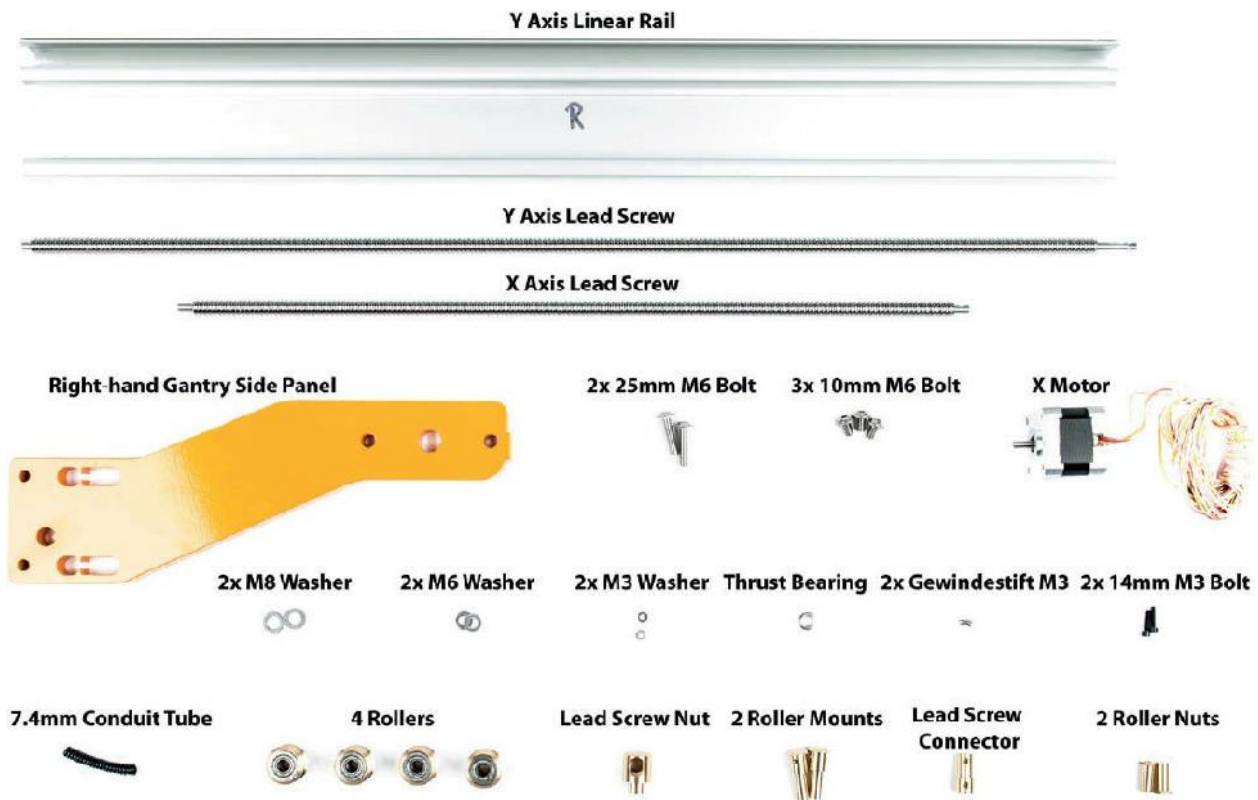


Fig. 2.30 *The parts required for the X axis and the right-hand side of the gantry*

1. The X axis is assembled the same way as the Z axis (see [figures 2.22](#) and [2.23](#)). Mount the thrust bearing, lead screw connector, and lead screw the same way too, and don't forget to apply a drop or two of oil to the face of the bearing. Make sure that there is no play between the shaft connector and the bearing.
2. Thread the motor cable into a 45 mm (1.8") piece of corrugated conduit.



Fig. 2.31 The finished X motor/lead screw assembly

3. Mount the lower rollers on the right-hand side panel. Thread the 25 mm M6 bolts through the holes and place M6 washers and rollers on the other ends of each. Fasten the rollers in place with roller nuts.

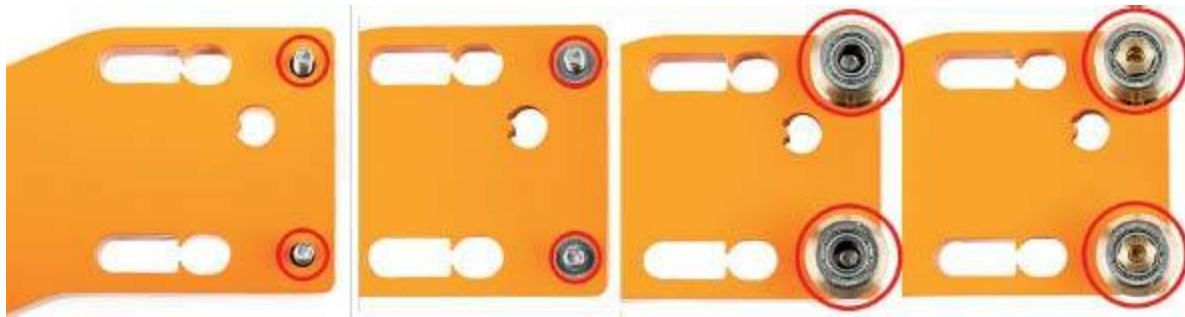


Fig. 2.32 Mounting the lower rollers

4. Place the lead screw nut in its hole and fasten it with a 10 mm M6 bolt, as shown in [figure 2.33](#).



Fig. 2.33 Attaching the lead screw nut

5. Insert 14 mm M3 bolts and M3 washers into the threads in the roller holders, leaving the bolts protruding.



Fig. 2.34 *Preparing the roller holders*

6. Insert the roller holders into their holes in the gantry side panel, as shown in [figure 2.35](#) (on the left), making sure that the washers are located between the head of the bolt and the panel.
7. Place an M8 washer and a roller on the other end of each roller holder and fasten them in place using 10 mm M6 bolts.

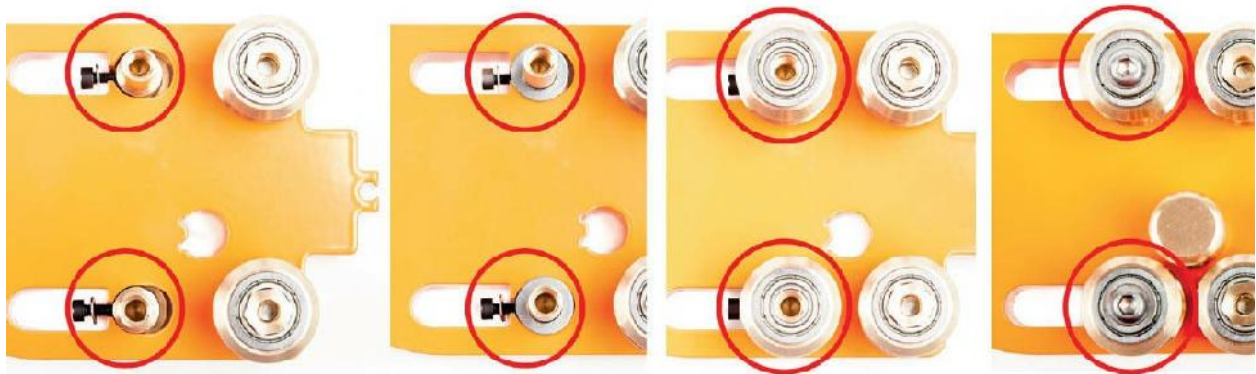


Fig. 2.35 *Mounting the upper rollers*

8. Label the Y axis rail so you can easily identify it as the right-hand one later on. This is important because we are about to adjust the play in the rollers, and the adjustments might not be correct if you unintentionally switch the rails during assembly.
9. Insert the side panel with the fitted rollers into the Y rail. If it doesn't fit, loosen the M3 bolts and move the rollers downward. At this point, all the rollers need to have 1-2 mm of play.

10. Tighten the M3 bolts bit by bit until the rollers no longer have any play. Move the side panel back and forth regularly while you are tightening the bolts. The rollers will click against the rail if there is still play in the system. Perform this step carefully, as it directly influences the quality and precision of your milling. If you are not sure whether the bolts are tight enough, loosen them and repeat the tightening procedure.



Fig. 2.36 Adjusting the play in the right-hand gantry rollers

11. Remove the side panel from the rail and put the rail aside. Thread the Y lead screw through the lead screw nut on the side panel until the nut is positioned in the center of the lead screw.
12. Adjust the lead screw nut until the offset distances X and Y (see [figure 2.19](#)) are identical at both ends. As previously, make sure that you don't alter the position of the aligned nut when you tighten its fixing bolt. Once the lead screw is correctly aligned,

remove it from the nut.



Fig. 2.37 Aligning the lead screw nut on the right-hand gantry side panel

Assembly of the left-hand side of the gantry is almost the same as the right-hand side.

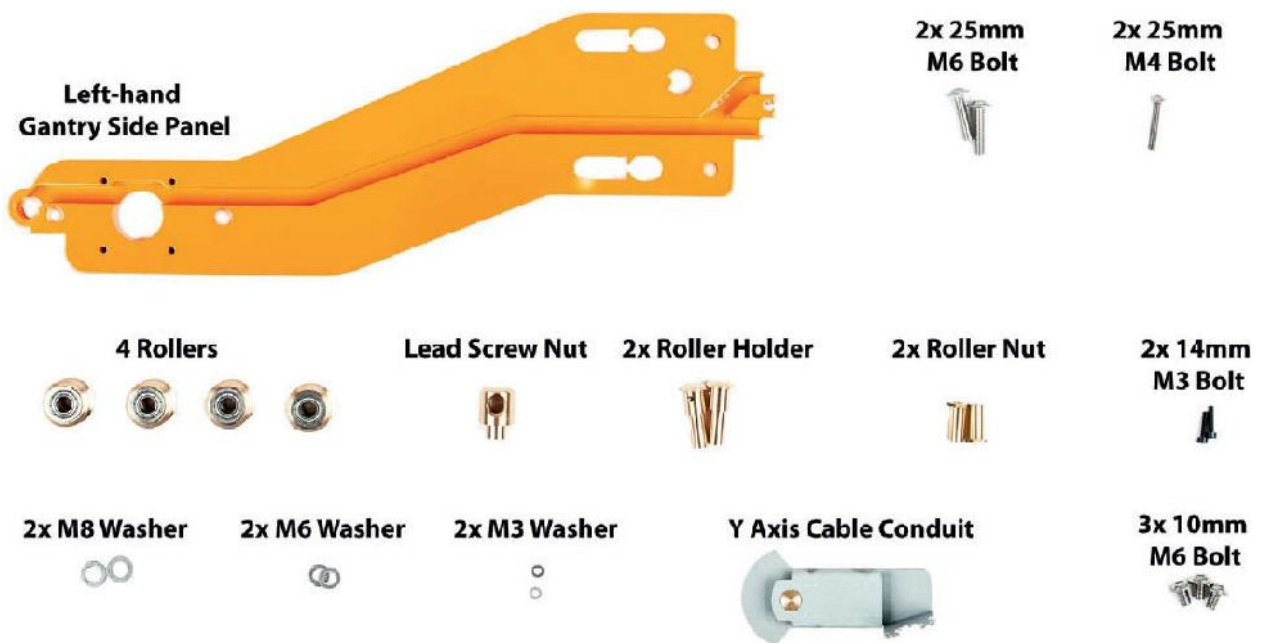


Fig. 2.38 The parts required to assemble the left-hand side of the gantry

13. Mount the two lower rollers on the left-hand gantry side panel.

Thread two 25 mm M6 bolts through the lower holes and place M6 washers and rollers on the opposite ends of the bolts. Add two roller nuts and tighten them securely.

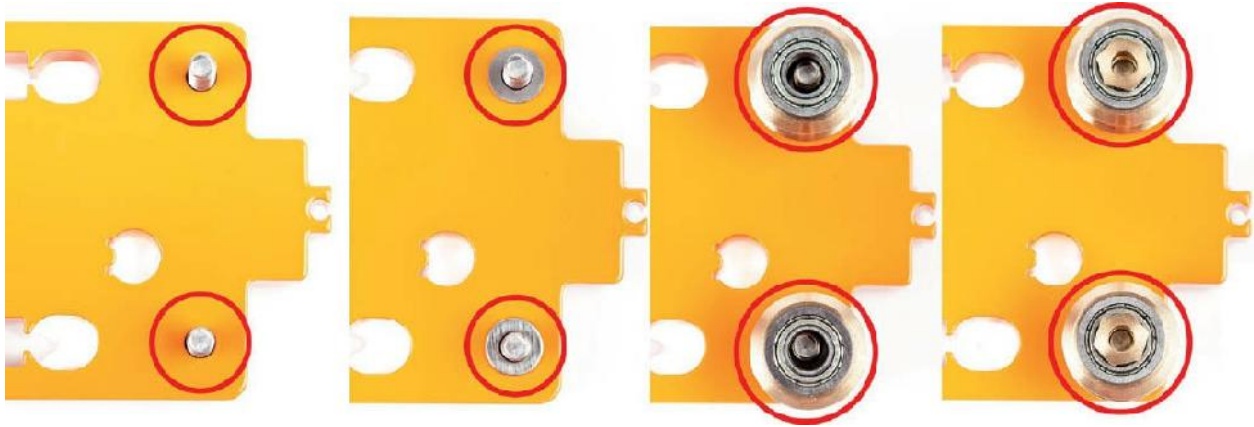


Fig. 2.39 *Mounting the lower rollers*

- 14.** As previously, prepare the upper roller holders by inserting 14 mm M3 bolts with M3 washers.
- 15.** Insert the roller holders in the holes indicated in [figure 2.40](#), making sure that the washers are located between the side panel and the bolt heads.
- 16.** Place an M8 washer and a roller on the other end of each bolt and fasten them securely using 10 mm M6 bolts.

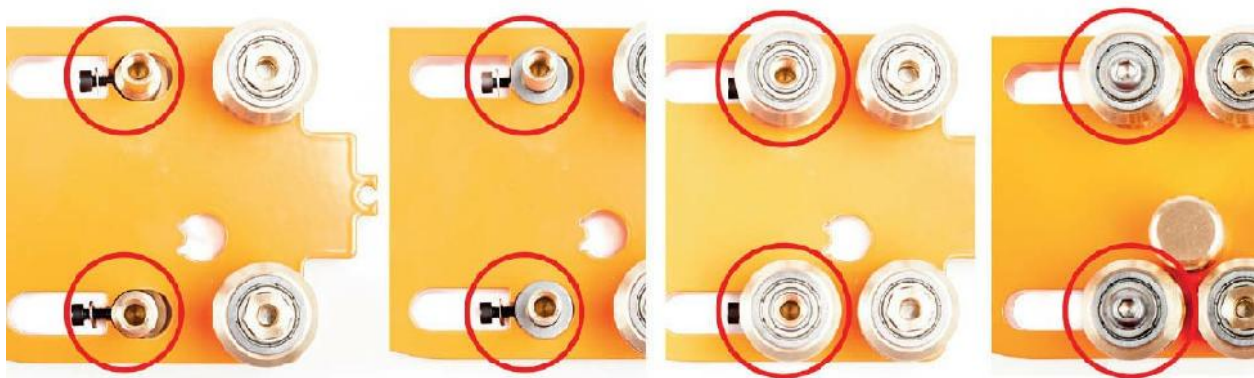


Fig. 2.40 *Mounting the upper rollers*

- 17.** Place the lead screw nut in its hole and fasten it using a 10 mm M6

bolt.



Fig. 2.41 Mounting the lead screw nut

- 18.** Insert the 25 mm M4 bolt through the hole at the lower end of the side panel and use it to fasten the cable conduit in place. Make sure the flat end of the fastener lies flush with side panel.



Fig. 2.42 *Mounting the cable conduit on the left-hand side panel of the gantry*

19. Thread the lead screw into the lead screw nut until it is positioned centrally. Adjust play in the nut as you did for the right-hand side panel. Take care with this step and check that you haven't shifted your setting while tightening the bolt. If necessary, loosen the bolt and repeat the tightening procedure. Once you are done, remove the lead screw and put it aside.

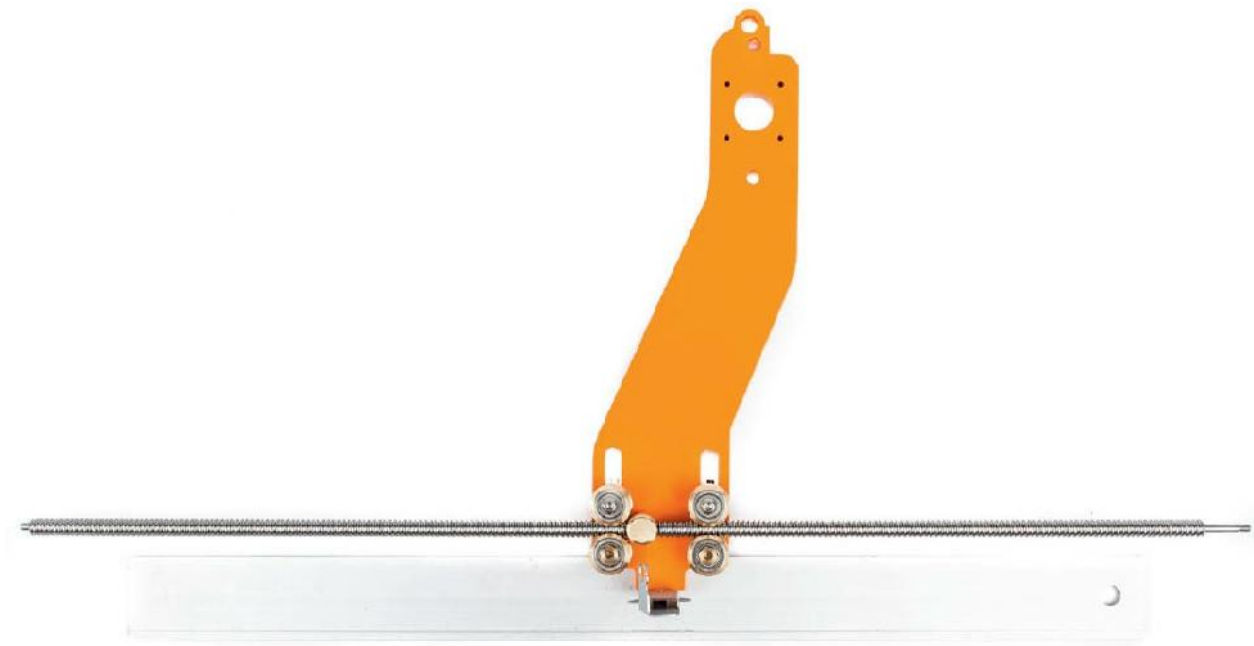


Fig. 2.43 *Aligning the lead screw nut on the left-hand gantry side panel*

20. Once again, label the rail so you can tell which is which during final assembly. Make sure all four rollers have 1 mm-2 mm of play before inserting the side panel into the rail.
21. Use a hex key to successively tighten the M3 bolts until you cannot detect any play.



Fig. 2.44 Adjusting play in the left-hand gantry side panel

2.6 Step 4: Assembling the Gantry

The individual gantry components are now ready to be assembled.

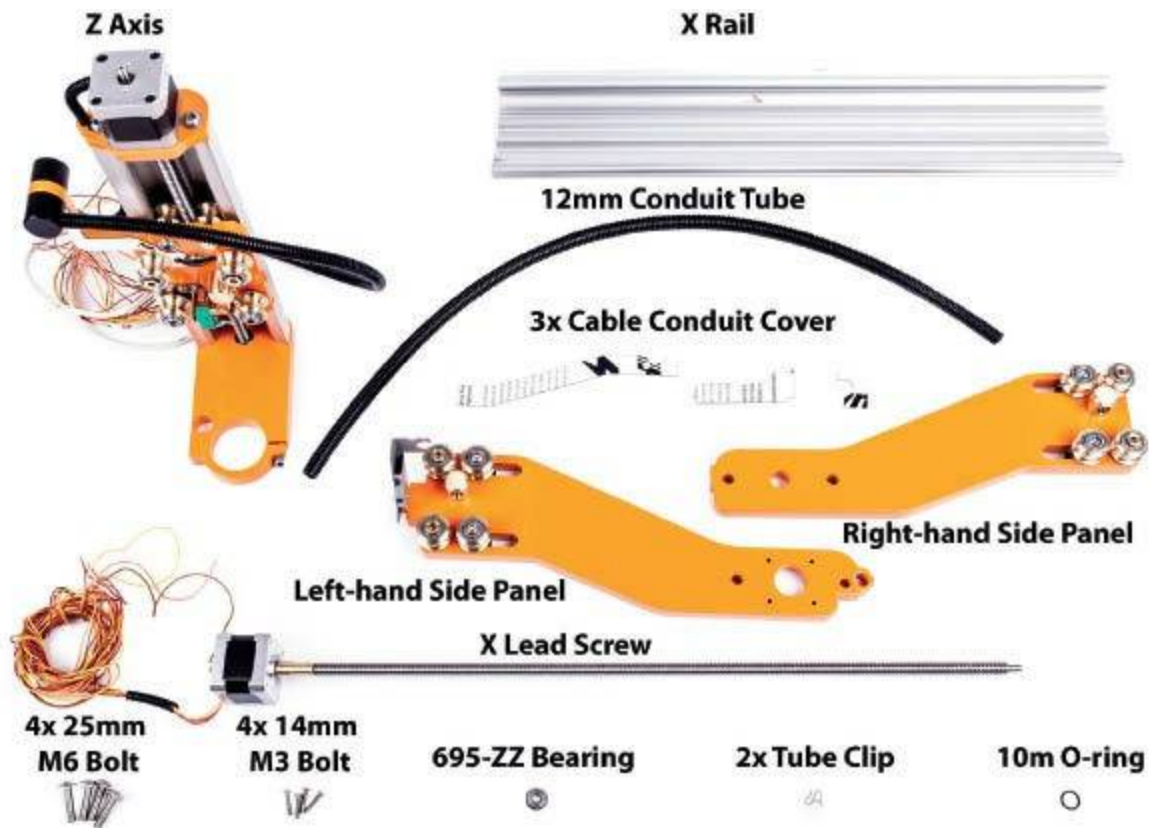


Fig. 2.45 The preassembled parts of the gantry

1. Insert the Z axis into the rail with the slot in the side of the rail facing upward. Make sure that all four rollers have 1 mm-2 mm of play.
2. Tighten the M3 bolts successively until there is no more play in the rollers. Perform this step carefully, as it can have a direct effect on the functionality and precision of the workpieces you produce. Once you cannot detect any play, tighten the bolts an extra quarter turn.



Fig. 2.46 Adjusting play in the X axis

3. Fasten the left-hand side panel to the rail using two 25 mm M6 bolts. Thread the Z axis cable through the hole at the top end of the panel.



Fig. 2.47 Mounting the left-hand side panel

4. Use two 25 mm M6 bolts to fasten the right-hand side panel in

place.



Fig. 2.48 The assembled right-hand side panel

Be sure to mount the side panels with a constant overhang of about 1 mm at the edge of the rail.



Fig. 2.49 Side panel overhang

5. Use two clips to fasten the Z cable conduit in the upper slot in the X rail. The distance between the left-hand side panel and the first tube clip varies from machine to machine (see the box on the right). Place the second clip equidistant from the first clip and the side panel.

Model Differences

The distance between the tube clip and the side panel varies from machine to machine:

Stepcraft-2/300: 120 mm (4.7")

Stepcraft-2/420: 155 mm (6.1")

Stepcraft-2/600: 205 mm (8.1")

Stepcraft-2/840: 280 mm (11")

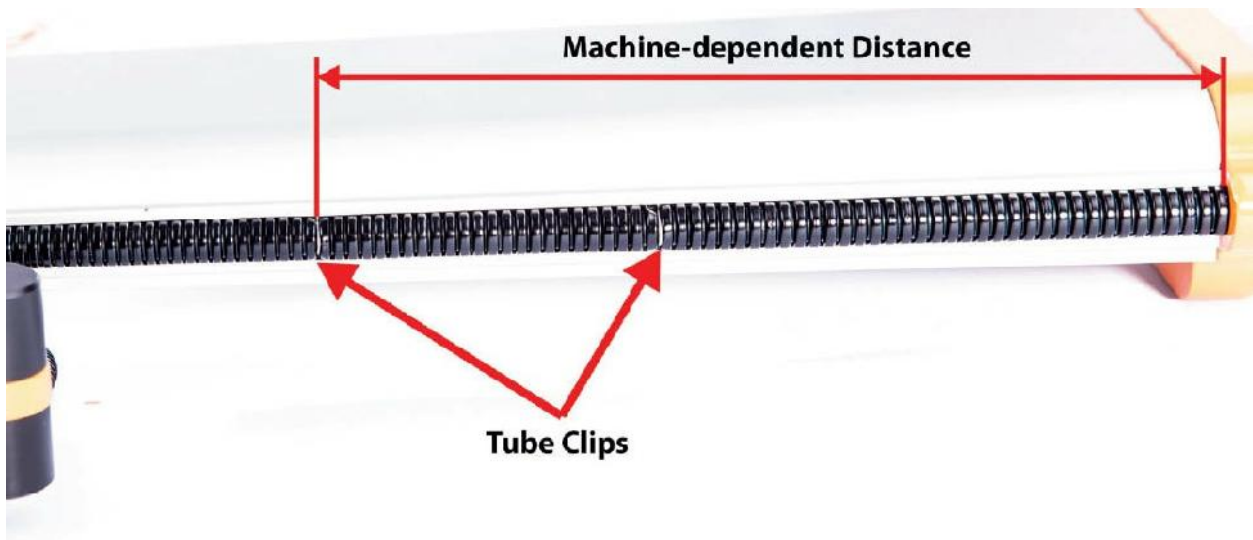


Fig. 2.50 Fixing the cable conduit in place

6. As shown in [figure 2.51](#), thread the X lead screw through the large hole in the left-hand side panel and between the Z axis rollers. Insert the lead screw 6-8 inches (15-20 cm) into the lead screw nut.

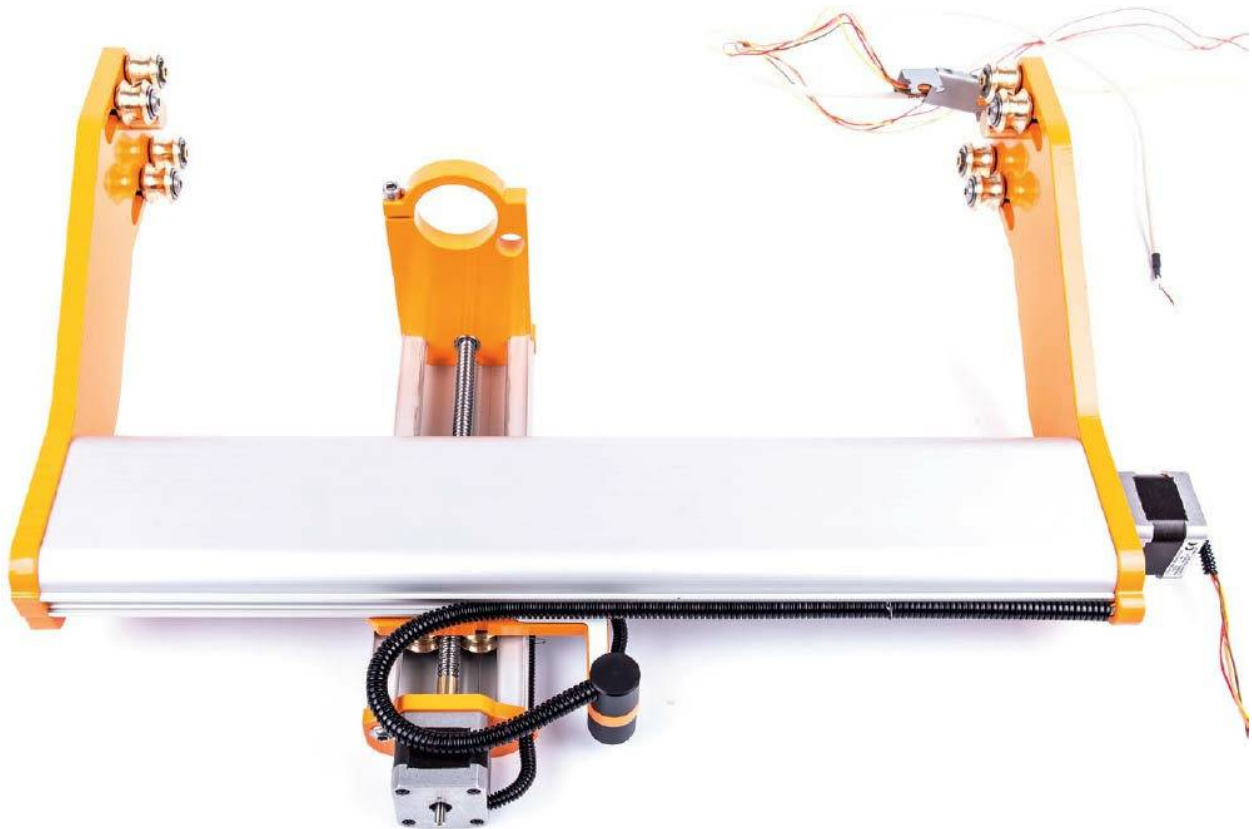


Fig. 2.51 *The mounted X lead screw*

7. Place the bearing in the opening near the end of the X lead screw and, together with the Z axis, push the X lead screw into the bearing.
8. Thread the Z axis cable through the conduit in the left-hand side panel.
9. Use four 16 mm M3 bolts to fasten the X motor to the outside of the side panel. If the lower bolts catch on the rail, loosen the bolts holding the rail first and tighten them again once you have mounted the motor.

Please Note

Make sure the Z axis cable conduit is threaded beneath the motor and not between the motor and the side panel, where it can easily snag.

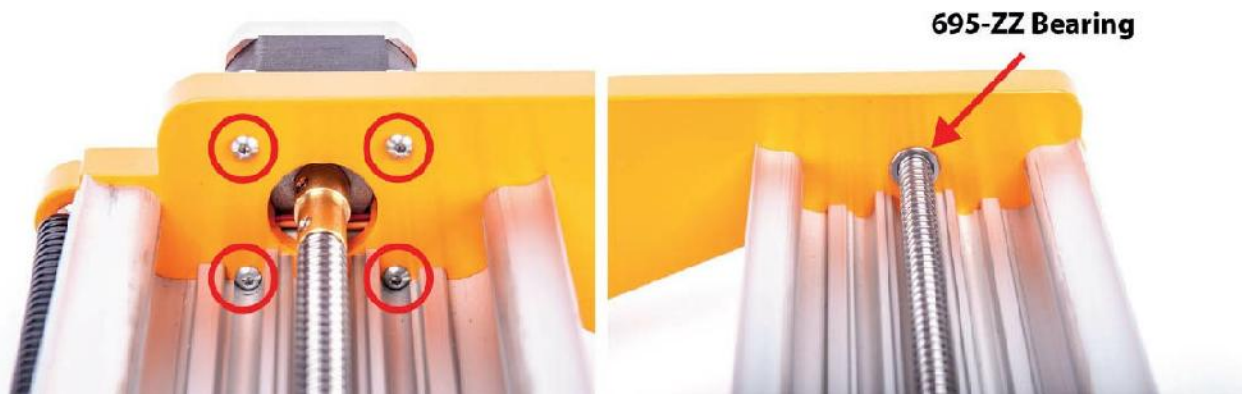


Fig. 2.52 Both ends of the assembled X lead screw

10. Lay the cables into the conduit and stick the self-adhesive covers into place above and below the motor, as shown in [figure 2.53](#). The longer of the two covers has a hole for the X motor cable.

Note

If you get grease from the lead screws or other parts on your fingers, clean your hands and then the area on the side panel surrounding the cable conduit. If the glue on the conduit cover gets greasy, it won't stick. It is easier to stow the motor cables if you tape the four wires together first. Taping the ends of the wires together also makes it easier to thread them through the holes in the panels. Labeling the motor cables makes it easier to connect them correctly later on.



Fig. 2.53 The stowed and covered motor cables

11. Thread the thick gray cable straight through the lower end of the side panel and into the flexible cable conduit you mounted earlier (see [figure 2.54](#)). The motor cables fork off and are threaded

through the other side of the conduit.



Fig. 2.54 Laying and threading the cables

12. Stick the shorter self-adhesive cover in place.



Fig. 2.55 The completed cabling in the left-hand side panel

13. Cut the large (12.8 mm) conduit tube to the correct length (see the box on the right) and thread all the gantry cables through it. Clip the end of the tube to the flexible conduit as shown in [figure 2.56](#) and fix it in place using the 10 mm o-ring.

Model Differences

The 12.8 mm conduit tube has to be cut to length to suit different machines:

Stepcraft-2/300: 350 mm (13.8")

Stepcraft-2/420: 470 mm (18.5")

Stepcraft-2/600: 650 mm (25.6")

Stepcraft-2/840: 830 mm (32.7")



Fig. 2.56 Routing the cable from the gantry to the frame of the machine

Congratulations! You have now successfully assembled the trickiest parts of your machine, so you deserve a break. Take time to check everything you have done so far and familiarize yourself with the parts before you begin assembling the main frame.

2.7 Step 5: Assembling the Main Frame

The main frame of the machine supports the gantry and moves it along the Y axis. It also contains the machine's electronics and major connectors. The following sections explain how to assemble the main frame (apart from the front panel) and how to install the gantry with the Y motor.

Due to the complexity of this part of the assembly process, the required parts are shown separately in [figures 2.57, 2.58, and 2.59](#).

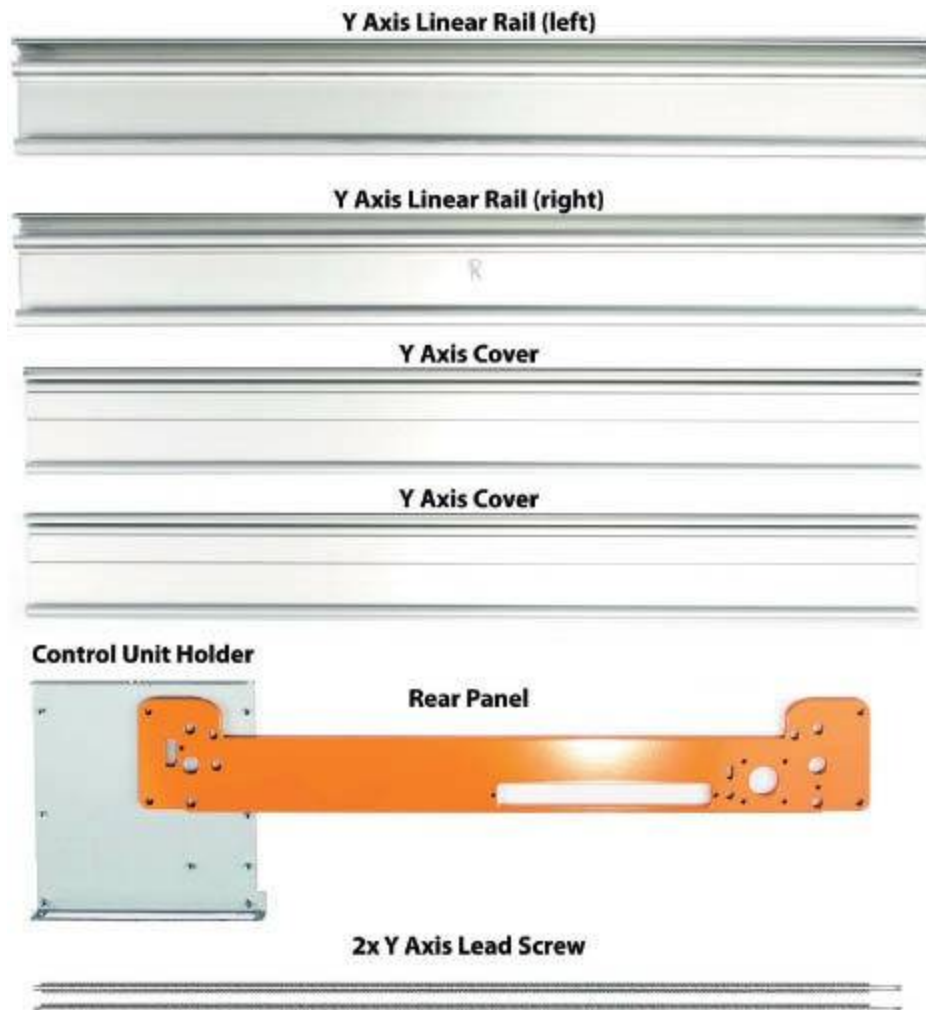


Fig. 2.57 *The major components of the main frame*

Caution!

Take care when handling the punched metal parts. These have sharp edges that can cause cuts. Check the parts before assembly and file or grind sharp edges if necessary.



Fig. 2.58 The minor components of the main frame



Fig. 2.59 The base plate and its associated odds and ends

1. Begin by inserting a 5 mm M3 grub screw into one of the belt pulleys and attach it to the shaft of the Y motor. Make sure the

grub screw is facing the flattened face of the shaft and that the shaft doesn't protrude from the pulley.



Fig. 2.60 Attaching a belt pulley to the Y motor

2. Attach the bracket to the end switch using two 6 mm M3 screws.
3. Fasten the mounted end switch to the back of the rear panel using a 12 mm M3 bolt and a 15 mm M3 spacer (see [figure 2.61](#)).
4. Attach the second 15 mm M3 spacer using a 12 mm M3 bolt.

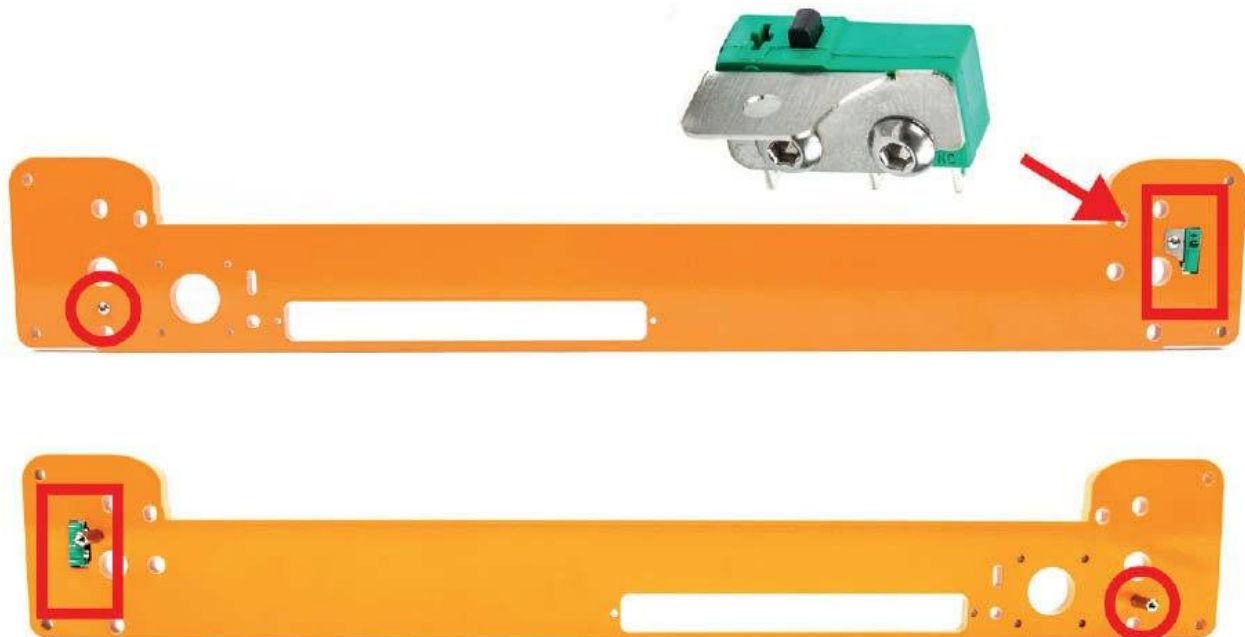


Fig. 2.61 The Y axis end switch

5. Fasten the Y motor to the rear panel using four 12 mm M3 bolts (see [figure 2.62](#)). Make sure the motor cables face toward the center of the panel.

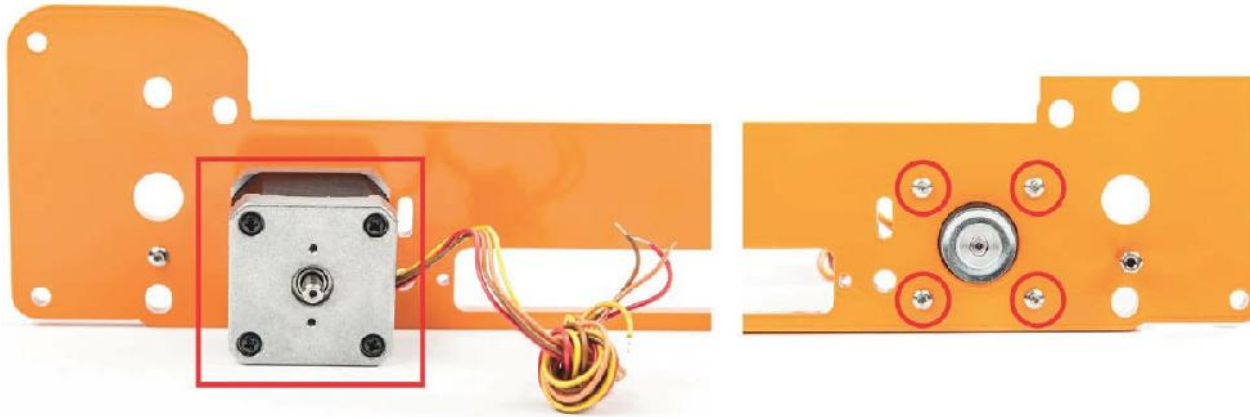


Fig. 2.62 The mounted Y motor

6. Thread the three 604-ZZ bearings and the M4 washer onto the 25 mm M4 bolt, thread it through the hole beside the Y motor, and fasten it using the square nut, as shown in [figure 2.63](#). The bearings keep the toothed belt that drives the Y lead screws tensioned.



Fig. 2.63 The drive belt tensioner

7. Attach the control unit bracket to the rear panel using two 12 mm M3 bolts, as shown in [figure 2.64](#).



Fig. 2.64 The holder for the control unit

8. Cut four pieces of rubber seal to match the length of the Y rail, taking care not to cut it while it is tensioned. The seal has a bulge on one side that fits into the slots in the Y rail and the Y axis covers. Insert the seal into each slot. This is easier to do if you pull it as close as possible to the metal. Inserting the seal causes uneven tension along its length, so you have to jiggle it within the slot until there is no longer any overhang at the ends.

Model Differences

The Y rails have different lengths according to which model machine you have. The rubber

seals have to match the following Y rail lengths:

Stepcraft-2/300: 380 mm (15")

Stepcraft-2/420: 500 mm (19.7")

Stepcraft-2/600: 680 mm (26.8")

Stepcraft-2/840: 920 mm (36.2")

9. Insert the ball plungers into the holes beside the seals, as shown in [figure 2.65](#).

10. Insert the plastic hinges into the remaining four holes.



Fig. 2.65 Preparing the Y rails and the side panels of the Y axis

11. Fasten the Y rails to the rear panel using one 16 mm M5 and two 25 mm M6 bolts, making sure you match the left- and right-hand rails to the appropriate sides of the panel (the rails fit both sides if you accidentally turn them around). Make sure, too, that the rubber seals don't get jammed between the rail and the rear panel.
12. Before you finally tighten the bolts, make sure that there is an even overhang of about 1 mm on all sides of the rail (see [figure 2.66](#)).

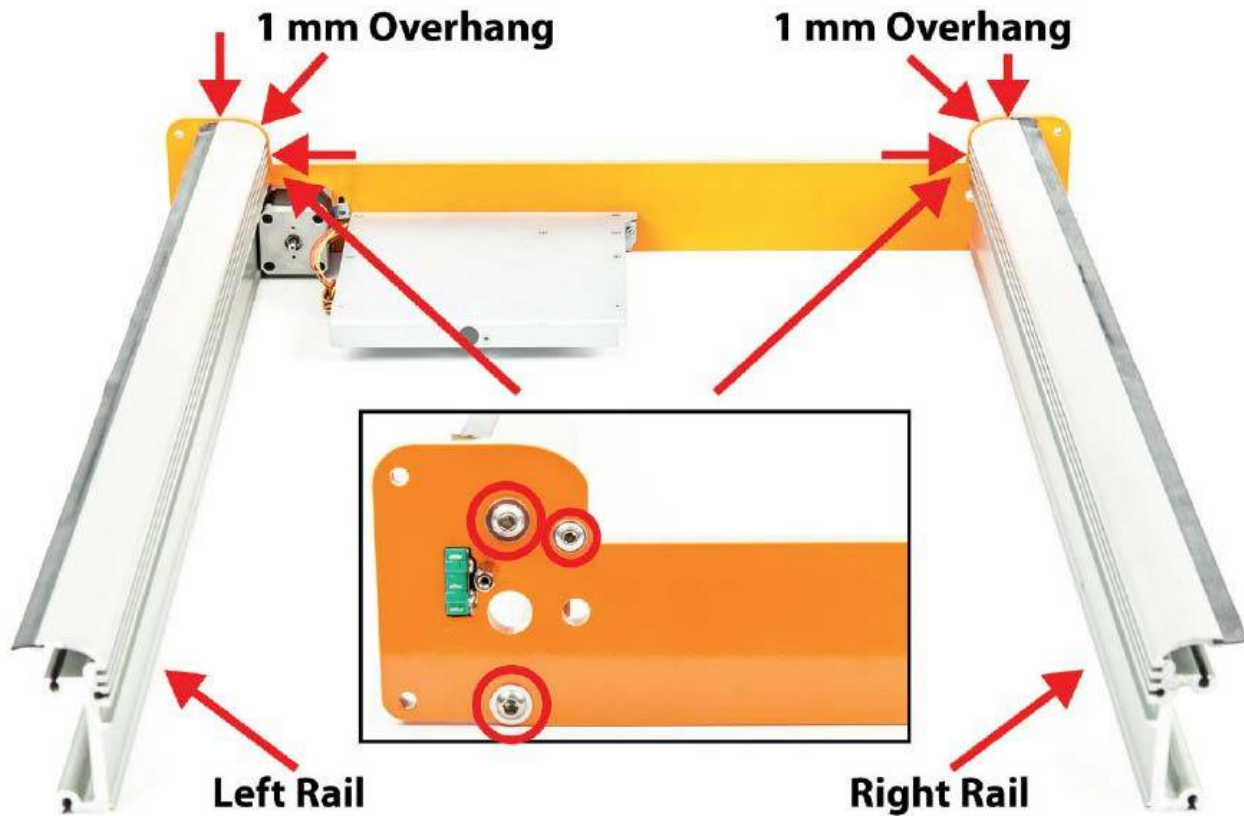


Fig. 2.66 *Mounting the Y rails*

- 13.** Use a 6 mm M3 screw to fasten the 30 mm M3 spacer to the base plate.
- 14.** Insert the rubber grommets into the holes in the base plate, making sure the metal sits in the slots in the grommets.
- 15.** Insert the base plate into the guide slots in the rails (see [figure 2.67](#)) and fasten the holder for the control unit to the base plate using a 6 mm M3 screw.



Fig. 2.67 *The mounted base plate*

16. Insert the gantry into the rails (see [figure 2.68](#)). Make sure you insert the left-hand side of the gantry (with the cables) into the left-hand rail. If you set up the rollers correctly earlier on, the gantry should be easy to mount in the rails.



Fig. 2.68 Mounting the gantry

17. Tighten the M3 bolts in the roller holders by a quarter turn. This creates the required pretension in the roller system.



Fig. 2.69 *Tensioning the rollers with a quarter turn to the left*

- 18.** Insert the ends of the Y lead screws with the M5 threads into the Y rails and through the lead screw nuts in the side panels. Rotate the lead screws by about two-thirds of their length.
- 19.** Insert a 688-ZZ bearing into the lead screw hole in the rear panel (see [figure 2.70](#)) and continue to rotate the lead screws until they reach their stop positions in the bearings.
- 20.** Thread a second 688-ZZ bearing and an M6 washer onto the end of each lead screw.
- 21.** Insert a 5 mm M3 grub screw into the each of the two remaining toothed pulleys and fasten these to the lead screws with the grub screws facing the flat side of the lead screw ends.



Fig. 2-70 Inserting the lead screw (shown here for the right-hand side panel)

- 22.** Thread the M5 lock nuts onto the lead screws and tighten them until they have no axial play but still rotate freely.

Once the lead screws are mounted, you have to align the gantry so that it is precisely perpendicular to the Y lead screws. To do this, you have to adjust the distance between the side panels and the ends of the Y rails until they are exactly the same on both sides.



Fig. 2.71 Mounting the toothed pulleys

- 23.** Rotate the lead screws until you can measure the distance between the side panel of the gantry and the end of the rail with a caliper

(see [figure 2.72](#)). Take a measurement and adjust the other side to match. If you move one side too far in relation to the other and you have to rotate the lead screw back, you will produce obvious backlash. Always rotate both lead screws from the same end when adjusting the gantry. Once things are adjusted to your satisfaction, don't move the machine. It is only once you have mounted the toothed drive belt that the system can no longer accidentally go out of alignment.



Fig. 2.72 Aligning the gantry

2.8 Step 6: Final Mechanical Assembly

This is where we finalize assembly of the mechanical parts. This involves

fitting the drive belt to the Y axis and completing the frame.

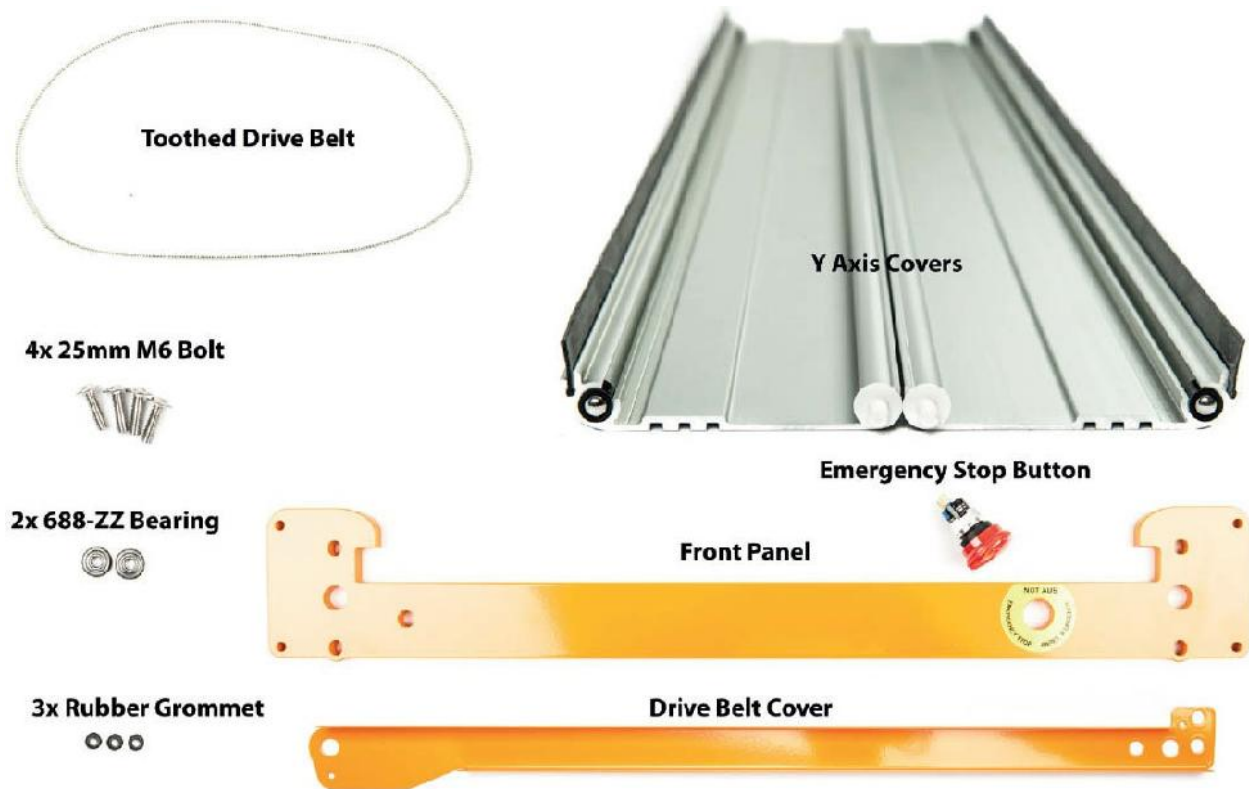


Fig. 2.73 The parts required for step 6

1. Remove the silver ring from the emergency stop button, insert the button into the hole in the front panel, and fasten it using the ring. Make sure the contacts on the switch face upward.

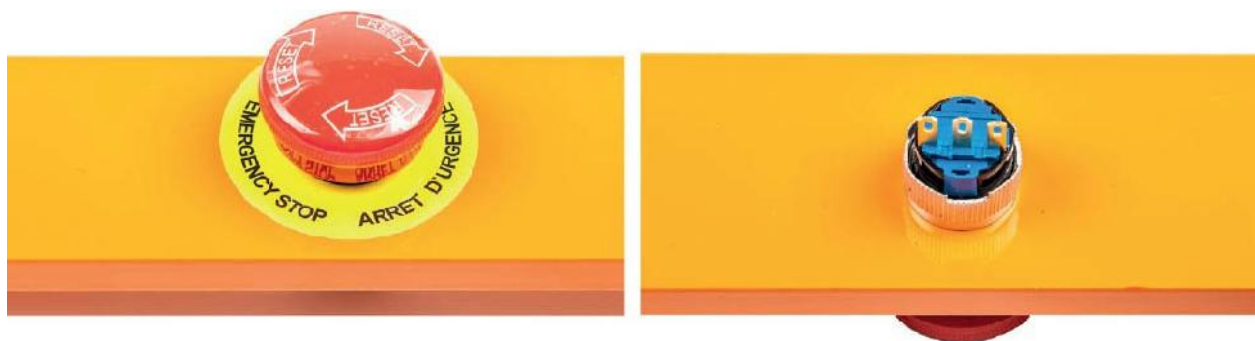


Fig. 2.74 The mounted emergency stop button

2. Loosen the screw in the belt tensioner and mount the toothed drive belt onto the three toothed pulleys and over the tensioner

(see [figure 2.75](#)). Make sure the teeth on the belt engage with the pulleys and that the belt is not twisted.



Fig. 2.75 *The correctly mounted drive belt*

Tensioning the Drive Belt

It is difficult to describe precisely how tight the tensioner needs to be, but plucking the tensioned belt should produce a deep vibrating sound. If it sounds like a tensioned guitar string, it is too tight.

3. Push the tensioner upward and tighten its bolt. You can now move the machine around, as the Y lead screws can no longer be adjusted separately.

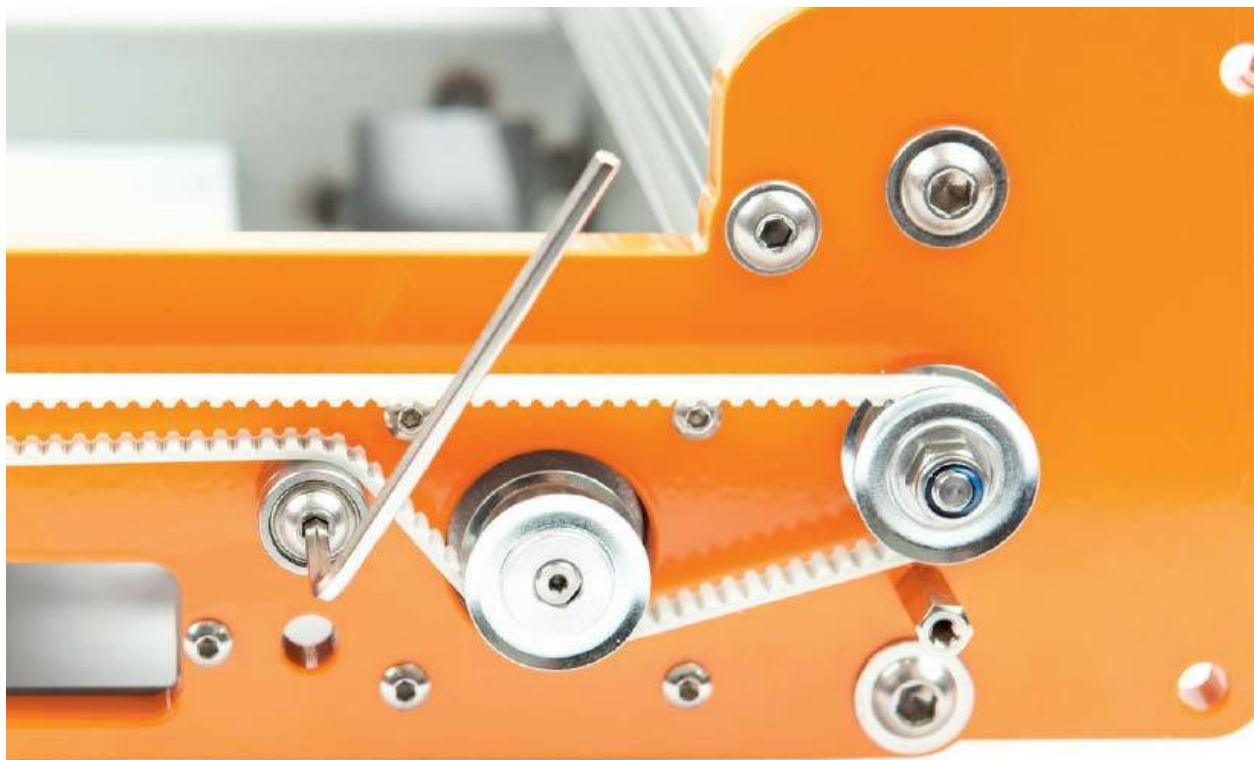


Fig. 2.76 *Adjusting the drive belt tensioner*

4. Insert the two rubber grommets into the holes in the drive belt cover (see [figure 2.77](#)) and bolt the cover into place.



Fig. 2.77 The mounted drive belt cover

5. Prepare the side covers as shown in [figure 2.78](#) and insert the pegs on the hinges into the rear panel.
6. Place a bearing on the end of each Y lead screw and mount the front panel so that the pegs on the hinges and the lead screws fit into their designated holes.

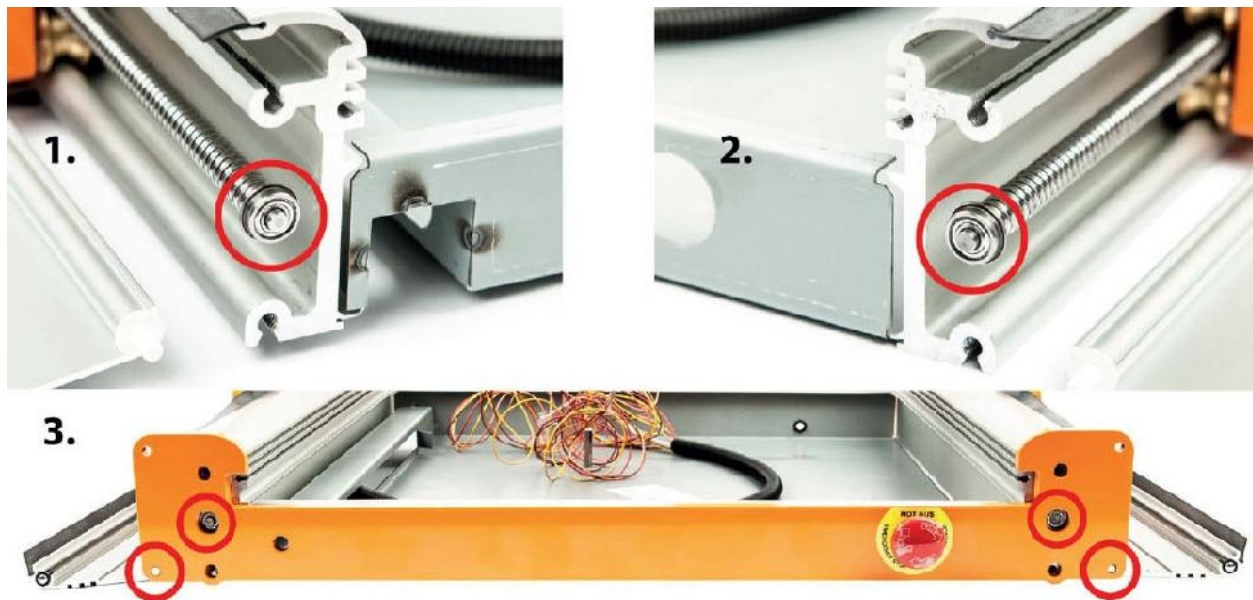


Fig. 2.78 Mounting the front panel

7. Fasten the front panel tightly in place using four 25 mm M6 bolts. Here, too, make sure that the panel has an even overlap all around its edges. Close the side covers, allowing the ball plungers to snap into place.



Fig. 2.79 Step 6 completed

Congratulations! You have now finished the mechanical assembly phase. You can now install the electronics and finish the cabling.

2.9 Step 7: Installing the Electronics

This step involves installing the control unit and laying/connecting the cables. The required steps vary slightly depending on whether your machine has a parallel or USB port. Before you begin, make sure you can correctly identify the X, Y, and Z motor cables.

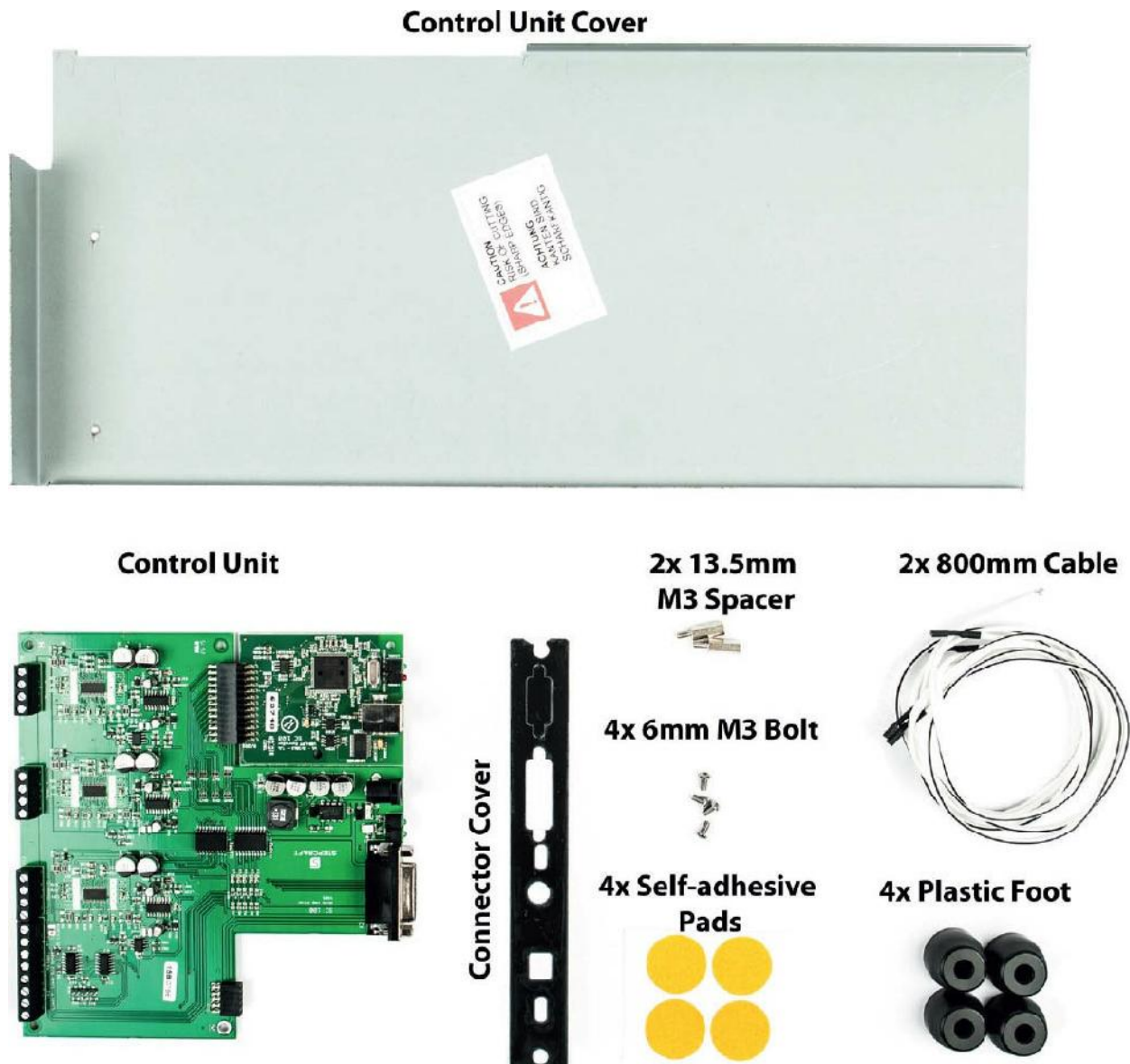


Fig. 2.80 The parts required for step 7

1. Attach one of the two-core cables to the outer contacts on the emergency stop button and feed the cable through the grommet.
2. Thread all the cables from the conduit tube and about 1 cm of the tube itself through the large hole between the base plate and the control unit holder. The tube will prevent the cables from snagging on the sharp edges of the metal parts. Lay the cable around the spacer (see [figure 2.81](#)). You will have to squeeze the tube to get it through the opening.



Fig. 2.81 Laying the cables

3. Attach the remaining two-core cable to the Y axis end switch. To do this, feed the connectors through the holes in the rear panel where you inserted the grommets earlier on (see [figure 2.82](#)).
4. Feed the end of the cable straight through the third grommet and into the inside of the machine.



Fig. 2.82 Connecting and laying the end switch cable

5. Insert the appropriate connector cover (in this case, the USB version) and mount the control unit onto the spacers on the holder (see [figure 2.83](#)). Make sure the connectors on the control unit match up with the holes in the connector cover and that the holes at both ends of the cover are lined up with the threads in the holder.

Model Differences

The connector covers are different for parallel port and USB models. Check that you use the one that matches the connectors and LEDs on your machine's control unit.

6. Fasten the control unit in place using four 6 mm M3 bolts and two spacers.

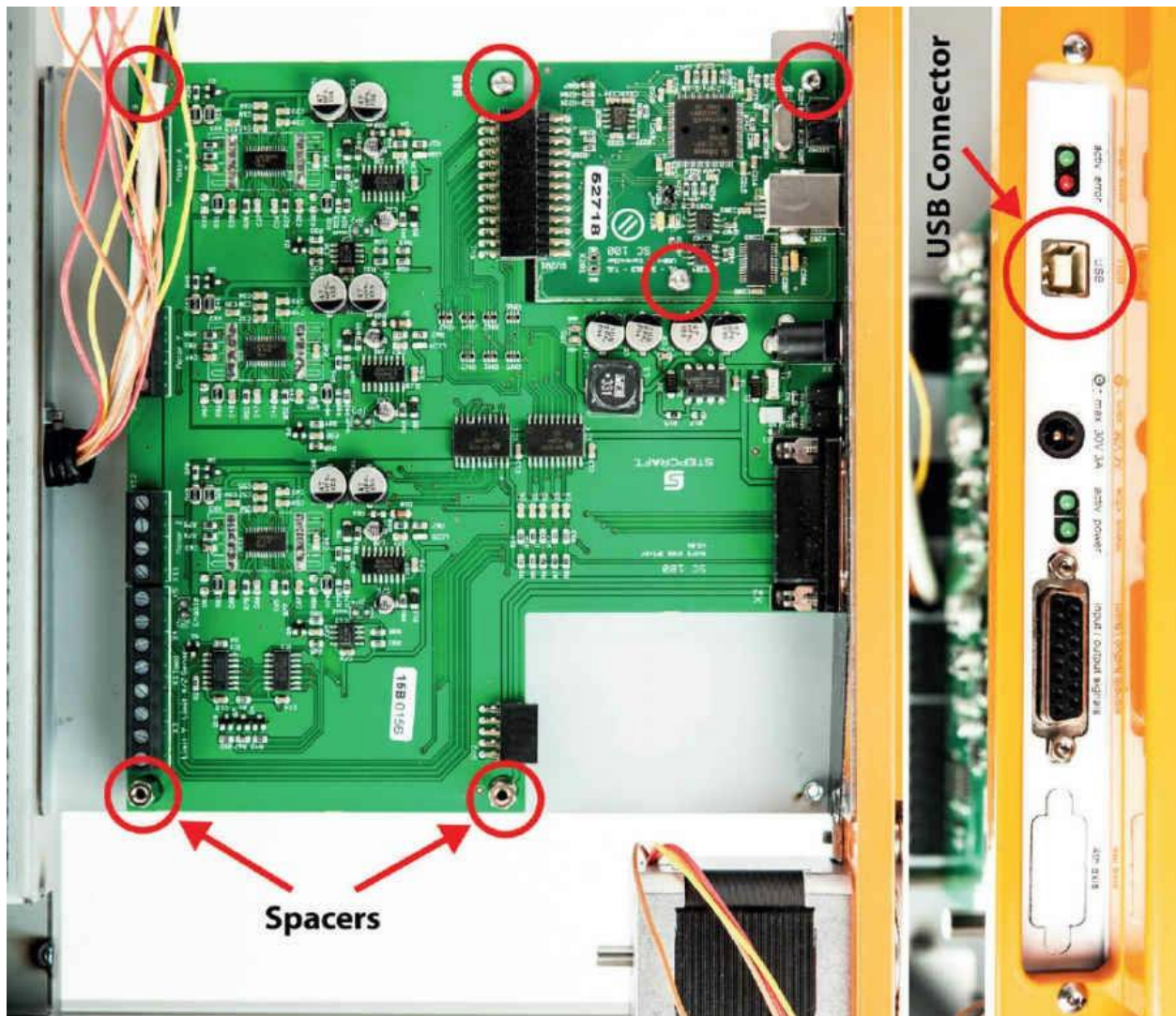


Fig. 2.83 *The mounted control unit*

Getting the Connections Right

You now have to connect the three motor cables, the two end switch cables, and the emergency stop button to the control unit. Be careful to get the order of the colored motor cables right and double-check them when you are done.

When inserting cables into the black connector block (on the left in [figure 2.83](#)), make sure that you only insert the stripped part of each wire. If you accidentally insert part of the insulation, the machine will either not work at all or will have a faulty contact.

7. Connect both end switch cables and the emergency stop cable as shown in [figure 2.84](#). The Y end switch cable is easy to identify, and the cable for the other two end switches is the gray-coated one with white and brown cores. It doesn't matter which way the two

cores are connected in all three cables. Make sure you insert each core as far as possible into the connector block and that the screws are tight. Pull gently on each core to make sure it isn't loose.

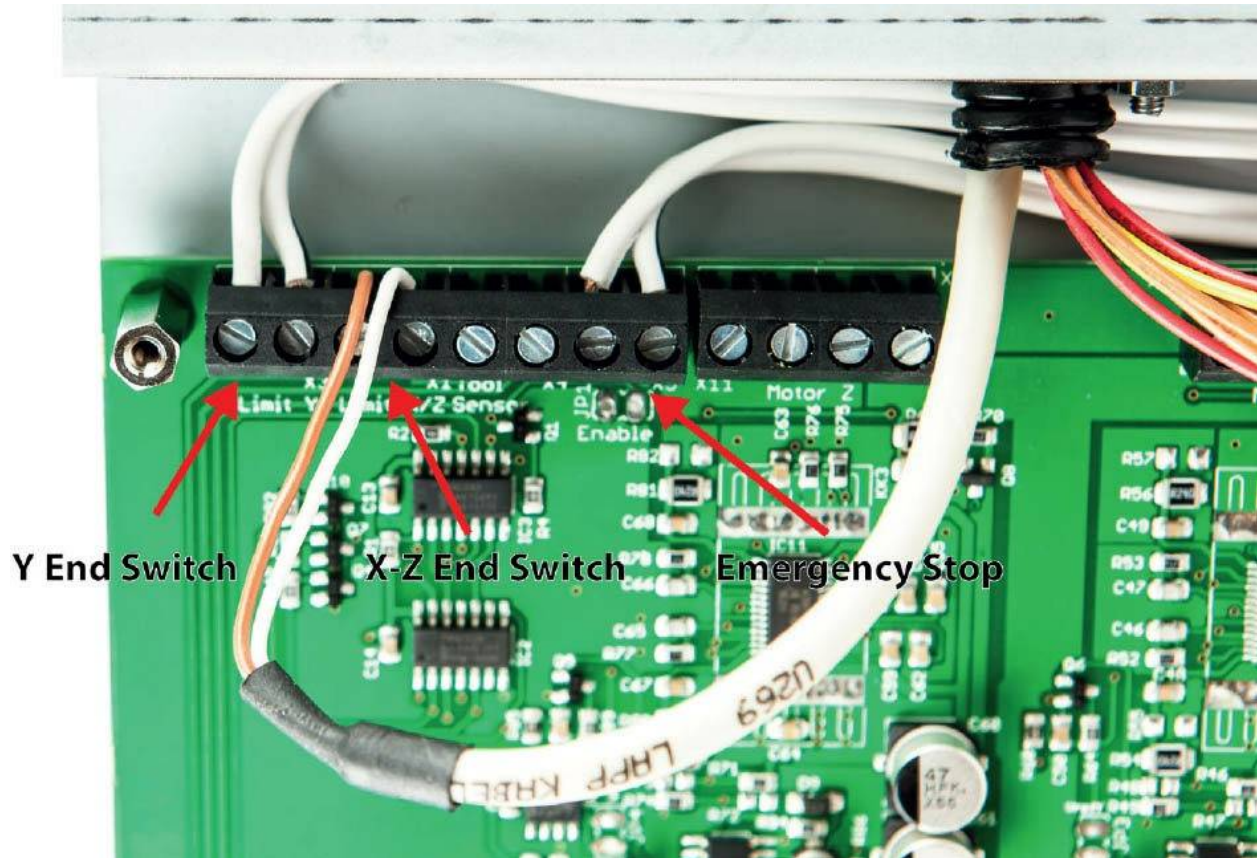


Fig. 2.84 *Connecting the emergency stop and end switches*

8. Connect the Y motor (the one directly beside the control unit). Take care to connect the four cores in the correct order for this and the other two motors. The connectors for the Y motor are labeled “Motor Y” on the control unit.

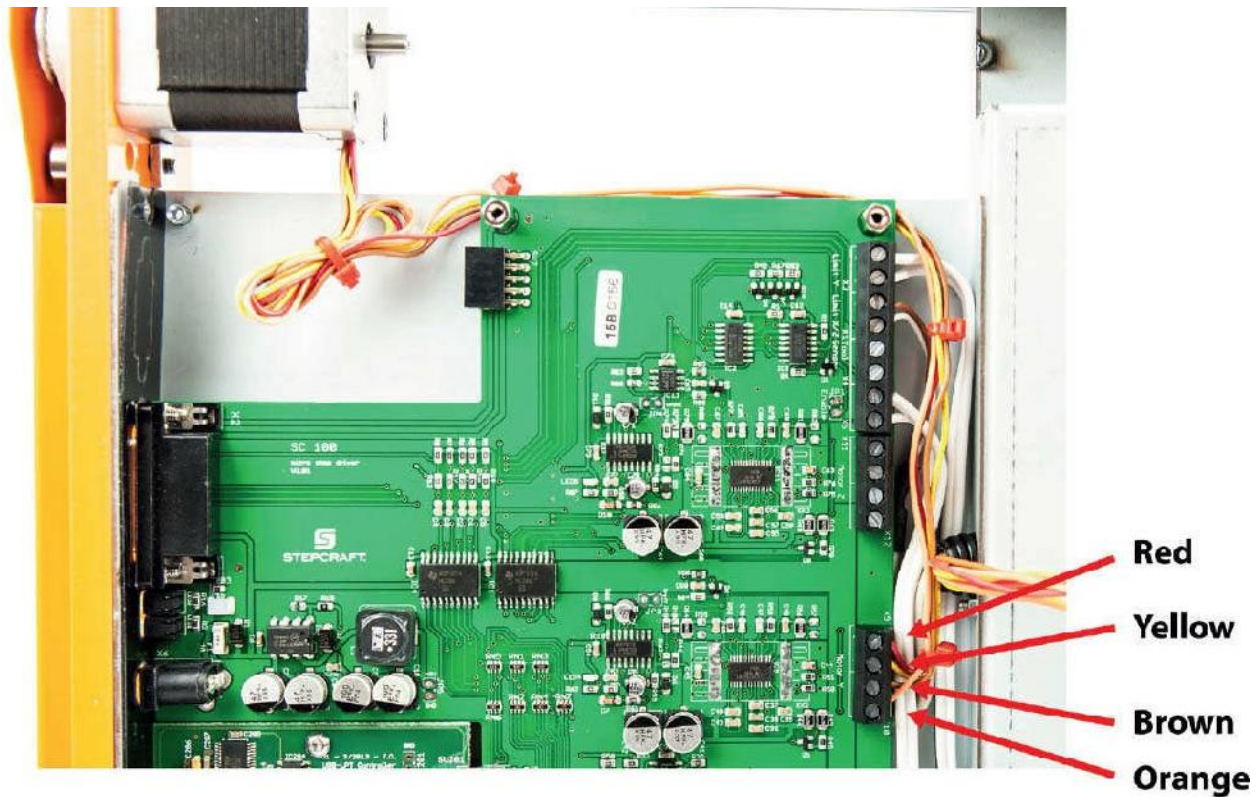


Fig. 2.85 *Connecting the Y motor*

Connect the other two motors the same way. If they are too long, shorten and solder the cable ends or roll them up and secure them with cable ties. To prevent unnecessary wear and tear or snags while the machine is in operation, we bundled all our wires using cable ties.

- 9.** Attach the cable for the Z motor. Use the same order for the colored cores and double-check your connections when you are done. The connectors for the Z motor are labeled “Motor Z” on the control unit.

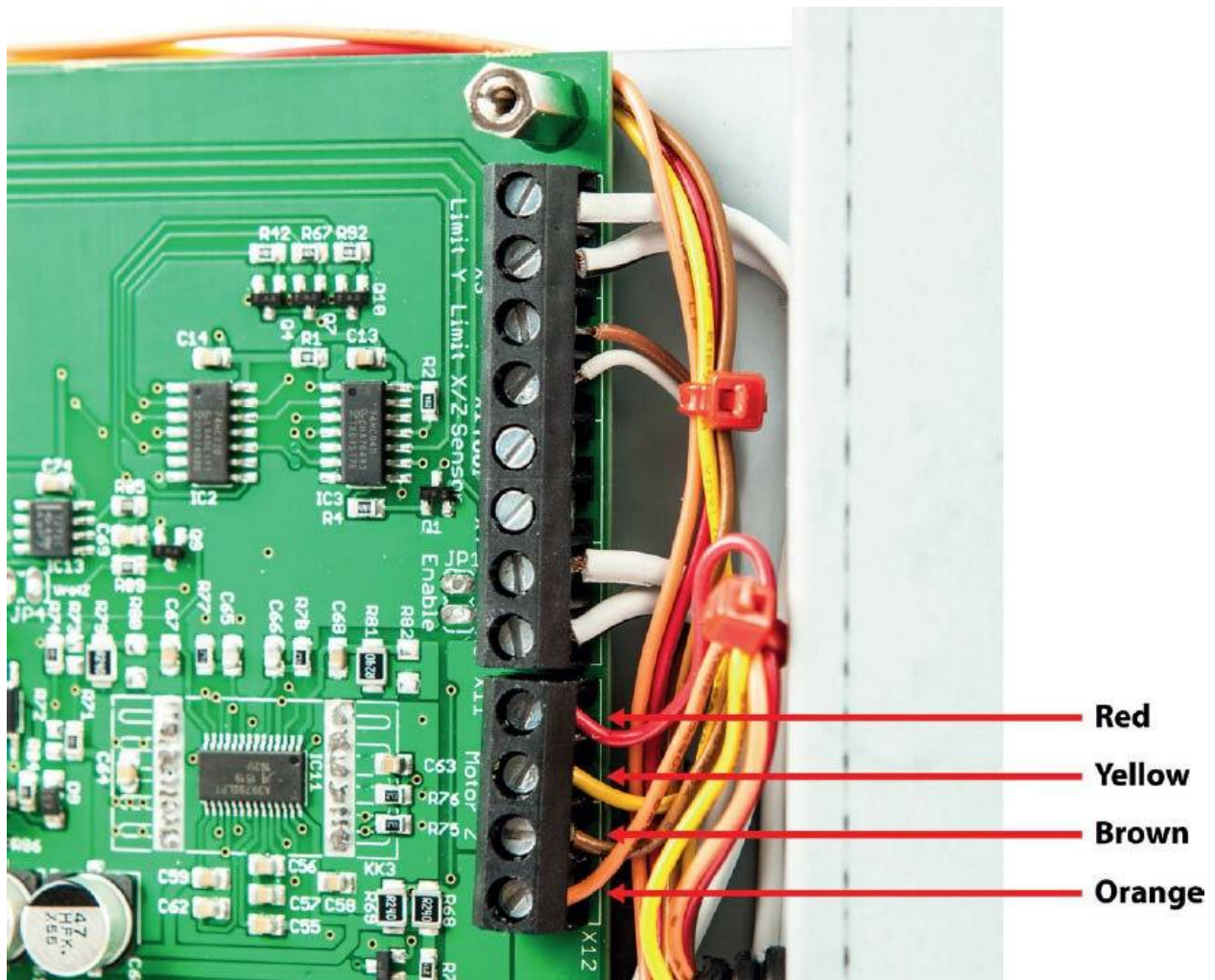


Fig. 2.86 *Connecting the Z motor*

10. Connect the X motor the same way and, once again, make sure the cores are connected in the correct order.

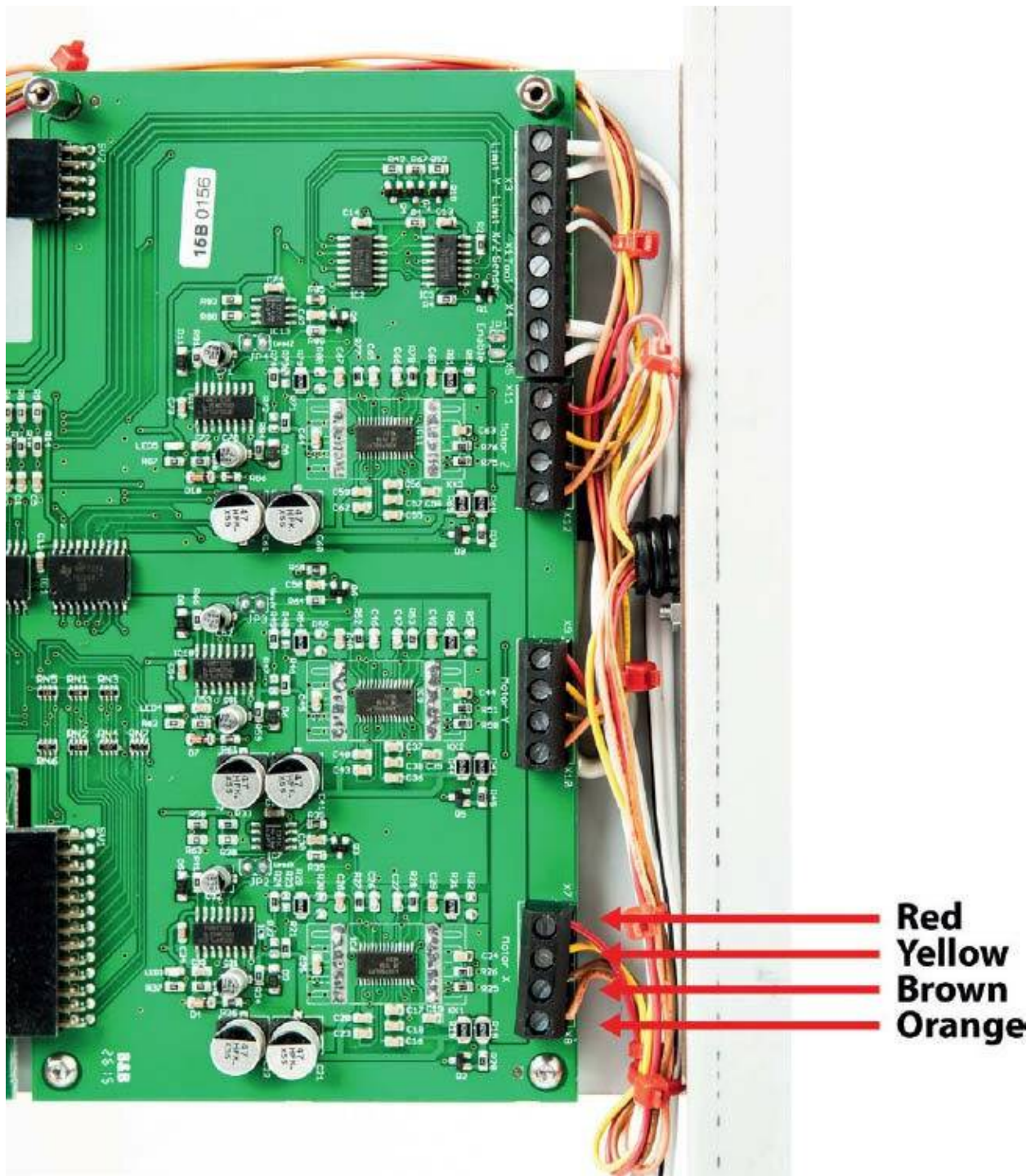


Fig. 2.87 *Connecting the X motor*

- 11.** Fit the control unit cover with its edge in the narrow slot in the rail and fasten it with two 6 mm M3 bolts.



Fig. 2.88 The fitted control unit cover

12. Connect the ground cable to one of the outside bolts on the control unit holder using a star washer.

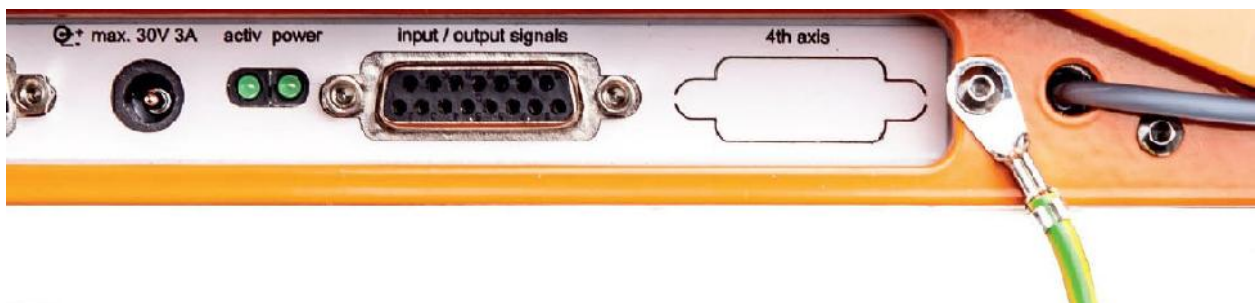


Fig. 2.89 The fitted ground cable

The machine itself is now finished. Now you have to assemble the machine table, attach the spindle, and run your machine for the first time.

2.10 Step 8: Assemble the Machine Table (5 minutes)

The standard table is a coated wooden board with two five-screw clamps that cover its full width. If you have followed all the previous steps correctly, you should be left with two 16 mm M5 bolts, an orange table clamp, the table itself, six 16 mm M6 cylinder head bolts, and two clamps.

1. Insert the machine table into the guides in the rails (see [figure 2.90](#)).
2. Fasten the table clamp in place above the front panel using the two remaining 16 mm M5 bolts.
3. Insert three cylinder head bolts into each clamp and insert the clamps into the guides. If the coating on the clamps is too thick, it will snag in the guides, so you may need to file it down a little to aid insertion.



Fig. 2.90 The finished machine

You can now add the decals and get set to run your machine.

2.11 Step 9: Installation and Startup (30 minutes)

Before you begin work on your first workpiece, you need to do the following things:

- Grease the rails and lead screws
- Install the software
- Prepare to align the lead screws

- Connect the machine
- Align the lead screws
- Start work

2.11.1 Grease the Rails and the Lead Screws

For the stepper motors in your machine to work correctly, you need to grease the rails and bearings. Use only the grease provided to lubricate:

- A total of eight running surfaces for the rollers—two each for both Y rails and two each for the X and Z rails
- A total of four lead screws—two for the Y axis and one each for the X and Z axes

For each component, apply a pea-sized blob of grease every 20 cm (8 inches) and spread it as evenly as possible using a brush.

Maintenance

After around two hours of operation, wipe off the grease you applied earlier with a cloth and regrease each component. Repeat this procedure at the latest after 20 hours of use, but sooner if your lead screws or rails get too dirty. The more you care for your machine and the less dirty it gets while you work, the longer it will last.

2.11.2 Install the Software

We will use *WinPC-NC USB* control software for our first test. Later on, we will show you how to use the alternative UCCNC package. Install WinPC-NC as follows:

Important!

Make sure the software is installed before you connect the machine to your computer.

1. Check that your computer fulfills the requirements specified by the software

2. Insert the CD into your drive and run *Setup.exe*
3. The install wizard will guide you through the setup process, and there are only a few options that require your attention. The only obligatory step is to accept the license agreement by clicking the *Yes* button. You can choose your own installation directory and change the names of the start menu and desktop items.



Fig. 2.91 The WinPC-NC licensing agreement and the window for changing the install directory

4. During the setup procedure, the program may require you to install a separate *FTDI-CDM driver*. This controls the USB machine interface. If this applies to your installation, accept the license agreement terms, click your way through the installation wizard, and complete installation by clicking the *Finish* button.
5. Once the WinPC-NC installation is complete, restart your computer.

2.11.3 Prepare to Align the Lead Screws

To align the lead screws, you have to slightly loosen the bolts holding the gantry side panels, the front panel, and the rear panel, as indicated in figure 2.92.

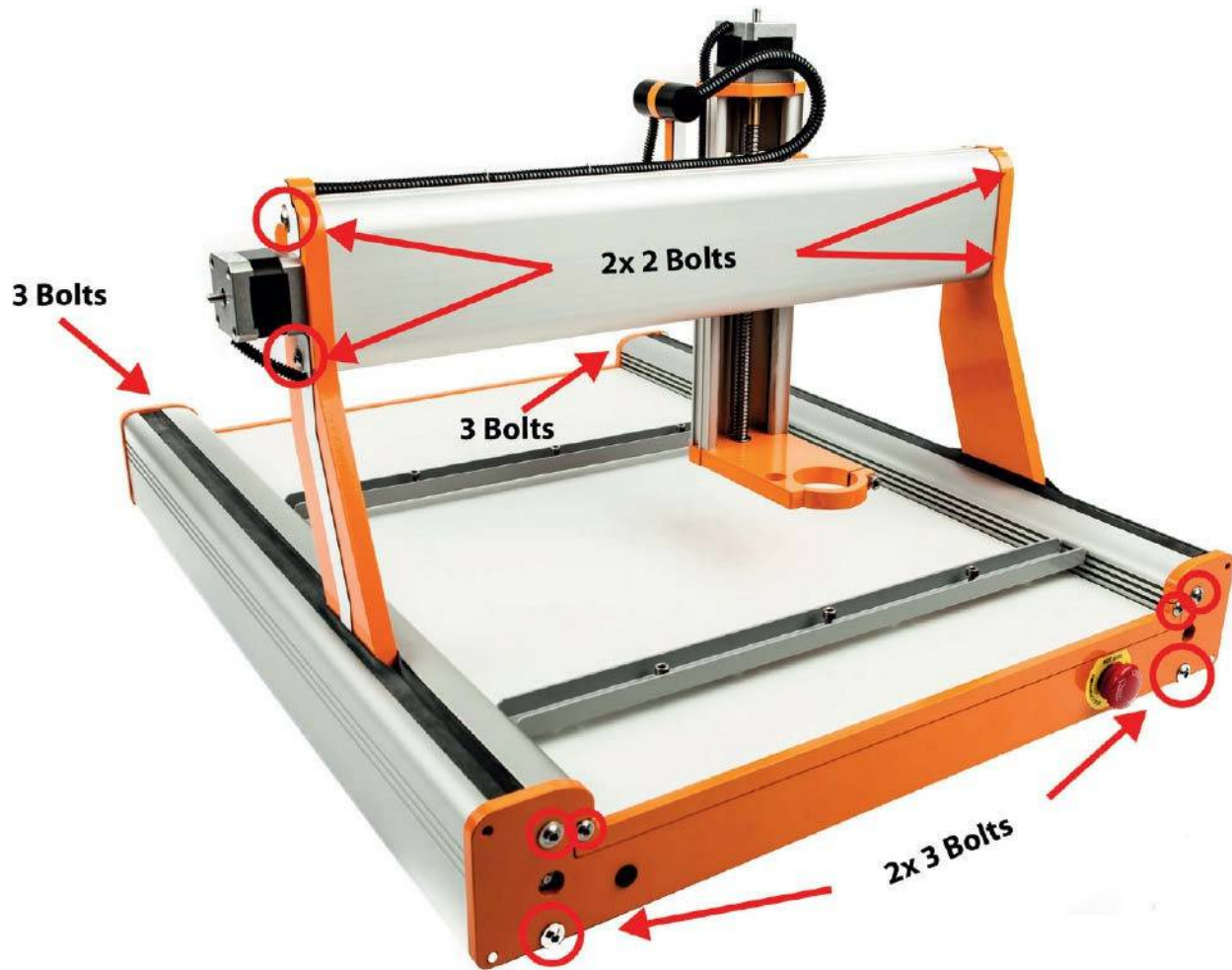


Fig. 2.92 The bolts you need to loosen to align the lead screws

2.11.4 Connect Your Machine

The following steps describe the procedure for connecting your machine via USB. If you are using a parallel port, please follow the appropriate alternative instructions.

1. Connect the power cable to the machine and an outlet. The green status and power LEDs will glow solid green. If they flash, check the table below for the appropriate diagnosis.
2. Connect the USB cable first to your machine and then to the computer. Windows will now recognize a new USB device and will automatically load the drivers you have already installed. Wait

until the “*This device is now ready to use*” message appears. The status and error LEDs should now be switched off.

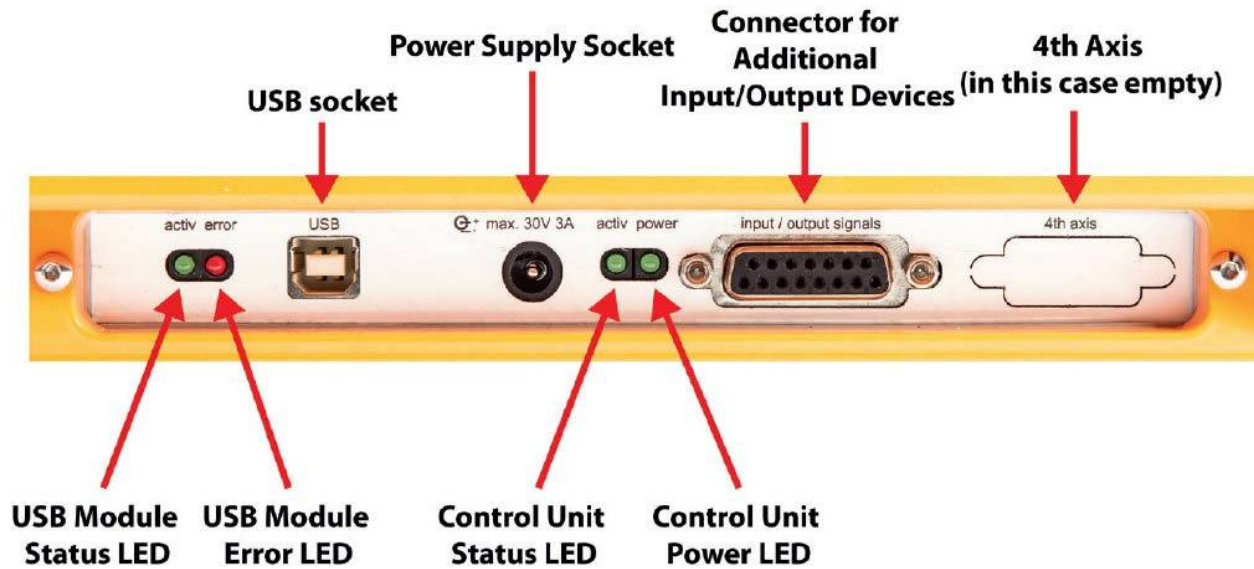


Fig. 2.93 The connectors and LEDs in a Stepcraft milling machine

3. Start WinPC-NC from the Start menu. During startup the software will ask you to select a language and your machine model. Once you have made your selections, the user interface will appear. This consists of a main window and a separate status window that displays various parameters.

USB Module		Control Unit		Diagnosis
Status	Error	Status	Power	
○	○	○	○	There is no power and WinPC-NC isn't running (or there is no USB connection)
○	○	○	●	Power is available, but the emergency stop has been activated. WinPC-NC isn't running (or there is no USB connection).
○	○	●	●	The machine is ready for use but WinPC-NC isn't running (or there is no USB connection)
●	○	●	●	The machine is ready for use and connected to a running instance of WinPC-NC

Table 2.4 The status display and the meaning of the flashing LEDs

The Emergency Stop Button

If your machine behaves in unexpected ways while you are setting it up or when it is in operation, don't hesitate to hit the emergency stop button. To reset the button, simply rotate it clockwise until it pops out.

2.11.5 Align the Lead Screws

Now is the time to align the lead screws so that they move freely through the lead screw nuts over their entire length. It is almost impossible to build a machine without some degree of skew, which causes tension in the rollers as they travel along the rails.

Keep Your Machine Level

To successfully align the lead screws, your machine has to be set up horizontally on a level floor. Make sure it sits firmly on all four feet and doesn't wobble.

If all three green LEDs are lit and *WinPC-NC USB* is running, you have full control of your machine. Thanks to the screws you loosened earlier on, the parts of the machine can settle into place while you manipulate the X and Y axes.

1. In the *Move* menu in *WinPC-NC USB*, select the *Manual movement* Command. Hit the *No* button to dismiss the *Go to reference position first* dialog.
2. The window that follows contains machine and workpiece coordinates as well as various buttons. At top-right in this window there is a set of arrow buttons that you can use to move the machine's axes. Clicking each arrow moves the associated stepper motor by one step (the motor will click to confirm this). Clicking and holding the single-arrow buttons moves an axis at 5 mm/s, while the double arrows move the axes at 30 mm/s. The axis will continue to move until you release the button.

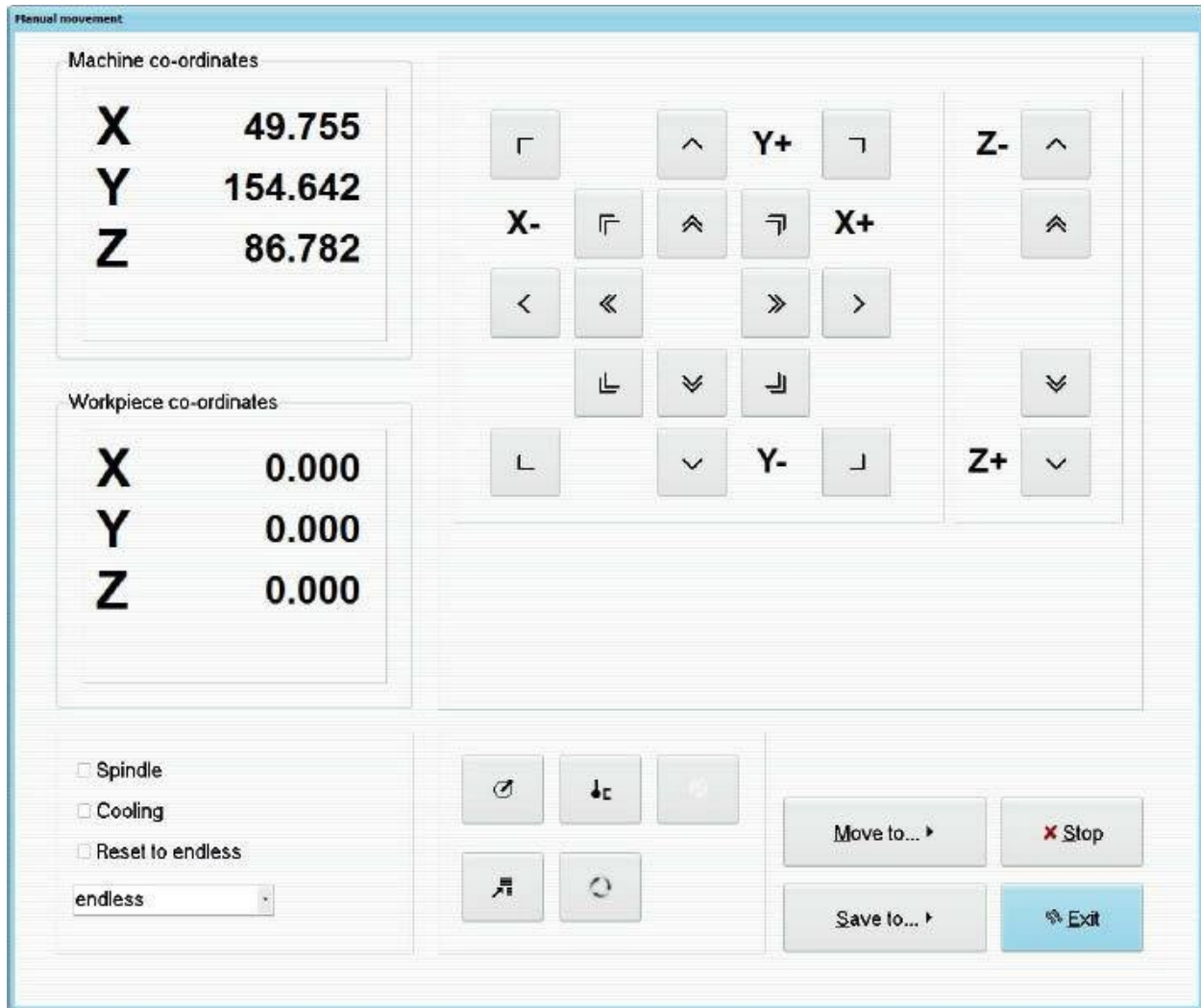


Fig. 2.94 The “Manual movement” dialog

3. Use the single-arrow buttons to check that all three axes move in both directions. If one or more axes don't work properly, you will have to troubleshoot it before you continue.
4. Move the Z axis to the middle of its vertical travel.
5. Move the X axis until it is nearly at the left-hand end of the gantry and retighten the bolts in the left-hand gantry side panel.
6. Move the X axis to the right-hand side of the gantry and retighten the corresponding bolts.

7. Move the Y axis almost completely to the rear and retighten the bolts on both sides of the rear panel. Make sure that the rear panel is fully supported and doesn't sag. If you are not sure, loosen the bolts, lift the rear panel gently at one end, move it up and down until there is the same amount of play at the top and bottom, and then retighten the upper bolt only.
8. Do the same for the other end of the rear panel and then retighten all six bolts.
9. Move the Y axis to the front and repeat the steps you have just performed for the front panel.

If an Axis Jams

If you cannot move one of the axes to the end of its travel, it is essential to troubleshoot it before you proceed. If one of the axes jams while you perform steps 5 to 8, this can be because the side, front, or rear panels exert pressure on the lead screws and the stepper motors cannot produce sufficient force to move them. If this is the case, you have to slightly raise the corresponding panel by hand before you tighten its bolts.

10. Move the axes back and forth to their end points. There should be virtually no noise from the lead screws and rollers. If you detect obvious creaks or drag, repeat steps 2.11.3 and 2.11.5 until everything is working smoothly.
11. Once everything is set up, click *Exit* to close the *Manual movement* dialog and open the *Reference* dialog in the *Move* menu. The controller will now move the X and Y axes to the left-rear position and the Z axis to its maximum height. A reference move resets the machine zero point. Don't be alarmed if the motors make a loud noise when they reach the end of their axes. This is normal and stops after a couple of seconds.

If you want to define a custom park position that the machine always uses when you stop work, use the arrows in the *Manual movement* window to select a position, click *Save to*, and select the *Park position* option.

Reference Moves

Always perform a reference move before you start work on a new workpiece. This enables the software to define a new machine zero point.

Congratulations! Your machine is now set up and ready to go.

2.12 Alternative Software

As an alternative to *WinPC-NC*, *Stepcraft* offers the UCCNC software package in conjunction with its *UC100 USB Motion Controller*. The UC100 is a kind of dongle that connects via LPT to Stepcraft's Series 2 machines and via USB to your computer. Without the dongle, UCCNC doesn't recognize the machine and cannot connect directly to a USB port like WinPC-NC.

UCCNC has a more contemporary user interface than WinPC-NC and looks more like applications such as Mach3, with its machine-style control panel. In fact, UCCNC can process Mach3 XML files for defining custom user interfaces. Stepcraft machines are supplied with predefined profiles that make setting them up a quick and easy process.

For beginners, UCCNC offers a range of functions that is very similar to that offered by WinPC-NC USB.

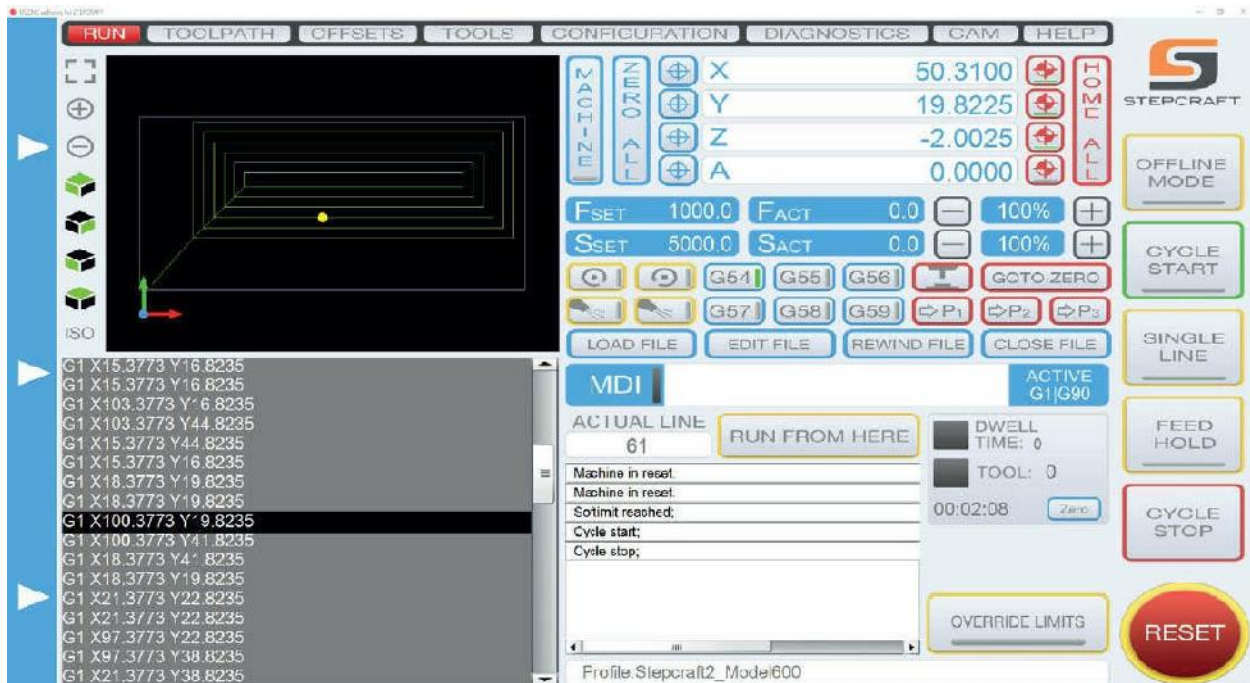


Fig. 2.95 UCCNC's user interface

2.12.1 Installing the Hardware

To connect the UC100 controller, you need an LPT-based control unit rather than the USB-based one we used with WinPC-NC. The basic control unit is the same in both cases, but the LPT version requires its own interface board, which is easy to install in just a few minutes.

If you installed the USB interface board as described in [chapter 2](#), you have to swap it out. You also need to swap the connector cover for the one with an opening for the LPT socket.

Caution!

Always wear an antistatic wrist band when you are working on the electronics in your machine. If you don't, you risk trashing the control unit.

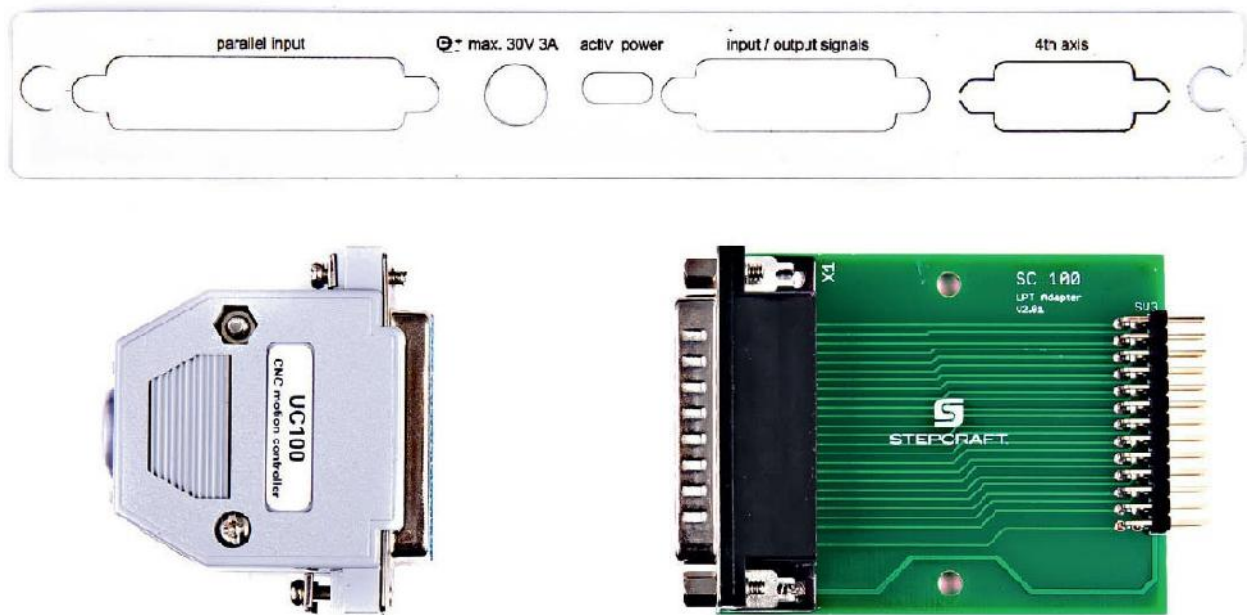


Fig. 2.96 The LPT connector and the UC100 dongle

1. If you have not yet assembled your machine, go directly to step 5.
2. Disconnect all power cables and remove the spindle and—if you have one—the exhaust adapter.
3. Place the machine on its right-hand side and remove the control unit cover.
4. Undo the four bolts holding the control unit and the two holding the USB interface board. Move the control unit to the right and twist it slightly to remove the connector cover and the USB board.

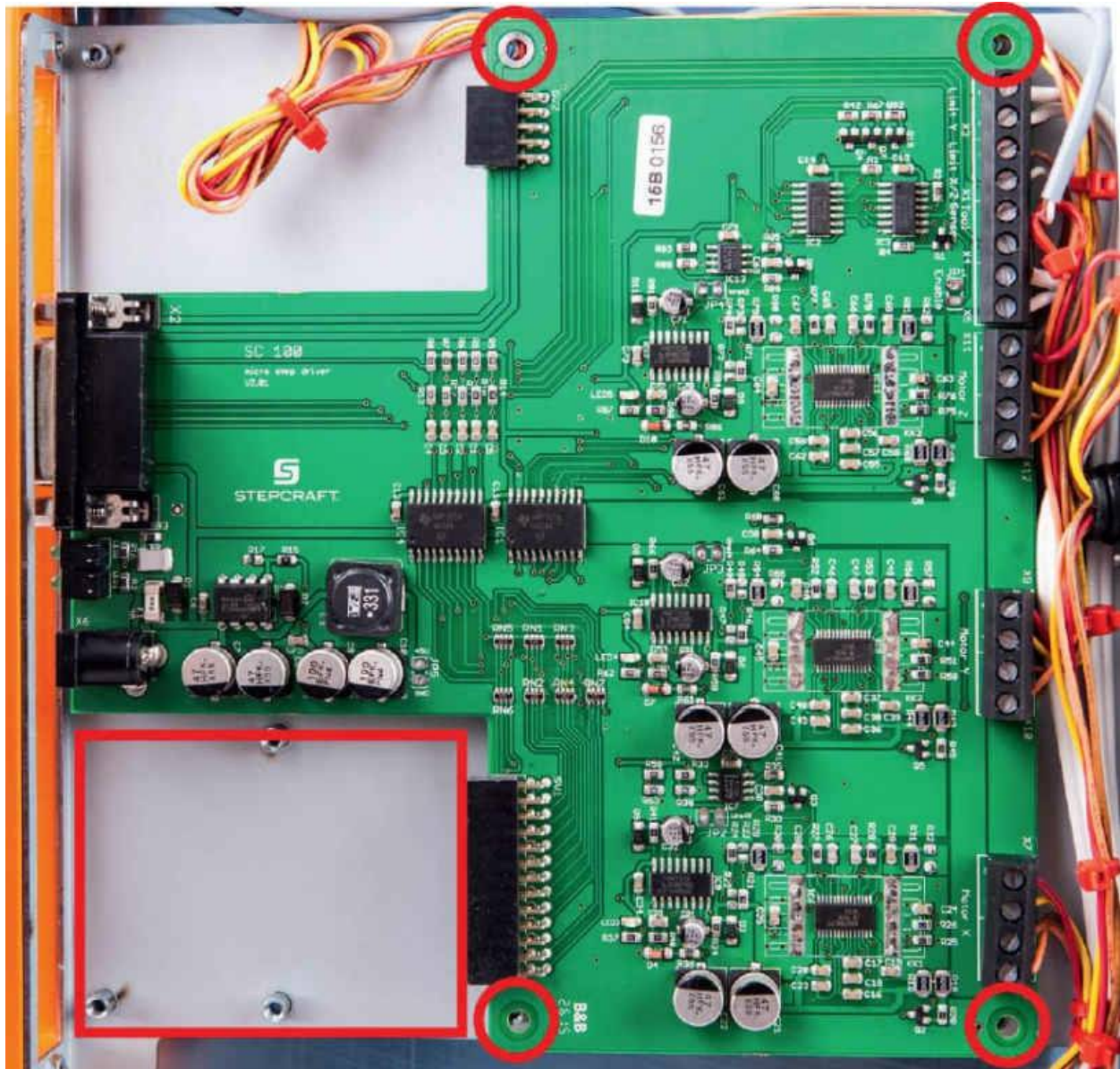


Fig. 2.97 *The slot for the LPT connector and the bolts holding the control unit*

5. The red outlines in [figure 2.97](#) show the slot for the LPT interface board and the four bolts holding the control unit. To install the board, insert the socket end through the opening in the machine's rear panel, and then push the pins all the way into the connector in the control unit.
6. Move the control unit slightly to the right and insert the connector cover.

7. Insert and tighten the four bolts for the control unit and the two for the LPT interface board.

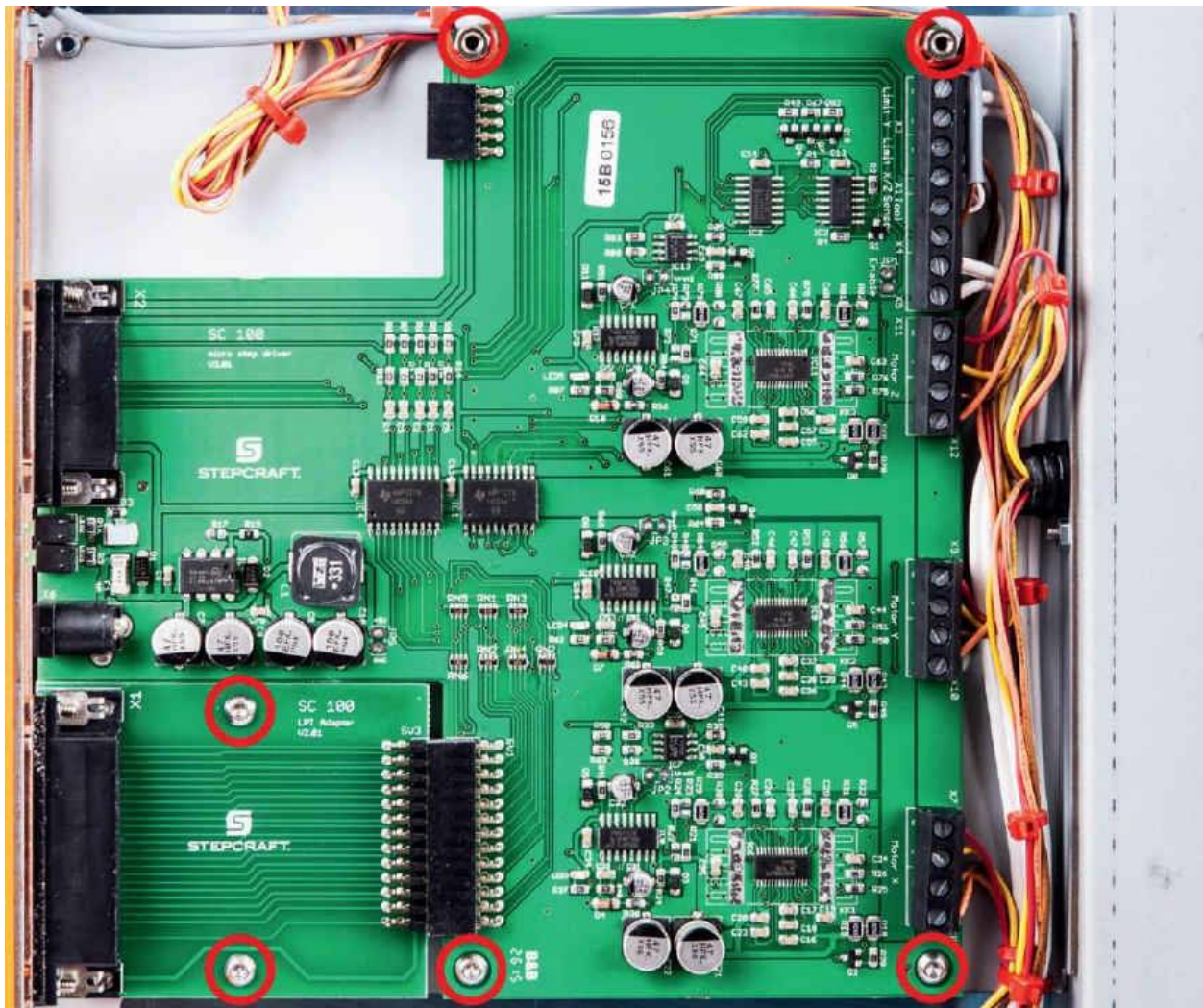


Fig. 2.98 The installed control unit and LPT interface board

8. Replace the control unit cover and tighten the bolts.
9. Insert the UC100 controller into the LPT socket but *don't* yet connect it to your computer.

2.12.2 Installing and Configuring UCCNC

The UCCNC package comprises a CD and a license key that is required to run the software with your router.

1. Insert the CD into your drive and run *Setup.exe*. Click *Next*.
2. If you agree with the licensing terms, check the option *I accept the agreement* and click *Next*.
3. Select the directory where you want to install the application and click *Next*.
4. Check the option *Overwrite the default UCCNC profile* (this erases previous installations) and click *Next*.
5. If required, change the name of the start menu item and click *Next* to get an overview of the settings you have made. If you are happy with your settings, click *Install*.
6. The installation process will request you to connect the UC100 controller to the USB port on your computer. Do this and click *OK* to proceed.
7. Once the installation has completed, you need to install a profile for your particular Stepcraft machine. To do this, navigate to the *Stepcraft* directory on the application CD and run *Screensetinst.exe*. You can then select your machine in the dialog that follows. If you aren't planning to do any 3D printing, uncheck the option, and then select *Start the installation* to install the profile.

Installing the Stepcraft profile generates a custom desktop icon, which you can double-click to start the application. Note that your profile will only be loaded if you start the application via the new icon, but not if you use the standard one. In our example, the new icon is labeled *Stepcraft2_Model600*.

Before you begin, you have to copy the license key file to your UCCNC install directory. The directory will already contain a file called *UCCNClicense.txt* (this is a demo license), which you have to overwrite with your own license key file.

Starting the application via the new icon will open a window like the one shown in [figure 2.99](#). If the license key and the profile have been correctly installed, there will be no flash screen warning that you are working in demo mode.

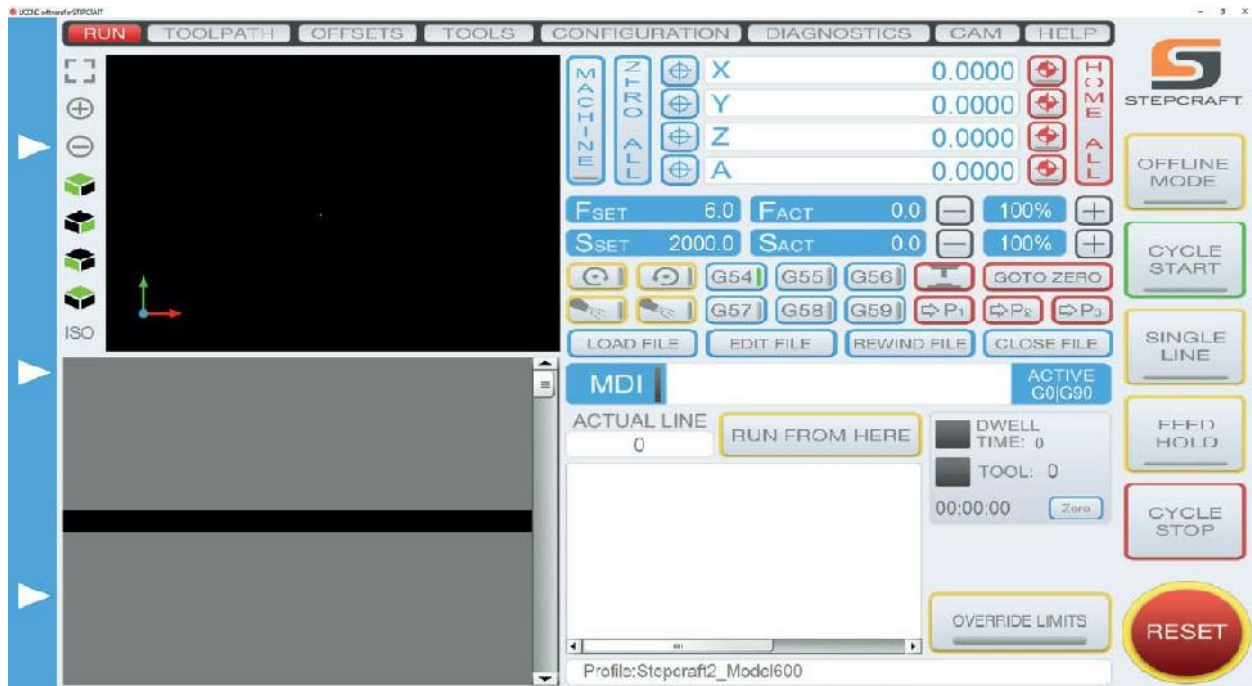


Fig. 2.99 The UCCNC application window running a Stepcraft profile

The huge number of buttons and displays can be quite daunting for beginners, but with the help of the manual and a little practice, you will soon find your way around. When you start the application, the *Reset* button will blink. *Don't click it!*

Begin by checking the status LEDs on the router and the UC100 controller. The LEDs on the router are located to the right of the power cable and have the following meanings:

Status	Power	What It Means
○	○	There is no power and the machine is not ready for operation
○	●	Power is connected, but the emergency stop is activated
●	●	The machine is powered up and ready for operation

Table 2.5 The meaning of the router's status LEDs

The LEDs at the rear of the UC100 controller have the following meanings:



Fig. 7.6 These LEDs show that the UC100 is ready for operation

Green	Blue	What It Means
○	○	There is no USB connection and no connection to the UCCNC application
●	○	USB is connected, but there is no connection to UCCNC
●	●	USB is connected and UCCNC is connected to the UC100, which is ready for operation

Table 2.6 The meaning of the UC100's status LEDs

To begin a job, you must be able to see three green LEDs and one blue one. If this is the case, check whether you can control the machine manually. The manual control buttons are hidden behind the three white arrows on the left-hand side of the application window. Mouse over them to reveal the controls shown in [figure 2.100](#).

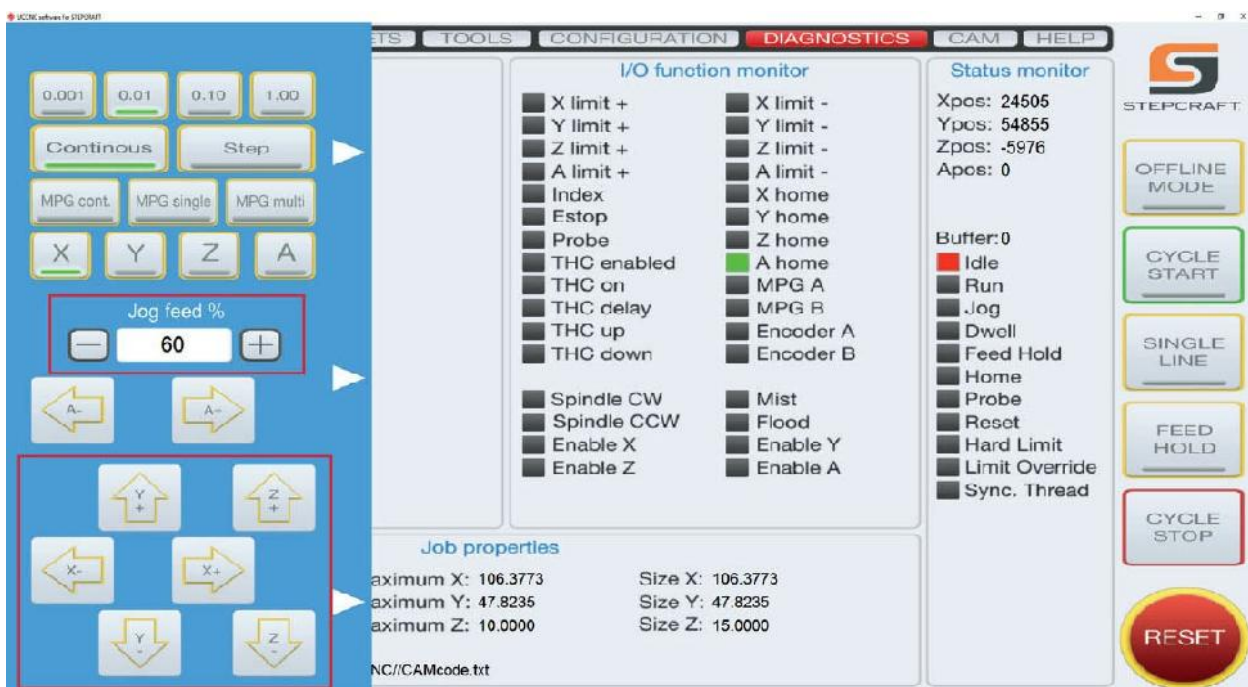


Fig. 2.100 The manual control window

The six buttons at the bottom control movement along the machine's three axes. Check that the Y- button moves the tool forward, the X+

button moves it to the right, and the Z- button moves it downward, and that the other three buttons have the opposite effects. The *Jog feed* setting determines the speed at which the tool moves in each direction. The Stepcraft profile's default setting is 3000 mm/min and can be altered in the *CONFIGURATION* → *Axis Setup* dialog. An adjustment of 10% equates to 300 mm/min.

Alternatively, you can control the X and Y axes using the arrow keys and the Z axis using the PgUp and PgDn keys on your keyboard. If you keep the Shift key pressed while doing this, the tool will move at the current maximum jog feed rate. The *Continuous* button moves the tool in the selected direction until you release the button, while the *Step* button moves it precisely one step according to the value selected in the top four buttons (between 0.001 mm and 1.00 mm).

The *RUN* window displays how far the tool will travel in each direction. Check that these values are correct by manually measuring the distance the tool moves and comparing your measured value with the one in the display.

If everything is set up correctly, you can run a test job.

2.12.3 Running a Test Job

If you have already worked with WinPC-NC, the following steps will seem quite familiar:

1. Mount your chosen tool and clamp the workpiece securely to the machine table. We used three stacked sheets of 10 mm MDF to prevent the mill from hitting the T-slot table if anything went wrong.
2. Perform a reference movement (*RUN* » *HOME ALL*). The router will move the tool to the left-hand rear corner and raise the spindle.
3. Load the DXF or G-code file for your workpiece. If you use a DXF file, you first have to generate G-code using the CAM

dialog. The G-code contains all the necessary tool, feed rate, and cut-per-pass data.

4. Set the workpiece zero point from which the job is to start.
5. Start the spindle, check its speed setting, and start the job..

Steps 3 and 4 are explained in detail in [sections 2.12.3.1](#) and 12.2.3.2.

G-code Variations

UCCNC uses a different coordinate system to the one used by WinPC-NC. Make sure the basic units and the feed rate units are the same in both applications.

2.12.3.1 The CAM Process

Our example cuts a rectangular 20×30 mm pocket that is 2 mm deep in an MDF blank using a 1 mm 2-flute fish-tail end mill. We drew the rectangle using a CAD application and saved it as a DXF file, which we loaded into the UCCNC CAM window, as shown in [figure 2.101](#).

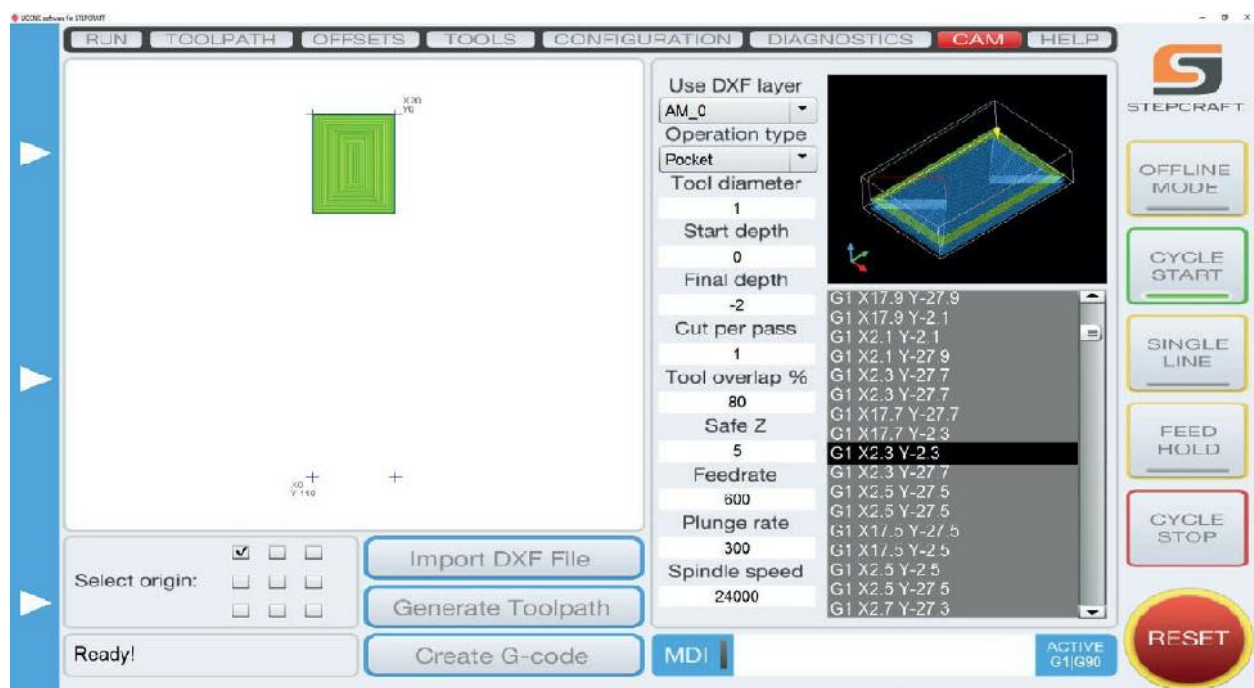


Fig. 2.101 The CAM process for our sample rectangular pocket

1. Check the top left-hand *Select origin* box to set the start point.
2. Our rectangle is stored on Layer AM_0. If your model includes components on multiple layers, you will have to mill each one separately, one after another.
3. Set *Operation type* to *Pocket*.
4. Set *Tool diameter* to 1 (mm).
5. In case of issues with the horizontal feed, we milled the top layer without actually engaging with the workpiece (using a *Start depth* of zero). To make doubly sure you don't damage anything, you can run a test pass at any distance above the workpiece.
6. Because the application uses an inverted Z axis, set *Final depth* to -2 (mm).
7. Set *Cut per pass* to 1 (mm).
8. Set *Tool overlap* to 80%.
9. Set *Safe Z* (the safety margin for the raised tool) to 10 (mm).
10. Set *Feed rate* to 600 (mm/min). This equates to 10 mm/s and is a moderate setting for MDF.
11. Set *Plunge rate* to 300 (m/min). This equates to 5 mm/s.
12. The spindle speed is irrelevant, as this is set directly on the spindle. Select Kress setting #4 (21,000 rpm).
13. The software now has all the data it needs, so you can go ahead and click the *Generate Toolpath* button. The rectangle will now be filled in, as shown in figure 7.8. The generated toolpath takes the form of a spiral.

14. Click *Create G-code* to generate and display the appropriate G-code.

2.12.3.2 The Milling Process

All you now have to do before you actually start milling is set the workpiece zero point.

1. Move the tool manually to the approximate X and Y positions, and lower it to a few millimeters above the surface of the workpiece.
2. Set *Jog feed* to 1 (%). You can enter the value directly in the box in the application window.
3. Correct your X and Y positions if necessary. Place a sheet of paper between the workpiece and the tool, then continue to lower the spindle while moving the paper back and forth until the tool tip just begins to touch the paper.
4. Remove the paper and set the Z value in the *RUN* dialog to 0.8 mm below the current position. This positions the tool fairly precisely on the workpiece surface.
5. To activate the zero point, click the *ZERO ALL* button. This sets the X,Y, and Z values to zero and saves the current position as the zero point.

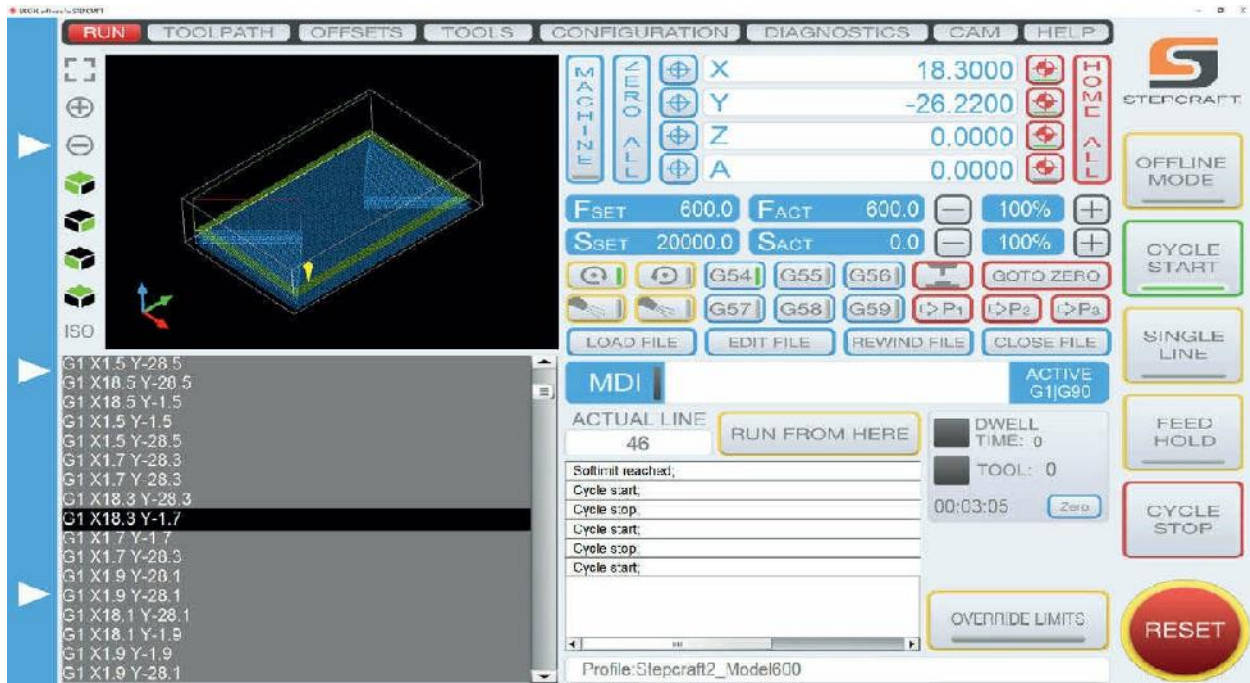


Fig. 2.102 The RUN window display while the job is running

6. Before you start the job, select the ISO view. This displays a real-time 3D view of the passes the machine makes, giving you a great overview of the current position and the path the tool will follow next. The screen also displays the G-code commands that correspond to the current cut.
7. Select speed setting #4 and switch on the spindle.
8. To start the job, click *CYCLE START*. If anything unexpected happens, either click *CYCLE STOP* or hit the emergency stop button on the machine and switch off the spindle. We encountered no issues during our sample run and the workpiece was finished cleanly.



Fig. 2.103 *More expensive machines can't do much better than this!*

The result is excellent and minimal discrepancies of one- or two-hundredths of a millimeter emphasize that everything went perfectly according to plan.

WinPC-NC or UCCNC?

Personally, I prefer the up-to-date interface design, the clear application structure, and the real-time views offered by UCCNC to the WinPC-NC approach. The frame-filling display on my 27" monitor also makes it simple to make settings at the machine and still have a clear view of the application interface 12 feet away. WinPC-NC's smaller window dimensions make working this way virtually impossible. UCCNC's CAM functionality is better organized and more controllable than WinPC-NC's too. However, both applications provide a similar range of functions, so which you select is really a matter of personal taste. At the time of writing, WinPC-NC was due for a major overhaul, so it will be interesting to see how the two applications compare in future.

2.13 Your First Workpiece

The following table defines the most important terms that you need to understand to get started with CNC milling:

Term	What It Means
Plunge Depth	Given in millimeters (mm), this is the depth to which the tool enters the workpiece at a predefined speed. Equivalent to the depth of a hole when drilling.
Spoilboard	Also known as a waste board or sacrificial board, the spoilboard is made of cheap material and is positioned beneath the workpiece to protect the machine table from damage while you work. Note that the quality of your spoilboard will affect the quality of your results. If it is not perfectly flat, the workpiece will be on a slant. If your workpiece requires a high degree of precision, you may have to mill your spoilboard flat before you begin.
Feed Rate	The feed-rate value defines the distance the tool covers per unit time. Horizontal and vertical feed values usually differ.
Incremental Depth	If a tool cannot cut to the required depth in a single pass, you have to repeat the movement to make the finished cut. Incremental depth defines the depth to which the tool is plunged for each partial cut.
Repetition	As an example, if you need to cut a slot that is 10mm deep but your tool can only cut 1mm at a time, you will have to perform an initial pass and nine repetitions at an incremental depth of 1mm.

Table 2.7 Some important CNC terms

The following example explains how to cut the text “cncbuch.de” into a rectangular plywood board. This exercise requires the following things:

- A 5 mm plywood board that measures at least 200 × 200 mm (8" × 8")
- An 5–10 mm MDF spoilboard that is about the same size as the plywood workpiece
- A 1 mm spiral end mill or other cutter suitable for working with wood
- A spindle with a suitable tool chuck or collet
- The *sample-plot-file.zip* file, which you can download from the book website

Connecting the Spindle

Some spindles have open wiring that can rub against the machine’s frame or snag the

workpiece. Make sure the cable is laid so that this cannot happen. If you tape or tie the cable, make sure it has enough play to allow the spindle to perform maximum movements in all directions. If you add an enclosure later on, you can tie the cables to one of its struts to keep them out of the way.

Begin by mounting the spoilboard and the plywood workpiece on the machine table. Make sure they are fairly straight and clamp them tightly in place.

Insert the end mill into the chuck until it protrudes by about 2 cm and tighten the chuck. Lock the spindle shaft so it doesn't move while you do this. Now mount the spindle in the machine's holder and tighten the retaining bolts.

If the spindle doesn't fit easily into the holder, file the orange coating until it fits snugly the spindle fits snugly. Make sure the spindle is positioned so that you have easy access to the on/off switch, the shaft lock, and the speed selector.

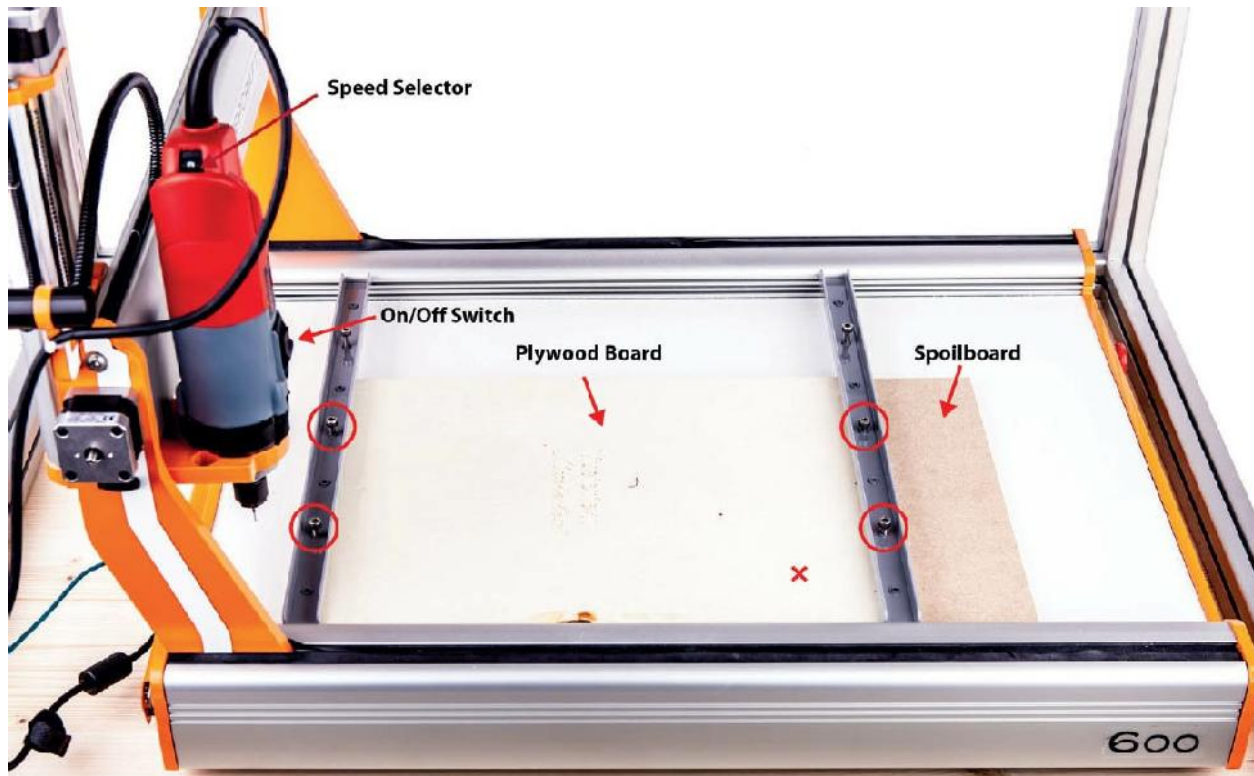


Fig. 2.104 The mounted spindle and the machine table with the spoilboard and workpiece clamped in place

1. Connect the machine to a power outlet and connect your computer.
2. Run *WinPC-NC* and make sure the machine is correctly connected (all three LEDs are green).
3. Select the *Reference* command in the *Move* menu. The spindle will move to the zero point as shown in [figure 2.104](#).
4. Use the *File* → *Open* command to open the .plt file you extracted from the *sample-plot-file.zip* archive. The text will be displayed in the program window, as shown in [figure 2.105](#).

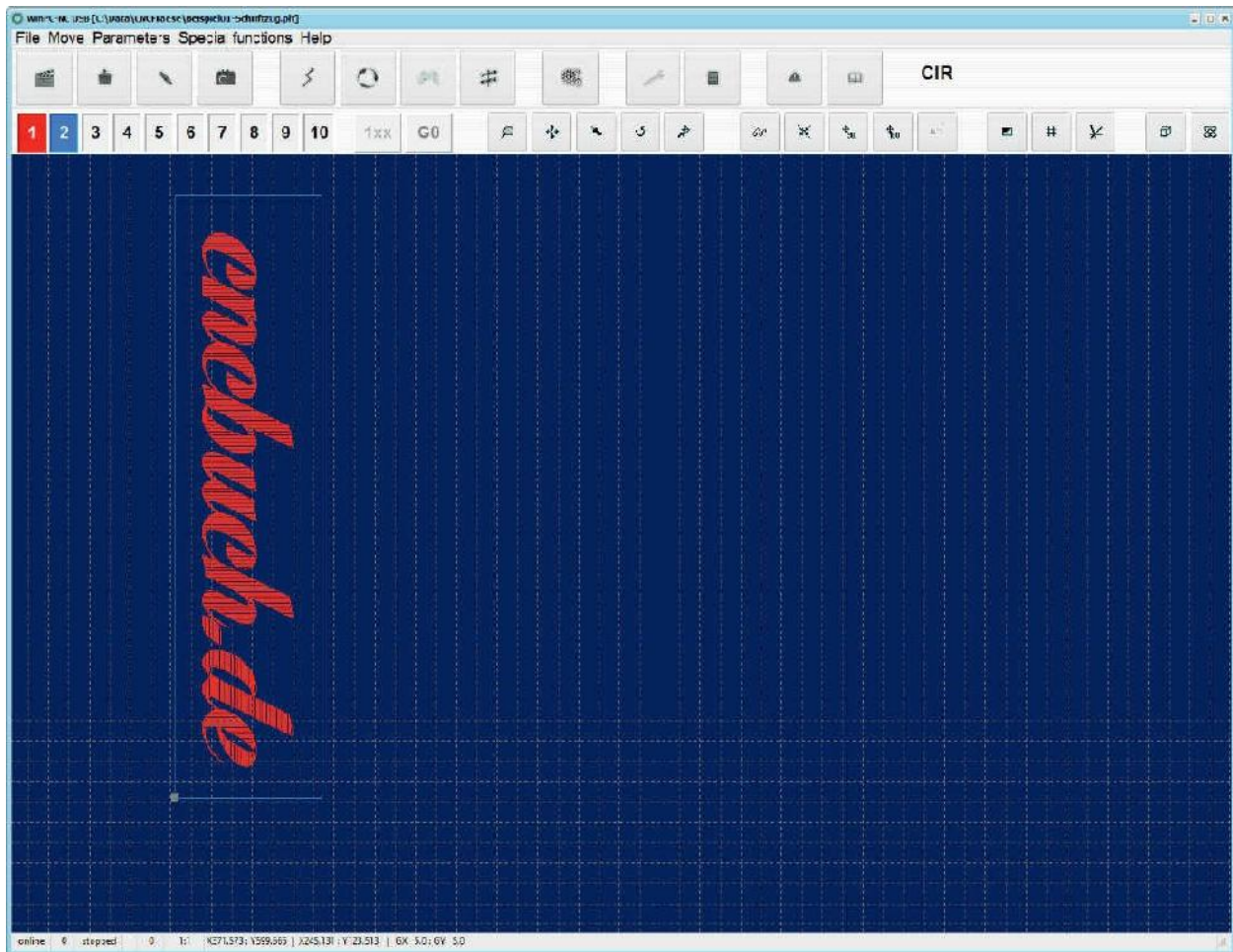


Fig. 2.105 The opened sample file (the values in your own control window may vary)

In this example, the CAM process is controlled by WinPC-NC's built-in

functionality. In [figure 2.105](#), the text is red, has a blue frame, and the numbers 1 and 2 (above the main window) are colored to match. In WinPC-NC, this means that the planned cut requires two separate tools. The text itself will be cut to a depth of 2 mm, while the frame will be cut completely free from the rest of the workpiece. This means cutting to a depth of 5.5 mm to cut all the way through the 5 mm blank.

The plywood is soft and can be cut to a depth of 2–3 mm in a single pass. The frame surrounding the text will be cut once at a depth of 3 mm, then again with a cut-per-pass increment of 2.5 mm. The plunge depth indicates how far the cutter will enter the raw material measured from machine zero, while the increment defines the depth of the cut that follows. In this case, it is important to cut the text first, before cutting out the frame—cutting the frame first would make it impossible to hold the workpiece still while the text is cut.

5. Navigate to the *Tools* tab in the *Parameters* window and click the *Speeds* button. For tools 1 and 2, enter 2 mm/s in the *V-Plunge* box and 5 mm/s in the *V-Advance* box. Leave all the other values unchanged (see [figure 2.106](#)).

Parameter

Co-ordinates Tools Misc. parameters Technology Data format Basic Settings

Activation Speeds Dimensions Length compensation

Tool speeds

	V-Plunge	V-Advance	V-Withdrawal	Brake angle
1	2.00	5.00	10.00	30
2	2.00	5.00	10.00	30
3	5.00	5.00	10.00	30
4	5.00	5.00	10.00	30
5	5.00	5.00	10.00	30
6	5.00	5.00	10.00	30
7	5.00	5.00	10.00	30
8	5.00	5.00	10.00	30
9	5.00	5.00	10.00	30
10	5.00	5.00	10.00	30

OK Abort Save...

Fig. 2.106 The tool speed settings

- Click the *Measure* button and set *Depth* for Tool 1 to 2 mm. Leave the *Repetitions* and *Z feed correction* values at zero. For Tool 2, set *Depth* to 3 mm, *Repetitions* to 1, and *Z feed correction* to 2.5 mm (see figure 2.107).

	Depth	Repetitions	Z feed correction
1	2.00	0	0.00
2	3.00	1	2.50
3	1.00	0	0.00
4	1.00	0	0.00
5	1.00	0	0.00
6	1.00	0	0.00
7	1.00	0	0.00
8	1.00	0	0.00
9	1.00	0	0.00
10	1.00	0	0.00

Fig. 2.107 The tool measurement settings

7. Switch to the *Misc parameters* tab. In the *Start/End position* drop-down list, select *Origin + Tool lift*. The default lift is 10 mm (see [figure 2.108](#)).

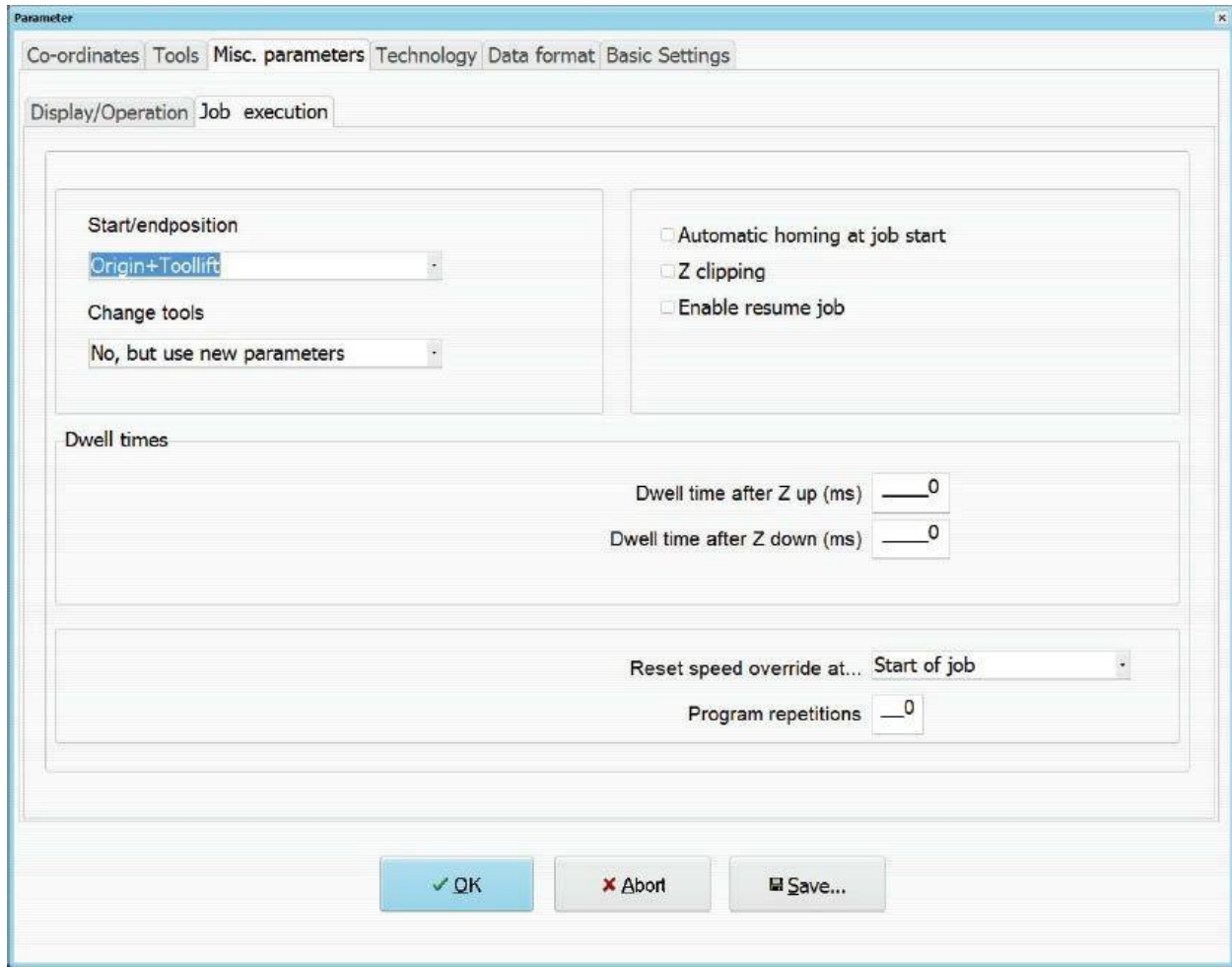


Fig. 2.108 Setting the tool start and end positions

Our trial workpiece measures about 165×40 mm ($6.5" \times 1.5"$). To cut it out, you need to establish a workpiece zero point, which leaves sufficient room for the cutter to the right and to the rear (viewed from the right in [figure 2.104](#)). The zero point is labeled with a red X.

For a workpiece like this, the precise location of the zero point doesn't really matter as long as you leave yourself enough space to work. However, if you are working on a part that already has a predefined shape and requires you to cut holes in specific places, the location of the zero point is extremely important. We will go into how to set a precise zero point later on.

8. Once again, open the *Manual movement* window in the *Move* menu. Move the tool along the X and Y axes to your marked zero

point and lower the tool to about 5 cm (2") above the workpiece.

9. Click *Save to . . .* and select the *Zero point XYZ* option. The machine will assume the tool is now positioned at the front left-hand corner of the workpiece.

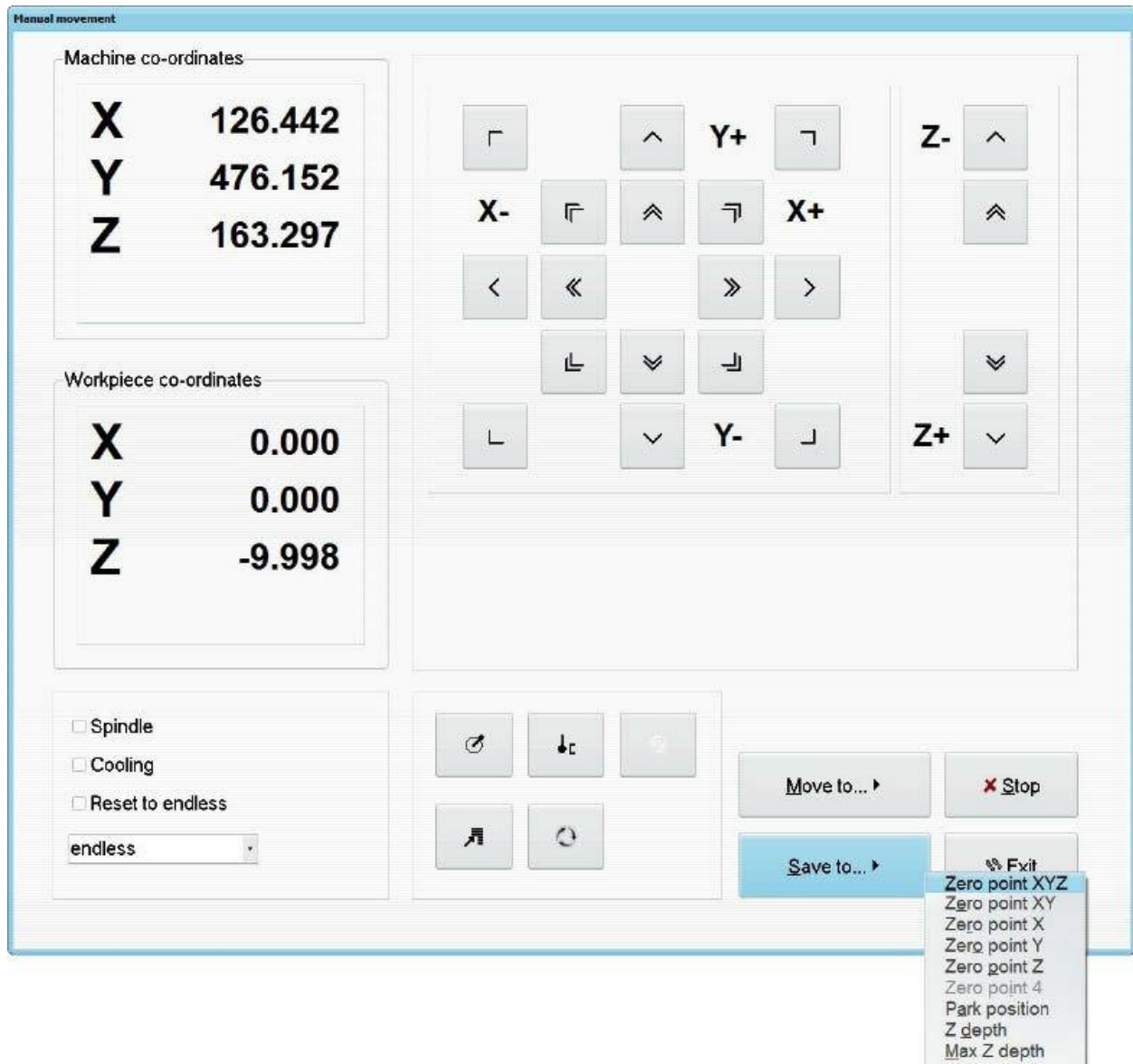


Fig. 2.109 Save the workpiece zero point

10. The machine doesn't know that the tool is actually up in the air, so it's safe to make a test run. Without actually starting the spindle, select the *Start* command in the *Move* menu. The machine will

lower the tool by 10 mm (the default lift value) and make all the passes required by the job. Because we are cutting a relatively small area ($165 \times 45 \times 5.5$ mm), it should be easy to spot any significant discrepancies. The spindle will be raised and lowered at various points where the cutting process drills holes to help it create smooth outlines. Let the machine go through its paces for a few minutes or simply wait until it is finished.

11. If you don't notice any discrepancies, repeat steps 8 and 9, but this time with the tip of the mill about 0.1 mm from the surface of the workpiece. Once the X and Y positions are correct, lower the spindle. Make sure the mill doesn't actually touch the workpiece. Once it is just a couple of millimeters from the surface, use the single-arrow buttons to lower it further. Place a sheet of paper between the mill and the workpiece and continue to lower it until the paper is just clamped in place. Save the new position as *Zero point XYZ*. The mill will then automatically move back up to its 10 mm lift position.
12. Make sure that there are no extraneous objects in or around the machine and that there are no loose cables anywhere. Place a vacuum cleaner nearby so you can remove the shavings while you work, although this particular piece doesn't produce too much dust. Be sure not to touch the spindle or the mill with the vacuum cleaner.
13. Switch the spindle on, select a speed setting of 5, and select the *Start* command in the *Move* menu. The machine will now begin to create your very first workpiece.



Fig. 2.110 Results that make a hobbyist's heart beat faster . . .

If everything went according to plan, your result should look like [figure 2.110](#), and you can knock the finished piece out of the blank. Remove any remaining dust and dirt from in and around the machine. You are now officially ready to roll. Remember to regrease your machine after two hours of operation.

3 Milling Tools

To produce successful milling results, you need to know a lot about the tools and raw materials you use. Simply taking the next best tool and forcing it into a blank at top speed is just as ineffectual as machining too slowly. Both approaches cause excess tool wear and unnecessary costs as a result. Incorrect use of tools also reduces the precision and quality of the parts you produce.

This chapter explains all you need to know about hobby-grade milling tools and how to use them with a gantry-type machine. This means we will be talking mostly about end mills.



3.1 Milling Tools

Various aspects of a milling tool are precisely defined for every type of raw material and each type of job and are aimed at providing the following:

- Optimal use of the tool
- Maximum lifespan
- High machining speeds
- High general quality and quality of finish in a workpiece

The requirements for hobby and industrial use differ significantly. Industrial users include tool wear in their costing schemes and generally maximize production speeds in order to reduce staffing and other costs. In contrast, hobbyists don't have to pay rent or staff and generally need to keep overall costs as low as possible. It therefore makes little sense for most hobbyists to purchase industrial-grade tools.

The following sections contain calculations involving the parameters listed in [table 3.1](#). Some of these are specified by tool manufacturers, while others have to be calculated or estimated by the user. The precise meaning of each term is explained in the course of the text.

Parameter	What it Means	Units
a_e	Cut width	mm
a_p	Cut depth	mm
d	Tool diameter	mm
f_z	Feed per tooth	mm
f	Feed rate	mm/min
n	Spindle speed	rpm
v_c	Cutting speed	m/min
z	Number of teeth/flutes	-
SL	Cut length	mm

Table 3.1 Important machining parameters

3.1.1 Teeth and Flutes

End mills have one or more teeth for removing material from a workpiece, and each tooth has a slot called a flute that helps remove waste material during the cut. The larger the flute, the better waste can be transported away from the workpiece. End mills are usually referred to by the number of flutes they have, and are correspondingly called single-flute, double-flute, and so on. These are sometimes also referred to 1-flute and 2-flute end mills. When chips and shavings clog the flute, the mill can no longer cut effectively. One flute per tooth is ideal, but the more flutes a mill has, the smaller each has to be, and smaller flutes mean poor waste removal. Flutes vary in steepness and are cut in right- or left-hand spirals, or run straight up the shank of the mill.



Fig. 3.1 A right-hand spiral 2-flute end mill

Right-hand spiral flutes remove material upward, and the shallower the pitch, the more effectively chips are removed. A flute has similar physical characteristics to a screw thread and exerts an upward force where it has contact with the workpiece. This leads to tears in the surface of fibrous materials such as wood, and unless you take preventative measures, can cause unwanted flex in thin materials.

The steeper the pitch of the flute, the weaker the vertical force it exerts. In a straight flute the pitch is infinite, thus creating zero vertical drag. In principle, this is a good thing, although it does impair chip removal. Straight flutes are suitable for use in situations where there is plenty of space to the side of the workpiece or if you use a vacuum system (or compressed air) to remove chips immediately.

Left-hand flutes exert a downward force on the workpiece. This further impairs chip removal and is only suitable for use with appropriate removal systems. Downward pressure means that the edges of cuts

seldom tear and the workpiece is pressed into the machine table, thus minimizing milling inaccuracies.



Fig. 3.2 Left-hand (front) and right-hand spiral end mills

You may be thinking that a single-tooth mill is most effective, but things are not quite that simple. The number of teeth is not a random choice. By nature, single-tooth mills have a large flute, which makes them less stable and more prone to breakage. Furthermore, a single cutting edge becomes blunt twice as fast as a double-tooth edge.

The following rules of thumb are a good guide to choosing the number of teeth and flutes in a mill:

- The harder the material, the less you can remove per cut. Mills for hard materials have more teeth and thus a higher milling speed.
- The softer the material, the easier it is to cut and the more chips it produces. Soft materials thus require a lower number of teeth and correspondingly large flutes.

3.1.2 Cutting Length and Shank Types

The cutting length is the total length available for cutting, measured from the tip of the mill. This dimension is especially important when the cut

width (i.e., the diameter of the teeth/flutes) is narrower than the diameter of the shank. In this case, the cutting depth is the maximum depth to which the mill can engage with the workpiece. Any attempt to cut deeper will cause the shank to collide with the workpiece surface. If the flute diameter is larger than that of the shank, you can mill over the entire free length of the mill.

There are various types of holders for attaching an end mill to the spindle. For the purposes of this book, we will be looking mainly at simple straight-shank mills, although hobbyists often use mills with a flat clamping surface, too. Flat-shank mills can be used with a normal chuck, although the space this leaves creates a slight imbalance in the rotating tool that can cause vibrations and poor-quality cuts.

3.1.3 End Mill Materials

To cut directly into solid metal, milling tools have to withstand enormous loads. The various parts of a cutter are subjected to different forces. The cutting edges especially are subject to a number of forces that vary within milliseconds during a cut.

To prolong the life of the teeth in an end mill, these have to be as hard and resilient as possible, while the shank doesn't need to be as hard as the teeth but needs to be less prone to breakage. High-quality cutters are therefore made of two or more different materials designed to deal with the specific forces each part is subjected to.

Milling tools, like most cutting tools, are usually made of two basic materials:

- High-speed steel (often abbreviated to HS or HSS)
- Carbide steel

HSS is softer and more resilient and tolerates greater force. HSS mills are therefore suitable for jobs that require the removal of large amounts of material, but wear out more quickly in the process. Carbide steel mills are usually made from tungsten carbide compounds and are extremely hard.

They are made by compressing and sintering tungsten carbide powder with cobalt- and nickel-based binding agents, combined with extremely hard materials such as titanium carbide and tantalum niobium carbide. These materials reduce absolute hardness but increase resilience. Carbide steel tools wear out less quickly than HSS tools but tend to break more easily.

Solid carbide tools are made entirely of carbide steel. HSS tools in which the cutting edges only are made of carbide steel are often used in continuous-use situations. Tools are also available with teeth made of synthetic polycrystalline diamond (PCD) or polycrystalline cubic boron nitride (PCBN).

Modular machining techniques involve the use of interchangeable tooling components. Modular cutting tools have multiple identical cutting edges that can be rotated into place when one wears out. When all the cutters are worn out, the entire tool has to be replaced.



Fig. 3.3 A TiAlN-coated end mill

Other types of tools are coated with nitrides such as titanium nitride (TiN) or titanium aluminum nitride (TiAlN). Such coatings are harder than the core material of the tool and provide a smooth surface that further reduces wear. Used with softer materials such as aluminum or copper, coated mills help prevent edge buildup. Coatings also create a thermal barrier that protects the softer core material from the high temperatures produced by the milling process.

Not All That Glitters Is Gold

Drills and end mills are the subject of plenty of scams. Genuine HSS and carbide steel tools are seriously tough and have a long useful life. However, the online world is full of exotic-sounding tools, supposedly with nitride coatings that are, in fact, simply painted on. Used for a while, the paint wears off and reveals the poor-quality material beneath. Instead of the promised lifespan of hundreds of jobs, bargain tools often break the first time they are used.

3.1.4 Cutting Speed and Feed Rate

The most important (and usually only) setting you can alter for the spindle is its speed. This and the X,Y, and Z feed rates, and the cut-per-pass and tool overlap increments are the most important milling parameters. Each of these parameters has to be set according to the machining speed suitable for the raw material you are working on. The tool has to be selected according to the material, and the spindle speed and feed rate depend on the type of tool. The corresponding values are not absolute and can be selected from a range that is limited by additional criteria.

The recommended cutting speed for a tool is given in m/min. Typical values lie between 50 m/min and 500 m/min. This may sound like a lot, but it doesn't mean that the tool travels 500 meters per minute. What it actually denotes is the distance covered by the rotating tool within a minute. This is calculated by multiplying the tool's diameter d (in mm) by the number of revolutions per minute:

$$v_c = \frac{d \cdot \pi \cdot n}{1000} \frac{\text{m}}{\text{min}}$$

For example, an end mill with a diameter of 3 mm is driven at 20,000 rpm. The resulting cutting speed is given by the following formula:

$$v_c = \frac{3 \cdot \pi \cdot 20000}{1000} \frac{\text{m}}{\text{min}} = 3 \cdot \pi \cdot 20 \frac{\text{m}}{\text{min}} = 188,49 \frac{\text{m}}{\text{min}}$$

If you know the cutting speed, you can calculate the spindle speed by converting the formula for v_c as follows:

$$n = \frac{v_c \frac{\text{m}}{\text{min}} \cdot 1000}{\pi \cdot d \text{ mm}}$$

For example, v_c for our 3 mm end mill is known to be 250-350 m/min. The spindle speed can be calculated thus:

$$n_{\min} = \frac{250 \frac{\text{m}}{\text{min}} \cdot 1000}{\pi \cdot 3 \text{ mm}} = 26525 \frac{1}{\text{min}}$$

$$n_{\max} = \frac{350 \frac{\text{m}}{\text{min}} \cdot 1000}{\pi \cdot 3 \text{ mm}} = 37136 \frac{1}{\text{min}}$$

This means the spindle speed range for this particular tool is 26,000-37,000 rpm. In comparison, the spindle speed for a 4 mm mill is just 12,000-20,000 rpm. If your spindle has a maximum speed of 10,000 rpm, the only way to increase cutting speed is to use a larger end mill. The example above shows clearly that a larger end mill enables you to increase cutting speed or retain cutting speed at a reduced spindle speed.

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 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.

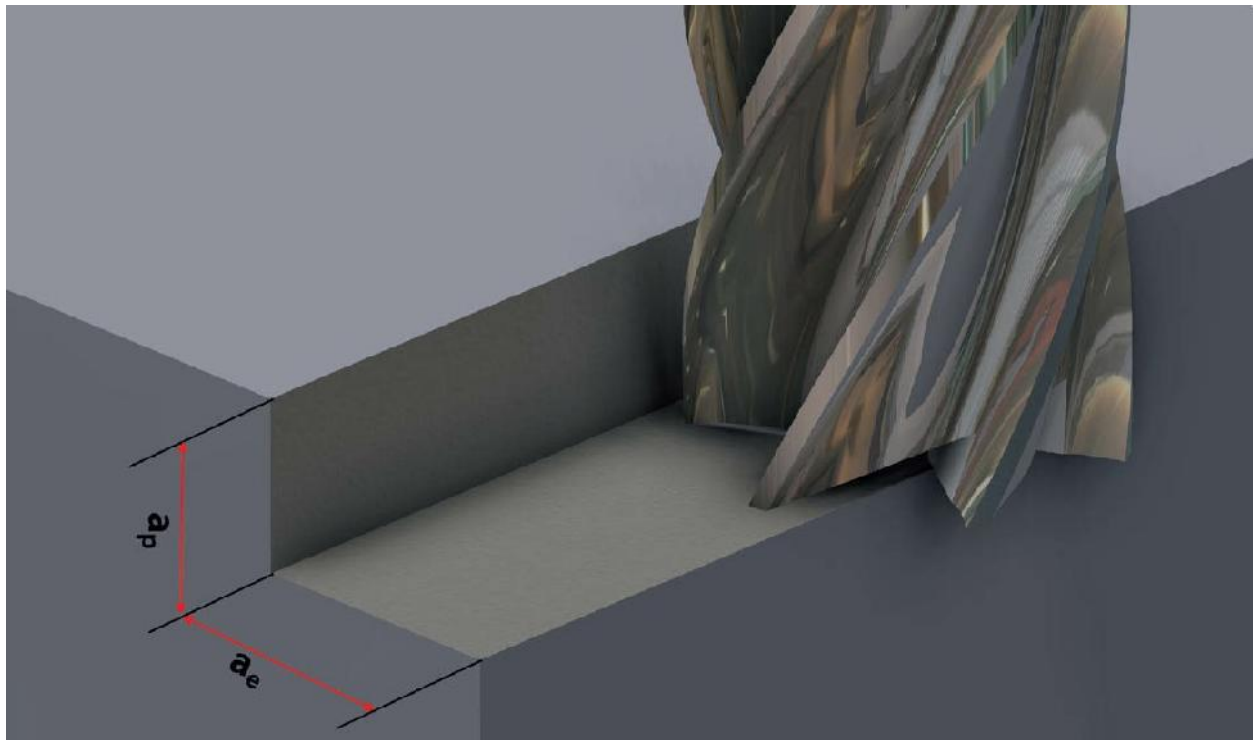


Fig. 3.4 *Cut width and depth*

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.

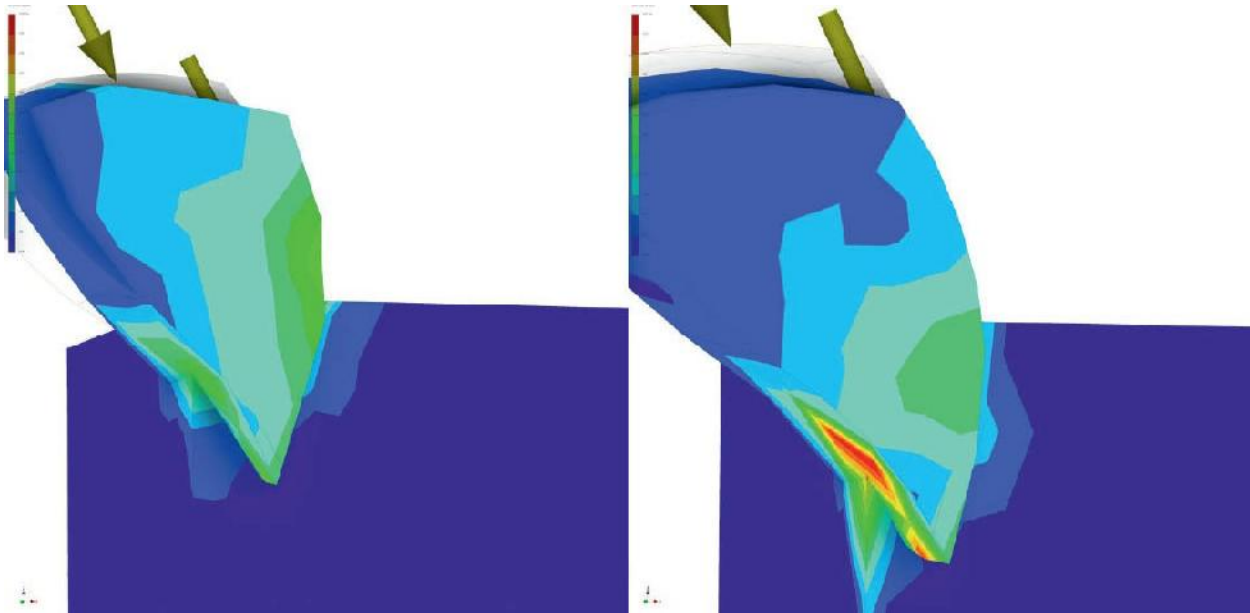


Fig. 3.5 An FEA simulation based on different feed-per-tooth values

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 in the following sections are all great aids to the milling process, but are not all suitable for use with all types of machines, tools, or materials.

Furthermore, some cooling techniques produce a lot of additional dirt and often require dedicated systems that are not suitable for home use. The best cooling system also depends on the material your tools are made of. Carbide steel tools tend to crack when cooled suddenly, thus significantly reducing their working life.

Compressed Air Cooling

A powerful stream of air pointed directly at the tool is one way to reduce the amount of heat generated during a cut. This is an ideal method to use for absorbent materials (such as wood) that prohibit the use of fluid coolants. Compressed air also helps to remove swarf. If you are cutting wood, make sure your airstream isn't corrupted with oil, as is the case with many compressors. Oil always produces spots and stains when absorbed into wood surfaces.

On the downside, compressed air tends to distribute shavings all around the workspace, and the additional oxygen that it pumps into the air increases the risk of combustion when tools and raw materials get hot.

Compressed air cooling doesn't actually cool the tool, but contributes to overall cooling by removing the hot waste material that would otherwise clog and overheat the tool.

Minimum Quantity Lubrication (MQL)

Applying a tiny quantity of lubricant makes many materials much easier to cut. Lubricants reduce friction between the tool and the workpiece and help prevent dust from sticking to the tool. MQL uses compressed air to distribute lubricant over the tool and uses as little lubricant as possible while maintaining a permanent effect.

MQL is highly effective for removing metal swarf and significantly

extends the life of high-quality cutters.

The heat generated by the tool and its rotational movement create an aerosol effect that atomizes lubricants. The resulting haze poses a health risk, so take appropriate care when using this cooling method.

Overflow and High-Pressure Cooling

Overflow cooling works by applying a constant stream of coolant to the tool and the workpiece. In addition to reducing friction, applying large amounts of coolant reduces the temperature more effectively than MQL. Some coolants are water-soluble while others are not, but both types have to be carefully disposed of. Overflow cooling involves large amounts of fluids that are distributed widely around the work area and that usually produce unhealthy suspensions of oil particles in the air. These factors make overflow cooling unsuitable for home use.

High-pressure cooling works in a similar way to overflow cooling, but applies coolant directly to the tip of the tool at very high pressures of up to 7,250 pounds per square inch (psi). This pressure not only clears chips but also pulverizes them, which in turn provides more effective cooling and allows faster cutting speeds. As you will probably not want to spend time removing coolant from the walls and ceiling, this method isn't suitable for home use (and your machine is probably not waterproof anyway!)

Liquid coolants are not generally used with roughing mills.

Cryogenic Cooling

Industrial users often use liquid nitrogen or carbon dioxide when cutting ultrahard materials such as titanium and nickel alloys. These are high-end techniques that are not practicable for home use.

Manual Cooling

Manual cooling is the simplest method, and involves using an oil can, a brush, or a sprayer to apply cutting oil or other coolants directly to the tool while you work. If you use this method, make sure that you don't

stray into the machine's hazard zone when applying coolant. You also have to find a way to effectively remove excess coolant from the work area and avoid breathing in atomized oil.

Underwater Milling

Underwater cooling is an alternative active cooling method that cools the tip of the tool and the entire workpiece. The main advantage of this approach is that it immediately binds the dust particles produced by the cutting process, thus eliminating any health risks. However, this method only works well with short chips. Longer chips tend to get caught in the mill and spray water from the coolant bath around the workspace. You also have to ensure that dust particles don't collect on the workpiece, as these tend to abrade the tool's teeth.

Underwater milling is commonly used when cutting hazardous materials such as carbon fiber or glass fiber, but can be used with metals and plastics too. It is not suitable for use with wood or other absorbent materials.

3.2 Which Tool for Which Material?

The choice of tools is enormous, and it can be hard getting to grips with all the exotic-sounding product names and specifications. When shopping online, it is amazing just how many of the tools have *extremely* long life, *extreme* chipremoval qualities, and fantastic properties in general. Descriptions are often peppered with phrases like "industrial-grade" and "high-precision" and, if we are to believe what we read, there appears to be no difference between cutters that costs US\$5 and ones that costs US\$100.

Genuine industrial cutters are usually made for use with parameters that a hobby-grade machine cannot provide, and it makes no sense to use high-grade tools at a fraction of their intended capacity.

It is often difficult to tell how good a tool really is, and the only way forward is to rely on the experience and recommendations of others or

simply to try it out for yourself. Stick to ordering individual tools rather than sets when you are starting out. If you buy a set and find that one of them is of poor quality, the others will probably be no better.

As you will probably not be milling hard or especially resilient materials, the following sections discuss only single- or two-flute end mills suitable for use with the following materials:

- Wood
- Fiber composites
- Plastic
- Circuit boards
- Aluminum
- Brass

3.2.1 Solid Carbide End Mills

Right-hand Spiral 2-Flute Flat-nose End Mill

This type of end mill is ideal for use with hardwood, aluminum, and brass, but not for softer materials.

The right-hand flutes enhance chip removal, making these tools great for cutting holes, pockets, slots, and outlines. The flat tip makes them robust, cuts the entire face surface, and helps keep the milled surface smooth.

Right-hand Spiral 2-Flute Fish-tail End Mill

This type of end mill is also ideal for use with harder woods, aluminum, and brass, but not for softer materials.

They are used for similar jobs to the ones mentioned above, but the fish-tail tip makes the face edge very sharp and enhances the tool's plunge capability.

Due to the V-shaped fish-tail tip, chips tend to get stuck between the

teeth when the tool enters the surface of a workpiece and have nowhere to go, which places excess load on the teeth if you cut horizontally once the tool has engaged the surface of the workpiece.

Right-hand Spiral-tooth Fish-tail End Mill

This type of mill is great for use with composite materials such as carbon fiber, glass fiber, reinforced plastic, and circuit boards, but is not suited to jobs that involve conventional plastics and metal.

The spiral flutes produce small chips that are easily removed by the right-hand spiral form, and the fish-tail tip makes them great for plunging and drilling.

The teeth are relatively thin and sharp, so they are ideal for making clean cuts and milling inner edges. Once again, due to the V-shaped fish-tail tip, when the tool enters a surface, chips tend to get stuck between the teeth and have nowhere to go, which places excess load on the teeth if you cut horizontally once the tool has engaged with the workpiece. Note that this effect is less significant when you are working composite materials that are softer than aluminum and brass.

Micro-toothed Fish-tail End Mill

This type of mill is great for use with composite materials such as carbon fiber, glass fiber, and reinforced plastic.

This type of mill is very similar to standard spiral-flute models but produce smaller chips. The fish-tail tip makes them great for plunging and drilling.

Right-hand Spiral Single-tooth End Mill

This type of mill has an extremely sharp cutting edge, a shallow lip angle, and a broad flute, making it ideal for use with soft plastics, thermoplastics, and softer woods. Coated versions are often used to mill aluminum, brass, and copper.

The broad flute ensures optimum chip removal and is ideal for cutting deep slots and channels. When used to cut soft materials, the sharp edge

produces large chips and little friction, thus producing less heat in the raw material and preventing melting, tool sticking, and edge build-up.

The single cutting edge wears out faster than multiple teeth. The broad flute and the asymmetric shape also make the tool less stable and more prone to breakage. Always use this type of tool within the prescribed tolerances.

Right-hand Spiral 2-Flute Ball-nose End Mill

The tip of this type of mill is rounded, making it a great universal tool for cutting wood and metal.

Ball-nose mills are ideal for plunging and horizontal milling, and make it simple to cut rounded slots and curves.

The rounded end is also well suited to smoothing sloped surfaces, making ball-nose mills great for 2.5D and 3D milling.

Engraving Cutters and Gravers

Engraving cutters and gravers are used to remove very small amounts of surface material. An engraving cutter has a spiral flute like a normal cutter, while a graver has a simple sharp point and no flute.

Due to the fine nature of engraving work, engraving tools are extremely sharp. Specialized broad-bladed gravers are used to engrave two-dimensional shapes into flat surfaces.

3.2.2 Unusual Types of Cutters

Polished and Lapped Flutes

Polishing the flutes in a cutter reduces friction, improves chip removal, and prevents small waste particles from sticking to the tool.

Lapping is a process that is similar to grinding, and is also used to smooth the flutes in milling tools.

Left-hand Spiral End Mills

Left-hand flutes drives chips downward rather than pulling them upward and out of a cut. This type of mill is mostly used to cut thin materials that would be pulled out of alignment by the forces produced by a right-hand spiral mill.

Left-hand mills are not suitable for cutting holes and slots, or other types of deep, narrow cuts, where they would simply fill the cut with chips and clog the tool.

Hard-coated End Mills

The coatings most often used for end mills are titanium nitride (TiN) or titanium aluminum nitride (TiAlN), which is the thermally more stable of the two.

Coatings are usually just a few micrometers thick and protect the tool's core material from excessive wear. Core materials are very hard, but are nevertheless prone to damage through oxidation, high thermal loads, and the adhesion of waste particles.

Hobbyists often use TiN- and TiAlN-coated tools to prevent edge build-up when milling aluminum. Some hobbyists also use tools coated with titanium carbide, zirconium oxide, and aluminum oxide.

Powder-coated End Mills

Some tools are coated with diamond or boron nitride powder by way of a sintering process. Such coatings have thicknesses between a few micrometers and 2 millimeters.

Polycrystalline diamond (PCD) coated mills are available on the broad market and are ideal for cutting plastics strengthened with a variety of fiber additives. Coated tools last much longer than their non-coated counterparts.

Monocrystalline diamond (MCD) edged tools are extremely expensive and don't find their way into most hobbyists' workshops. They are often used for extreme polishing and creating curves in plastics.

3.2.3 Long Mills

Regular end mills generally offer cutting lengths up to around 10 mm. If the diameter of the toothed end is greater than that of the shaft, you can cut deeper; whereas if the shank and the toothed end have the same diameter, the cutting length is the limit of the depth you can reach.

The shorter the tool, the more rigid it will be and the fewer vibrations it will generate during a cut, so it makes sense to use shorter tools when cutting hard materials such as brass. The less rigidly a tool sits in the chuck, the more likely it is to bend or break under load.

Tools with extra-long shanks are also available for jobs that require especially deep cuts. Using this type of tool requires great care due to the extreme leverage involved.

3.2.4 Forming Cutters

Forming cutters are specially designed to cut custom radii, chamfers, and slots. They are made the same way as other cutters but are usually designed to perform one specific task.

3.3 Upcut and Downcut Milling

Section 1.2.4.2 describes the pros and cons of upcut and downcut milling. This section tells you all about when and how to use each technique.

During upcut milling, the forces exerted on the tool increase during the cut and cease abruptly when the cutting edge exits the workpiece. If you use this technique to cut a soft material such as wood, these forces can cause the edge of the cut to splinter. In contrast, the forces created by downcut (synchronous) milling are greatest at the moment the tool engages with the workpiece. The chips get thinner and the force exerted on the tool diminishes during the cut, thus reducing the risk of splintering. Because this technique produces no sudden dissipation of force when the tool exits the workpiece, it also produces much fewer vibrations—known as chatter—in the tool.

Cutting a slot all the way through a workpiece simultaneously involves up- and downcuts. Because the forces on the up- and downcut

sides of the slot vary, the tool will tend to wander toward the weaker force and away from the target position.

Generally speaking, synchronous milling is preferable to upcut milling, but only if you are sure your machine is completely free of play. If this is the case, the tool will pull the workpiece in the direction of feed.

If your machine's drive has any play at all, the workpiece can get pulled toward the tool until the feed drive re-engages. This effect produces permanently fluctuating feed rates and, as a result, inconsistent cuts. If this is the case, it is better to use an upcut strategy, as this produces constant pressure against the tool and there will only be play in the system for a brief moment when you change the direction of feed.

3.4 Tool Maintenance

Taking proper care of your tools will extend their life. It is particularly important to ensure dust doesn't build up on the teeth and flutes, as these eventually cause edge buildup while you are cutting. Dust particles stuck to a mill create an ideal surface for more particles to adhere to.

Fine dust can be removed using acetone, turpentine, or methylated spirit. Natural resin makes wood chips sticky, and these are easiest to remove using resin remover for chainsaws and hedge trimmers. Wood resin gets really hard if left to set and can create really stubborn edge buildup or flute clogging if you don't remove it immediately.

Industrial cutting tools are designed to be reground or their cutting edges replaced, whereas hobby-grade cutting tools are usually disposable. It is virtually impossible to sharpen the teeth of a conventional end mill without significantly altering its geometry.

Always check your tools before you start a job and dispose of any worn or damaged ones. Certain types of damage are often caused by incorrect usage:

- If teeth or other parts of a mill are discolored yellow or blue, this is a sign that the tool has been allowed to overheat. Too much heat in hard metals causes them to soften, which reduces their efficiency and

causes them to wear out much faster than they otherwise would.

- If the edges of a cutter blunt more quickly than you expect, you are probably using cutting speeds or feed rates that are too low, or it may simply be an indication of a poor-quality tool.
- Aside from manufacturing defects, broken teeth are a sign that the spindle speed is too low, or that the tool has been plunged too deeply into the workpiece.

4 2D Milling in Practice

2D milling is performed using a model that determines the position of the cutting tool on a two-dimensional horizontal plane. However, in order to actually make a cut, we also have to move the tool vertically. The amount of material that can be removed per pass is limited, so we have to program multiple vertical and lateral passes to complete each job. After all, milling would be an extremely limited process if it didn't enable us to cut to different depths in different places.



To give us the flexibility we need, a milling machine is equipped with a third, vertical axis of movement. However, because a 2D model includes no data regarding the depth of the cut, this has to be entered manually. The 2D workpiece in section 2.11.6 was made without the use of complex CAD or CAM software. In fact, a simple drawing application and a plotter driver can produce a plot file that does the same job.

2D milling is ideal for producing all types of signs and boards like the

Perspex logo shown in [figure 4.1](#). This is also a great way to create custom gifts, place tags, or even visiting cards made from carbon reinforced plastic.



Fig. 4.1 A logo cut into translucent Perspex

It is also great for making signs out of wood, brass, and aluminum, and is a favorite technique among airplane and ship model makers for creating brackets and frames made of wood or composites.

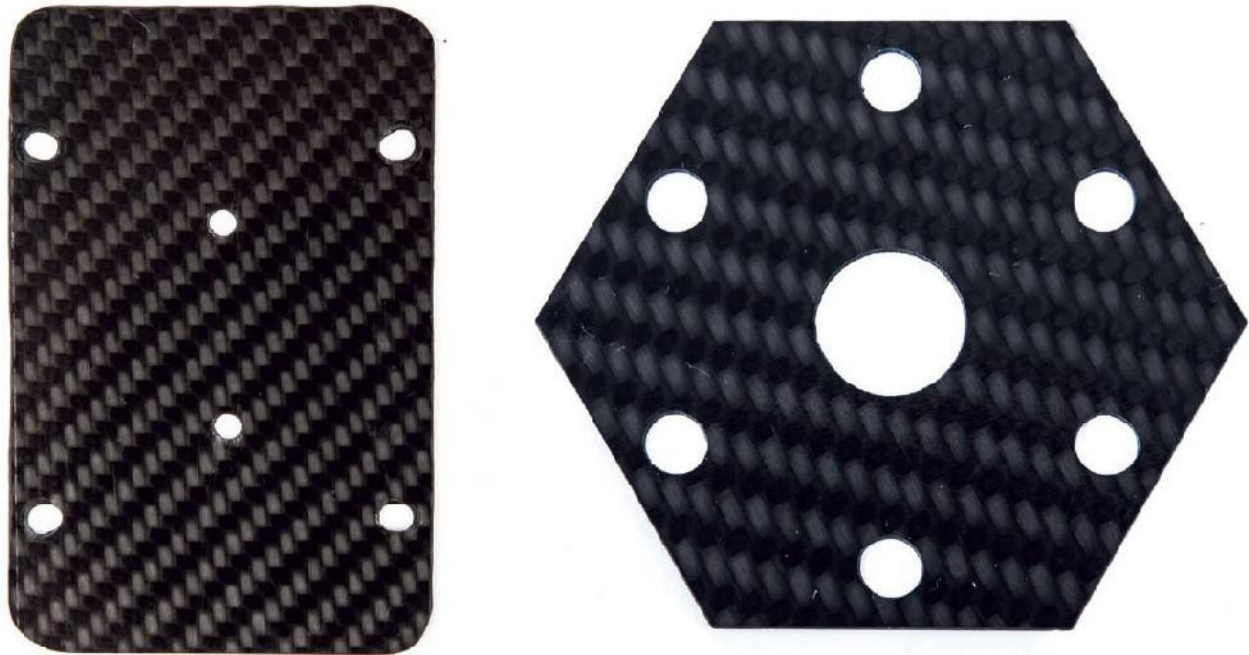


Fig. 4.2 *Carbon fiber reinforced plastic workpieces. The one on the left was created manually, while the one on the right was milled.*

This chapter goes into detail on the tools and settings required to successfully 2D mill various materials. It includes sample parameter calculations and, to keep things consistent, all our examples are based on the sign shown in [figure 4.3](#).



Fig. 4.3 *Our sample 2D milling pattern*

Even if the pattern itself appears simple, it is a challenge to mill with

certain materials. To keep the text looking clean, you have to use the smallest possible end mill. Because we used a circular mill, the vertical inner edges of the cuts in the finished piece all have rounded radii. A 1 mm mill produces curves with a radius of 0.5 mm. To keep the work as simple as possible, we used a plot file and WinPC-NC instead of more complex CAM software.

Milling Quality

To keep things as simple as possible, we chose a relatively simple pattern created using a plot file. However, this approach isn't ideal and produced some rather strange passes in the course of the job. For example, the curves of the letter "b" have two obvious edges that aren't part of the original model. These are not milling errors, but rather errors that occurred during the conversion of the plot file to G-code.

This chapter doesn't aim to produce super-precise milling results, but rather to explain which tools and parameters you can use to produce satisfactory results with a range of materials. Even if your machine control software has built-in CAM functionality, it is always preferable to use a dedicated CAM application whenever possible.

Wherever possible, we milled the pattern using an initial pass at a depth of 1.5 mm followed by a second 0.5 mm pass. The results shown in the illustrations are not digitally processed in any way and the workpieces aren't completely freed of chips and dust.

Cutting Parameters

Some of the required spindle speeds and feed rates are not available using a Stepcraft-2/600 and a Kress 800 FME spindle. However, the values quoted in the course of this chapter are all more or less suited for use with this or similar machines. The photos indicate the degree of accuracy you can expect from the quoted settings, and don't be afraid to conduct your own experiments!

4.1 Clamping the Workpiece

The greater the accuracy required by the job, the more important effective clamping of the workpiece becomes. It is essential not only to ensure that it doesn't slide around, but also that the lower levels of the horizontal X and Y planes are orthogonally oriented with regard to the cutting tool. If this is not the case, the following issues can ensue:

- Engravings in a sign will have uneven depth or won't get cut at all
- In facing jobs, the upper surface will be straight but won't be parallel to the lower surface
- If you then flip the workpiece over and mill the lower surface flat, it still won't be parallel to the upper surface. The inconsistency will then be perpetuated throughout all subsequent steps.

This kind of issue is usually due to the use of an inappropriate or incorrectly used spoilboard. Machine tables and spoilboards made of wood or other soft materials are most susceptible to errors, but even a solid aluminum T-slot table can flex by a couple of hundredths of a millimeter if it is subjected to inconsistent load. Rock-solid stability is only really a given with solid-steel machine tables with a thickness of a couple of inches or more.

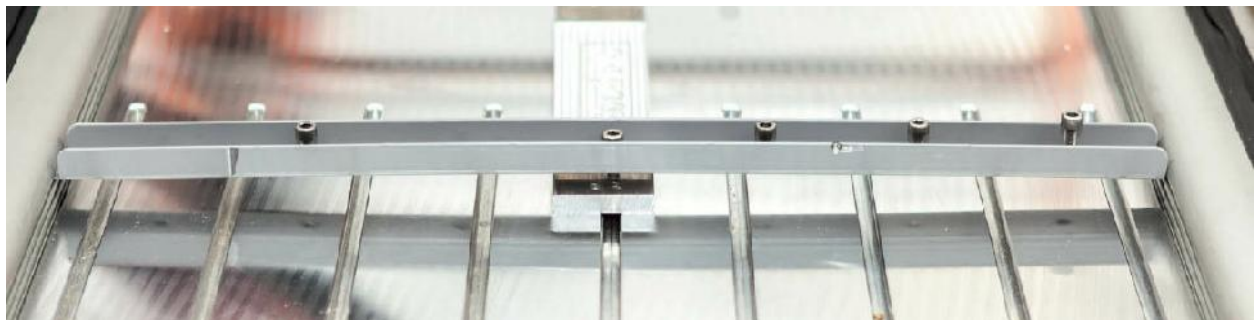


Fig. 4.4 *Because it is held in place using only a single bolt, the clamp in this photo is unevenly tensioned*

It is not sufficient simply to tighten your clamps as far as possible. You need to use enough force to keep the workpiece in place but no more, and this is often less than you might think. Furthermore, it is essential to avoid leverage effects like the one shown in [figure 4.4](#). Because the clamp in the photo is held only by the lateral guides and a single screw in the center, the gap between the fixing points is too long and causes the clamp to flex. Clamping using t-slots close to the workpiece would create a much shorter lever that cannot flex so easily.

As well as minimizing leverage, it is important to clamp your workpiece evenly. If you are fastening a workpiece in place using two

clamps and you tighten the first one too much, the workpiece can end up off-kilter and can no longer be leveled using the second clamp.

To check whether your workpiece is level, move the cutter over the workpiece and, using a sheet of paper as a buffer, lower it until it holds the paper in place and note the Z value displayed by the control software. Repeat this several times in different places and compare the Z values you have recorded. This method is not ultra-precise, but it is sufficient for detecting discrepancies of up to 1/10 mm.

To stabilize your workpiece, you can simply use more clamps or a different type. [Figure 4.5](#) shows an example of how to clamp a workpiece so that it doesn't shift laterally. In this case, much less downward force is required to secure the workpiece.

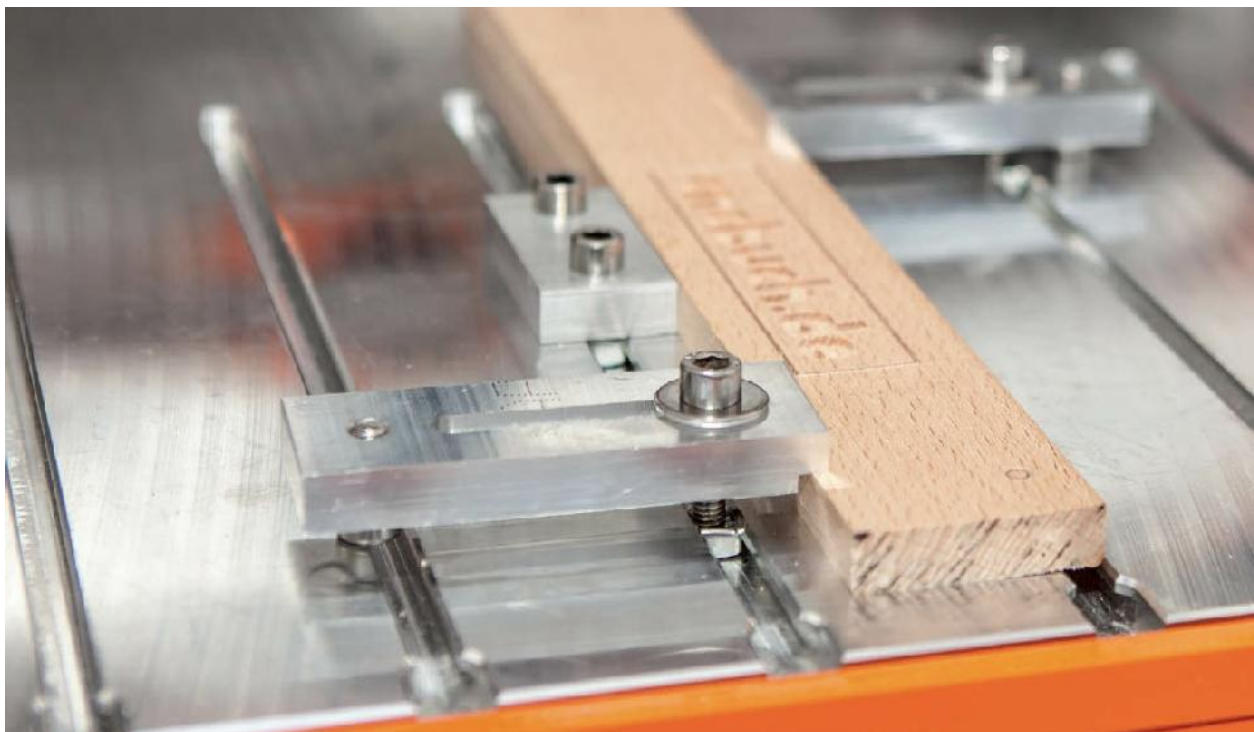


Fig. 4.5 Clamping a workpiece using lateral rather than downward force

4.2 Softwood

Because wood is a poor conductor of heat, a tool with blunt teeth or one used at too low a feed rate will quickly heat up and either blacken wood

or make it so hot that it glows. Using incorrect cutting speeds is a sure-fire way to blunt even the most expensive mills in record time. Softwood has to be milled at moderate cutting speeds using sharp tools and high feed rates.

Softwood is generally quite cheap, and is even cheaper if you purchase large boards and cut them to size yourself. This makes it ideal for creating prototypes and spoilboards. Prototypes are used to check the accuracy, functionality, and design of a piece.

A square yard of 1/2-inch MDF costs around US\$10, while a similar-sized sheet of carbon-fiber-reinforced composite costs around US\$1,500. It therefore makes a lot of sense to check your model for errors using cheap wood before you actually go ahead and cut an expensive piece of carbon fiber board.

Machine Data	
Feed per Tooth f_z	0.01-0.1 mm
Cut Depth a_p	0.5-3mm
Cut Width a_e	Up to 50 per cent of the tool diameter where possible
Plunge Rate	5-20mm/s
Milling Type	Downcut wherever possible
Tools	
# of Flutes	1 or 2
Spiral Type	Right-hand. For thin materials, use mills with a shallow flute pitch or left-hand flutes.
Material	Solid carbide
Coating	Not necessary
Cooling	Air cooling if necessary, but also works without

Table 4.1 Basic cutting data for softwood

4.2.1 Example #1: MDF

For this example, we used a noncoated 1 mm single-flute right-hand spiral end mill made by Sorotec (model # L1S VHM). For use with plastics, the manufacturer quotes a feed-per-tooth value of 0.04-0.065 mm per revolution at 32,000 rpm. The cutting speed for MDF is 40-80 m/min.

We used an rpm value n of 21,000 (setting #4) and a feed-per-tooth value f_z of 0.05 mm. This gives us a feed rate of 1,050 mm/min (17.5 mm/s) and a cutting speed of around 66 m/min.



Fig. 4.6 *The result of milling our pattern in MDF*



Fig. 4.7 *Detail of the edges of the cut*

Overall, the result is clean and shows no charred edges. In [figure 4.6](#), you

can see how the circles in the centers of the letters “b” and “d” have broken off due to the softness of the material.

4.2.2 Example #2: Poplar Plywood (using inappropriate parameters)

This time, we selected a noncoated 1 mm right-hand spiral 2-flute fish-tail end mill. We had no detailed specifications, so we assumed the same 0.05 mm feed per tooth that we used with MDF. The recommended cutting speed for this type of material is around 20 m/min. This gives us a spindle speed of around 7,000 rpm, which our Kress spindle isn’t capable of producing.

Instead, we selected the same 21,000 rpm speed that we used for MDF and a feed-per-tooth value of 0.025 mm. This gives us a feed rate of 1,050 mm/min (17.5 mm/s) and a cutting speed of just under 66 m/min.

In this case, we deliberately selected unworkable parameters to get a clear picture of the effects they have.



Fig. 4.8 *A poor result produced using poplar plywood*



Fig. 4.9 *Detail of the edges of the cut*

The photos clearly show the frayed edges and a tear in the surface between the “b” and the “u.” These effects are caused primarily by the very short fibers in the wood that provide virtually no vertical or lateral stability in the thin uncut areas between the individual letters. Furthermore, the bad parameters we used caused the mill to exert too much force on the wood, and the right-hand spiral of the flute exacerbated this effect by pulling on the workpiece. This effect was less prominent in the MDF example because the material is soaked in resin, making it behave more like plastic than wood.

4.2.3 Example #3: Poplar Plywood (using appropriate parameters)

Using a 1 mm right-hand 2-flute fish-tail end mill significantly improved the result. We used the same cutting speed but reduced the feed rate to 10 mm/s. We also clamped the workpiece so that the thin fibers between the letters matched the direction of the grain. These parameters are better but still *not optimal*.



Fig. 4.10 The same cut made with a different tool and more useful parameters

You can see how much less the edges have frayed and how altering the direction of the grain has helped to prevent damage to the thin ridges between the letters. [Figure 4.11](#) shows that the vertical surfaces within the letters are clean and that there are no signs of charred wood. In other words, even though the parameters are not ideal, they are still pretty good.



Fig. 4.11 Detail of the improved version

4.2.4 Example #4: Poplar Plywood (left-hand spiral mill)

For this attempt, we used a left-hand spiral 1 mm fish-tail mill. We

further reduced the feed rate to 8 mm/s and set the spindle speed to about 11,000 rpm (setting #1.5). This gives a cutting speed of about 34 m/min and a feed-per-tooth value of 0.02 mm. We once again clamped the workpiece with the grain running parallel to the text (see [section 4.2.2](#)) to demonstrate the effect of using a left-hand mill.



Fig. 4.12 The cleanest result so far using poplar plywood

You can still see a few loose fibers, although it is impossible to completely avoid this effect when milling soft wood. This is obviously the best of our three attempts at cutting poplar plywood.



Fig. 4.13 Detail of the edges of the cut

4.3 Hardwood

Hardwoods such as oak, birch, or sycamore are much stronger and more resilient than softwood. Hardwood has to be milled using medium cutting speeds and higher feed rates than softwood but is generally just as easy to work on.

Machine Data	
Feed per tooth f_z	0.1-0.1 mm
Cut depth a_p	0.5-3mm
Cut Width a_e	Up to 50 per cent of the tool diameter where possible
Plunge Rate	5-10mm/s
Milling Type	Downcut wherever possible
Tools	
# of Flutes	1 or 2
Spiral Type	Right- or left-hand
Material	Solid carbide
Coating	Not necessary
Cooling	Air cooling preferable, but manageable without

Table 4.2 Basic cutting data for hardwood

4.3.1 Example #1: Solid Beech (right-hand spiral)

We selected a non-coated 1 mm 2-flute right-hand spiral fish-tail end mill. Once again, we had no specifications to work with, so we used the same basic settings as we did in [section 4.2.3](#), with a spindle speed of 25,000 rpm and a feed rate of 10 mm/s.



Fig. 4.14 Our first attempt at milling solid beech

The result is generally fairly good, but the frayed edges spoil the overall effect. This can be improved by using a left-hand spiral mill.



Fig. 4.15 Detail of the edges of the cut

4.3.2 Example #2: Solid Beech (left-hand spiral)

For our second beech workpiece, we used a left-hand spiral fish-tail mill and the same settings as for the previous attempt (see [section 4.3.1](#)).



Fig. 4.16 *This is about as good as it gets*

You can see that the edges no longer tear and fray, although the spindle speed was still a little high. The curve of the letter “b” and the top-right edge of the “u” are both slightly charred, but reducing the spindle speed by 1,000–2,000 rpm should rectify this issue.



Fig. 4.17 *Detail of the edges of the cut*

4.4 Perspex and Polycarbonates

Polymethyl methacrylate (PMMA) is a thermoplastic and is widely known under the trade names Perspex and Plexiglass. Polycarbonate (PC) thermoplastic polymers have similar attributes. The best-known

commercial polycarbonate is Makrolon. Both materials are available in completely transparent, colored, and satin-finished versions.

Perspex becomes soft at 105° Celsius (221° F), while polycarbonate softens at 140° C (284° F). Like softwood, both have to be machined at high cutting speeds and feed rates. Incorrect parameters will quickly produce temperatures that melt the material and clog the tool, leading to lateral pressure that causes the tool to break away or snap.

Machine Data	
Cut Depth a_p	Up to 3 mm
Cut Width a_e	Where possible up to 50 per cent of the tool diameter
Feed per Tooth f_z	0.04-0.07 mm
Plunge Rate	5-10 mm/s
Milling Type	Up- or downcut
Tools	
# of Flutes	1
Spiral Type	Right-hand (left-hand for thinner materials)
Material	Solid carbide
Coating	Not necessary
Cooling	Air cooling, water cooling, or underwater machining. Also practicable without additional cooling.

Table 4.3 Basic cutting data for Perspex and polycarbonates

4.4.1 Example #1: Perspex

We used a 1 mm right-hand spiral non-coated mill. The manufacturer quotes feed per tooth as 0.04-0.065 mm per revolution at 32,000 rpm. Our Kress spindle has a maximum speed of 29,000 rpm, so we selected this (setting #6) and a feed-per-tooth value of 0.05 mm.

This gives us feed of 1,450 mm/min (about 24 mm/s) and a cutting

speed of about 91 m/min.



Fig. 4.18 *Our Perspex milling result*



Fig. 4.19 *Detail of the edges of the cut*

The result is clean and, even using a magnifier, it is hard to detect any flaws. Only using a microscope were we able to detect discrepancies in the order of around 0.1 mm. These were caused by the high feed rate, which made the tool skip slightly out of its prescribed path. Braking to change direction led to slight overcutting in the corners.



Fig. 4.20 The top right-hand corner of the letter “u”, viewed under a microscope

4.5 Extruded Foam

Polyvinyl chloride (PVC) and Styrofoam are very soft materials available in a variety of colors. They need to be cut at slow speeds and high feed rates.

Machine Data	
Cut Depth a_p	Up to 5 mm
Cut Width a_e	Where possible up to 50 per cent of the tool diameter
Feed per Tooth f_z	0.05-0.2 mm
Plunge Rate	5-20 mm/s
Milling Type	Up- or downcut
Tools	
# of Flutes	1
Spiral type	Right-hand (left-hand for thinner blanks)
Material	Solid carbide
Coating	Not necessary
Cooling	Air cooling, water cooling, or underwater machining. Also practicable without additional cooling.

Table 4.4 Basic cutting data for extruded foam products

4.5.1 Example #1: PVC

We used the same 1 mm right-hand spiral tool that we used for our Perspex example (see [section 4.4.1](#)). We set the spindle speed to 17,000 rpm (setting #3) and feed per tooth to 0.1 mm.

This gives us feed of 1,700 mm/min (around 28 mm/s) and a cutting speed of around 53 m/min.

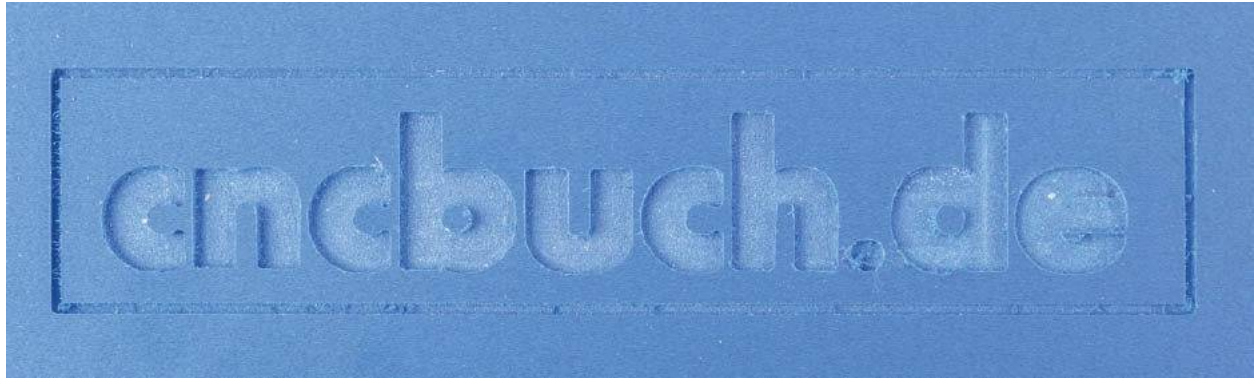


Fig. 4.21 Our PVC milling result

Once again, we produced a clean result with no melted edges. The tool was neither clogged nor discolored.



Fig. 4.22 Detail of the edges of the cut

Our microscope image shows no significant flaws. The consistent texture of the extruded foam makes it possible to mill it without leaving traces of the tool's shape in the sides or the bottom of the cut.

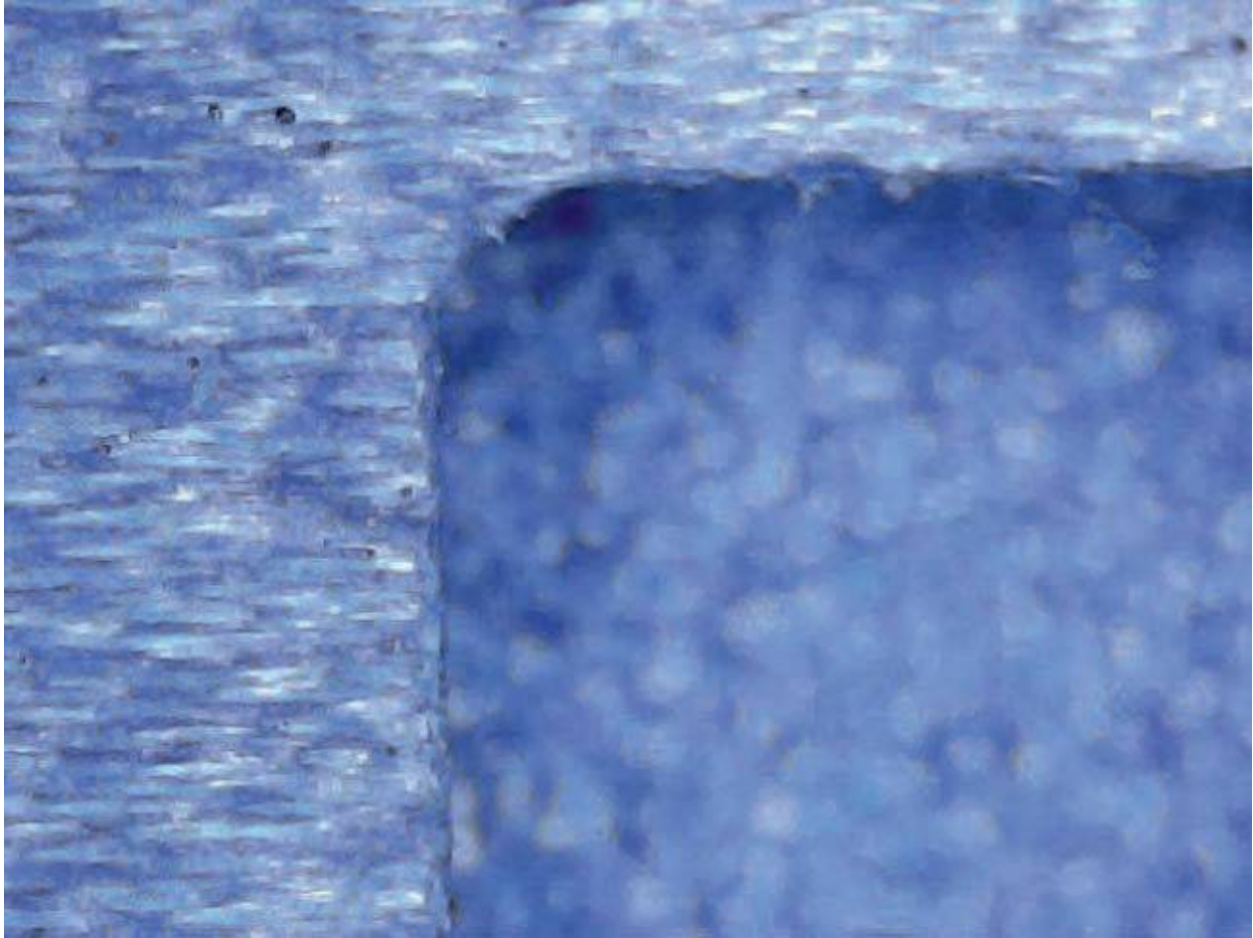


Fig. 4.23 The top right-hand corner of the letter “u” viewed under a microscope

4.6 Engraving Plastic

Plastic engraving material is popular for creating signs and is made of two different-colored sheets of bonded plastic. Milling the upper sheet reveals the color of the lower sheet through the cut.

The upper sheet is usually only a few tenths of a millimeter thick, requiring just a single pass with a graver or conventional end mill. The process is generally the same as for extruded foam.

Machine Data	
Cut Depth a_p	0.1-0.4 mm
Cut Width a_e	Doesn't need to be set
Feed $v_{f,x,y}$	0.4-1.5 m/min. The greater a_p , the lower the feed value
Plunge Rate	5-20 mm/s
Milling Type	Up- or downcut
Tools	
# of Flutes	Graver with a 30-120° tip
Spiral Type	n/a
Material	Solid carbide
Coating	Not necessary
Cooling	Air cooling preferable, but practicable without additional cooling

Table 4.5 Basic cutting data for engraving plastic

The angled tip of the graver means that the cut gets wider the deeper the tool plunges into the material. Accordingly, the spindle speed increases with shallower cuts and can reach as much as 60,000 rpm or more for a sample cut diameter of 0.1 mm. Hobby-grade spindles simply don't provide this kind of speed, so you will have to stick to values below the theoretical optimum.

4.6.1 Example #1: Engraving Plastic

We used a 60° Sorotec LGS.0060 VHM graver at a cut depth of 0.4 mm, which equates to a cut width a_e of 0.6 mm. The manufacturer quotes a required spindle speed of 45,000 rpm, but our Kress spindle has a maximum speed of 29,000 rpm, so we used this with a feed setting of 0.6-1.0 m/min.

In spite of the low spindle speed, the lowest feed rate of 10 mm/s gives us a satisfactory cutting speed. In order to remove all the required

material, the tool diameter had to be set to 0.6 mm in WinPC-NC.



Fig. 4.24 The sign we made from engraving plastic

The result is highly satisfactory with no flaked edges. Due to the graver's very fine tip, the bottoms of the cuts show slight scoring.



Fig. 4.25 Detail of the edges of the cut

4.7 Aluminum

Aluminum is a versatile favorite among model makers and can be equally well machined in sheets or solid blocks. It is highly rigid and stable at

high temperatures, making it ideal for use in load-bearing components in lightweight structures.

Milling aluminum demands much more from the machine than milling wood or plastic. Pure aluminum is quite soft and tends to build up easily on cutting edges and in slots, which hinders chip evacuation and reduces the quality of the cut. Aluminum alloys made with copper, magnesium, manganese, silicon, or lead alter the base attributes of the metal, making it harder or more resilient, or changing its weldability and anodization characteristics.

Some aluminum alloys with good machinability attributes are:

- AlMg4.5Mn
- AlCuMgPB
- AlMgSiPb
- AlCuMg1

Machine Data	
Cut Depth a_p	Up to 0.5mm
Cut Width a_e	Where possible up to 50 per cent of the tool's diameter
Feed per Tooth f_z	0.001-0.01 mm
Plunge Rate	0.5-1 mm/s
Milling Type	Up- or downcut
Tools	
# of Flutes	2 or 3, or dedicated single-flute aluminum mills
Spiral Type	Right-hand (left-hand for thin workpieces)
Material	Solid carbide
Coating	TiN, TiAlN, or non-coated
Cooling	Minimum Quantity Lubrication (MQL) or using water/methylated spirit solution

Table 4.6 Basic cutting data for aluminum

4.7.1 Example #1: Aluminum

We used a 1 mm 2-flute fish-tail end mill to machine a AlCuMgPB workpiece. We set the spindle to 29,000 rpm (setting #6) to give us a cutting speed of around 90 m/min. This is much too slow for this material, which actually requires a cutting speed between 370 m/min and 1,500 m/min.

The thin mill and the low spindle speed make it impossible to select an appropriate feed rate. The machine has its limits, too, and we had to keep lateral forces to a minimum to keep the setup stable. Values of 1–2 mm/s (60–120 mm/min), which equate to feed per tooth of 0.002 mm provided adequate results.

In contrast to our wood and plastic examples, aluminum requires a low plunge depth and, due to the thin mill, a small cut-depth per pass of 0.2 mm. Thicker mills can be used to machine up to 2 mm of material in a single pass. When milling aluminum, it is essential to keep the plunge rate low (we used a value of 1 mm/s). These settings make milling a slow process, and our sample workpiece took about four and a half hours to finish.



Fig. 4.26 The finished aluminum sign

The result is very good. The bottom of the cuts might have been smoother if we had used a flat-nosed mill, although the detail view in [figure 4.27](#) shows that the ridges have only a very slight depth of one- or two-hundredths of a millimeter. All of the edges are excellent.



Fig. 4.27 Detail view

Although softer materials exert less force on the machine and theoretically have fewer issues with accuracy, the parameters we used here produced a finished workpiece measuring 99.96×20.00 mm, which is extremely good for a model that measures 100.00×20.00 mm.

We could perhaps have accelerated the process by using a larger cut-depth per pass of 0.3–0.5 mm. The size of the increment you can use depends on the precision of the spindle, the amount of play in the machine, and the quality of the tool you use.

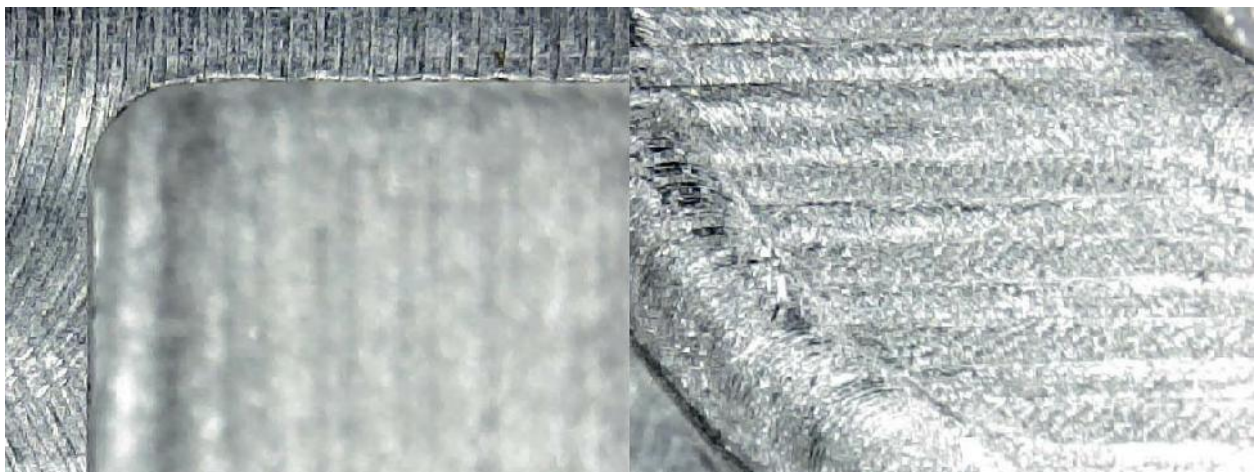


Fig. 4.28 Detail of our aluminum workpiece viewed under a microscope

The microscope image in [figure 4.28](#) shows clean, straight edges, and the bottom of the cut for the second “c” (which looks quite wavy) is actually almost perfectly flat. The texture you can see is due to the alternating direction of the passes made by the tool during the cut.

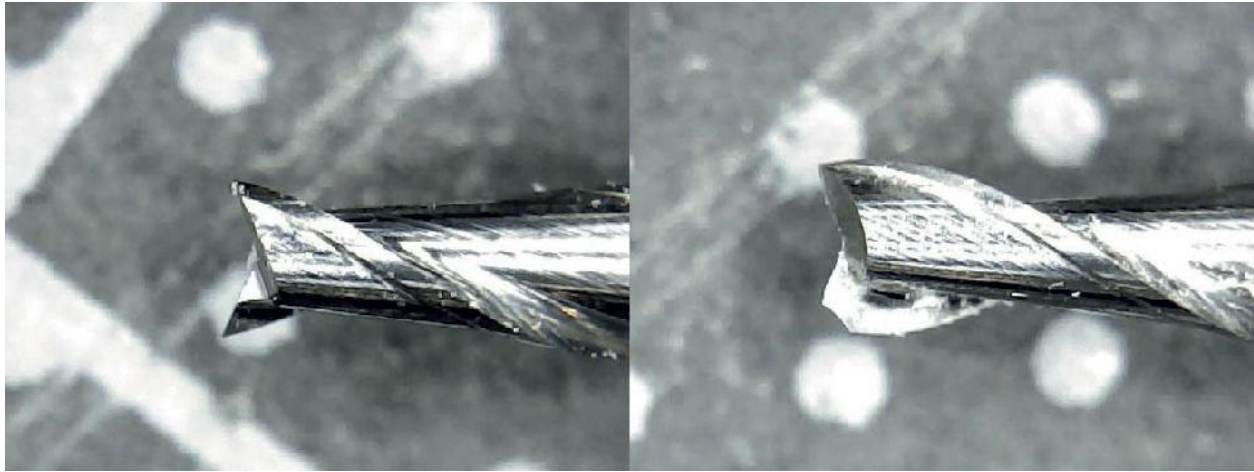


Fig. 4.29 The tool before and after our test

Our end mill completed the job without reducing cutting quality along the way, and finished up milling the lower third of the second letter “c.” Viewing the mill we used under a microscope shows that it is worn out and no longer fit for use. However, the lack of edge build-up nevertheless underscores the quality of the material and the usefulness of the parameters we used. A 0.15 mm piece has broken off one of the teeth, while the other one is quite blunt. When considering the state of the end mill, don’t forget:

- We used a cheap US\$5 end mill
- We dry-machined the workpiece without any additional lubrication
- The model came from a plot file that was imported directly into WinPC-NC without using an intermediate CAM process. The cutting parameters produced a large number of plunges that the tool is not designed for. When machining solid aluminum, entry points are usually pre-drilled or machined using an angled ramp to enter

the workpiece. The latter method involves additional complex CAM processes.

- Cutting speed wasn't ideal
- We didn't use a roughing mill first
- The 0.2 mm per pass cut depth only used the fine tips of the mill's cutting edges. The frame in the pattern (without the text) involved milling a distance of 2.4 m, and the finished piece comprises a total distance of more than 10 m. In view of the suboptimum conditions, the end result is excellent.

This example shows that a hobby-grade gantry router can be used to produce accurate results, even with tricky materials.

4.7.2 Example #2: Facing an Aluminum Blank

Face-milling techniques can be used to remove the entire surface of a workpiece in order to make it smooth and flat, and to unify the distance between the workpiece and the tip of the mill. Once you have milled one side flat, you can clamp the workpiece with the smooth side down and machine the other side with perfect parallel accuracy.

Because the entire surface is being milled, the diameter of the tool is largely irrelevant, although the broader it is, the cleaner the surface finish will be. We used a 6 mm flat-nosed VHM mill made by HOLEX. We set the spindle speed to 21,000 rpm (setting #4), which gave us a cutting speed of about 400 m/min. We selected a cut-depth per pass of 0.1 mm and a feed rate of 240 mm/min (4 mm/s). This equates to feed per tooth of 0.01 mm. Higher feed rates cause significant chatter. In situations like this, you can increase the cut depth to anything between 0.5 mm and 2 mm, but you have to reduce the feed rate correspondingly to about 2 mm/s. To ensure a high-quality finish, the final pass should be executed with a depth of no more than 0.1 mm–0.2 mm.

Before we began milling, we checked that the workpiece was clamped at an even height in the center and at all four corners. If you discover

obvious height discrepancies when clamping a workpiece, set the Z-axis zero point to match the highest point and mill your workpiece flat in multiple passes until the entire surface is level.



Fig. 4.30 A face-milled aluminum bar

To avoid having to plunge the tool into the surface, we made the model wider than the actual workpiece and started the spindle in the air beside it. To do this yourself, set only the Z-axis zero point as usual and move the tool to the side of the workpiece before saving the X and Y coordinates.

Our aluminum blank turned out great, with a smooth surface and only slight waviness. The texture is due to the alternating direction of the tool and the fact that the passes overlap by about 50 per cent. Depending on which CAM software you use, you can adjust the order of the passes to produce varying surface patterns.

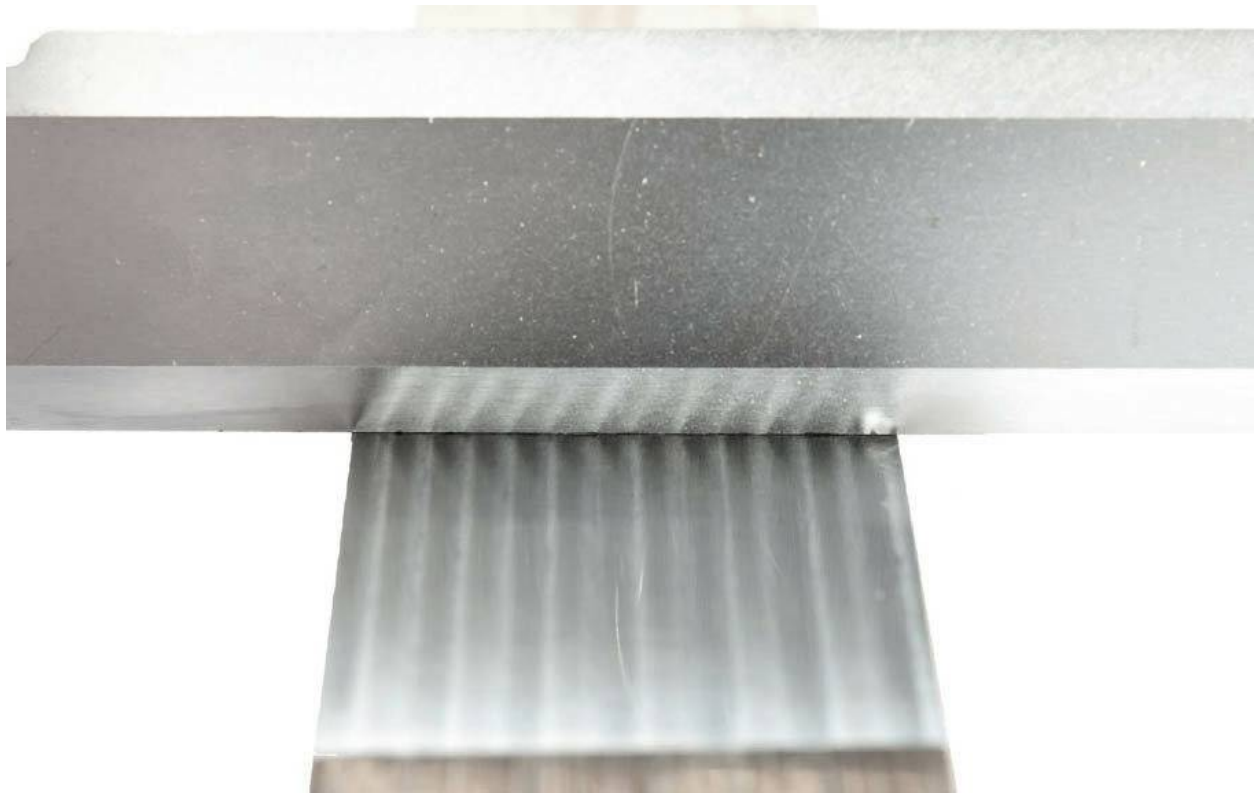


Fig. 4.31 *Very slight waviness*

The detail image made under a microscope shows slight scoring. The distance between each pair of lines equates to the feed-per-tooth value of $1/100$ mm. The “seam” at the lower edge of the image is where two opposing passes meet.

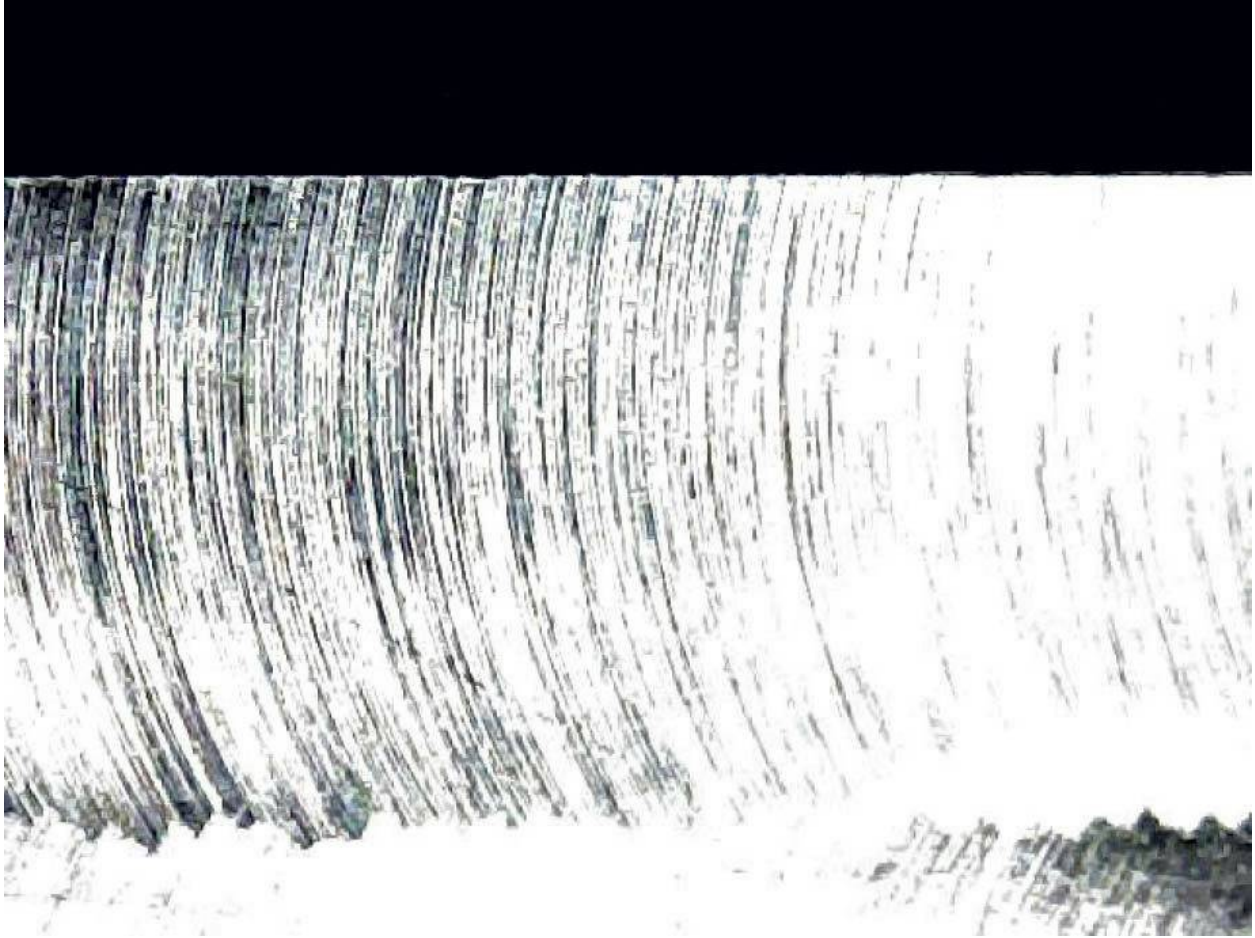


Fig. 4.32 The surface of our face-milled aluminum blank viewed under a microscope

4.8 Brass

In everyday situations, brass is most often used to make signs or accessories for doors and other furniture. Thanks to its high luster when polished, model makers often use it when constructing locomotives and steam engines.

Brass is an alloy made largely of copper and zinc, but is also available in compounds that include lead, aluminum, manganese, tin, and other additives that alter the base characteristics of the alloy. Brass/lead alloys such as CuZn35Pb1 and CuZn39Pb2 are more easily workable than pure brass, which is very hard to mill. Make sure you know exactly what you are getting when buying brass.

Milling brass is more difficult than aluminum and takes forever if you

use a thin end mill. Like aluminum, many brass alloys produce edge build-up.

Machine Data	
Cut Depth a_p	Up to 0.2 mm
Cut Width a_e	Where possible up to 50 per cent of the tool's diameter
Feed per Tooth f_z	0.005-0.01 mm
Plunge Rate	0.5-1 mm/s
Milling Type	Up- or downcut
Tools	
# of Flutes	2 or 3
Spiral Type	Right-hand
Material	Solid carbide
Coating	TiN, TiAlN, or non-coated
Cooling	Water/methylated spirit solution

Table 4.7 Basic cutting data for brass

4.8.1 Example #1: Brass

We used a 1 mm 2-flute fish-tail end mill to machine a leftover workpiece made from CuZn37 (also known as Ms63.) This alloy is not particularly easy to work with, but can be milled successfully using appropriate parameters and a well set-up machine.

We set the spindle speed to setting #6 (29,000 rpm) to give us a cutting speed of around 90 m/min, which is about half of the required value. We selected feed per tooth of 0.001 mm to give us a feed rate of 87 mm/min (1.45 mm/s), and a cut-depth per pass of 0.1 mm. Earlier tests showed that we could have used a 0.2 mm setting, but increasing it to 0.3 mm caused this particular tool to break after just three passes.

Milling our model would take 14–15 hours using the selected settings,

so we reduced the final cut depth to 1 mm, which reduced the job time to around seven hours.

Cooling

You should definitely use some kind of cutting fluid when milling brass. There are many ready-made fluids available, such as the range sold by Tap Magic (www.tapmagic.com), although home-made mixtures of water with methylated spirit or household detergent work well too.

Apply small amounts of your chosen fluid regularly to the cut using a syringe or a soft brush. Applying cutting fluid extends the life of your tools and prevents waste from sticking to the mill and the workpiece. Waste particles are clearly visible on the upper edge of the letter “u” in [figure 4.34](#).



Fig. 4.33 *The finished brass nameplate*

We didn’t smooth our blank before we milled it. The outlines turned out very nicely, and the bottom of each cut is of a similar quality to that in our aluminum sample. However, overall accuracy is lower than that in the aluminum piece—for example, the upper curve in the letter “b” is less accurate at the 45° position, where the tool has obviously slipped off course.

This example serves mainly as a comparison with our other sample materials, and most people engrave rather than mill brass. Generally speaking, you should smooth brass nameplates before you engrave them, as polishing them later often spoils the overall finish.

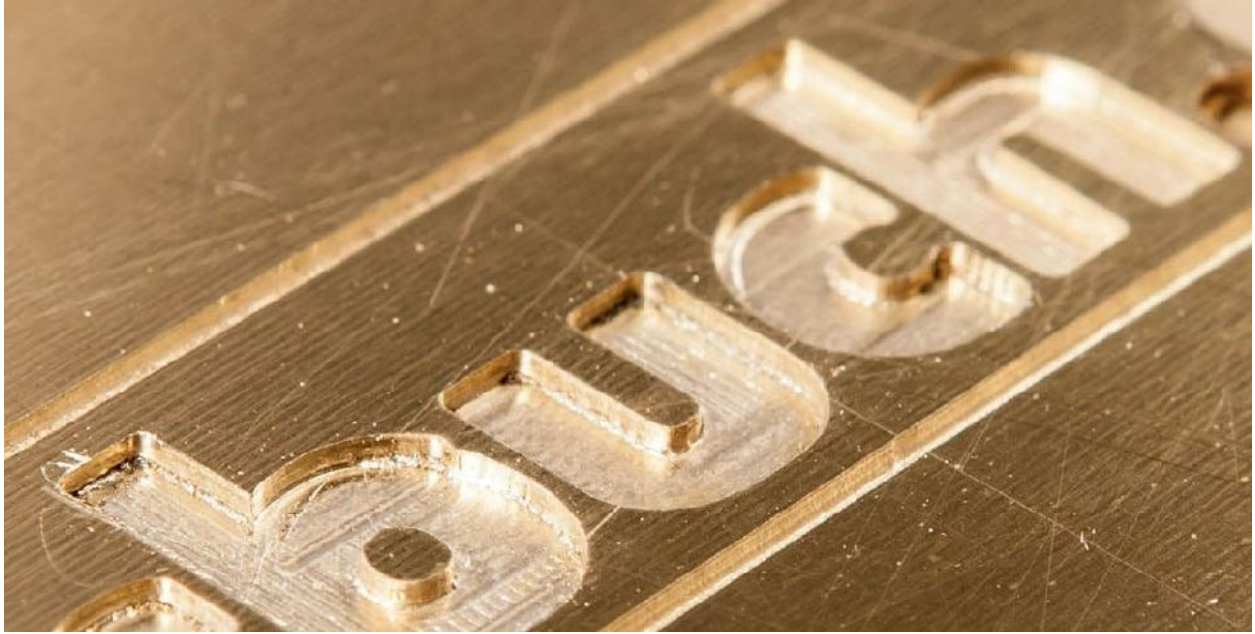


Fig. 4.34 Detail of the edges of the cut

The microscope view in [figure 4.35](#) shows the damage this job caused to the mill. This degree of deterioration is normal for a 1 mm mill, and equates to about 5/100 mm of wear.

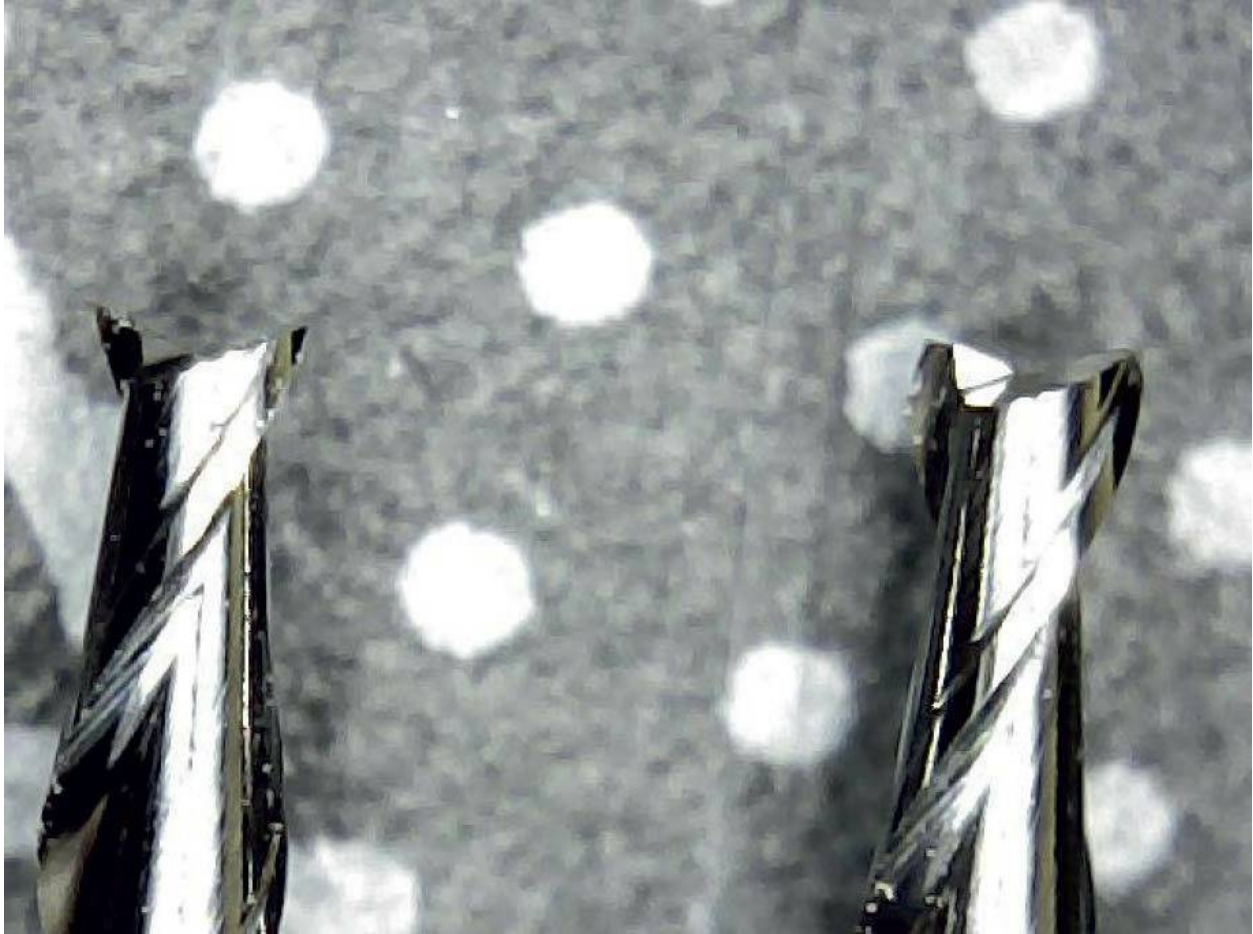


Fig. 4.35 The new end mill (left) and the same tool following work on our brass blank

4.9 Aluminum Dibond

Dibond is a composite material with outer layers made of aluminum and a core of plastic or mineral filler. The aluminum facing gives Dibond a solid look, but it is a lot easier to work with than solid aluminum.

Although the two materials have different requirements with regard to tool types and cutting speeds, it is still quite easy to machine a composite sheet as long as you make sure that neither the aluminum nor the core melt and clog the mill.

The outer layers of our Alucobond sample are only 0.5 mm thick and are made of AlMg1, which is pretty soft, so it was no problem to machine all the way through the sheet in a single pass. You can machine the top layer only (as we did with our engraving plastic example) to leave the core

showing through. However, Dibond core is not particularly attractive, and the material is designed for use in one piece, often as a carrier for photo prints or in the facades of buildings.

Machine Data	
Cut Depth a_p	0.1-2 mm
Cut Width a_e	0-100 per cent
Feed per Tooth f_z	0.005-0.05 mm
Plunge Rate	0.5-1 mm/s
Milling Type	Down- or upcut, according to the manufacturer's specifications
Tools	
# of Flutes	1 or 2
Spiral Type	Right- or left-hand
Material	Solid carbide
Coating	TiN, TiAlN, or polished non-coated
Cooling	Suitable cutting fluid or a 2:1 water/methylated spirit solution

Table 4.8 Basic cutting data for aluminum Dibond

4.9.1 Example #1: Aluminum Dibond

We used a right-hand spiral 1 mm two-flute fish-tail end mill. We set the spindle speed to 21,000 rpm (setting #4), which gave us a cutting speed of about 66 m/s. The feed rate was 2 mm/s, giving us feed per tooth of 0.005 mm. Due to the relative softness of the material, we used the same approach as we did for softwood, milling in two passes of 1.5 mm followed by a 0.5 mm finishing pass. To cater for the relatively hard aluminum layer, we used a plunge speed of 1 mm/s.

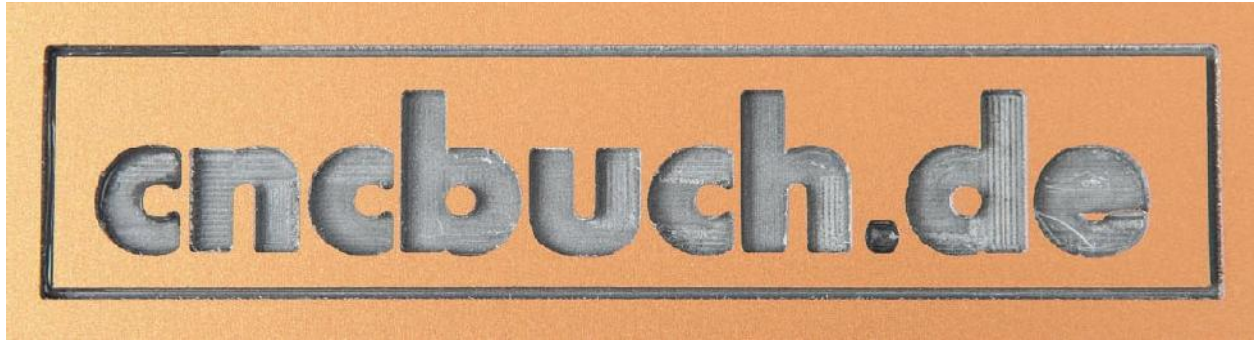


Fig. 4.36 Our nameplate cut into copper-colored Alucobond

The result is generally quite good, although the matte-gray core doesn't look too good, especially with all the visible grooves made by the tool. The edges of the cuts are clean, but the core material shows some fraying.

We followed the manufacturer's instructions and cut the frame anticlockwise, which meant the inner edge was a downcut and the outer edge an upcut. The inner edge is obviously cleaner than the outer one.

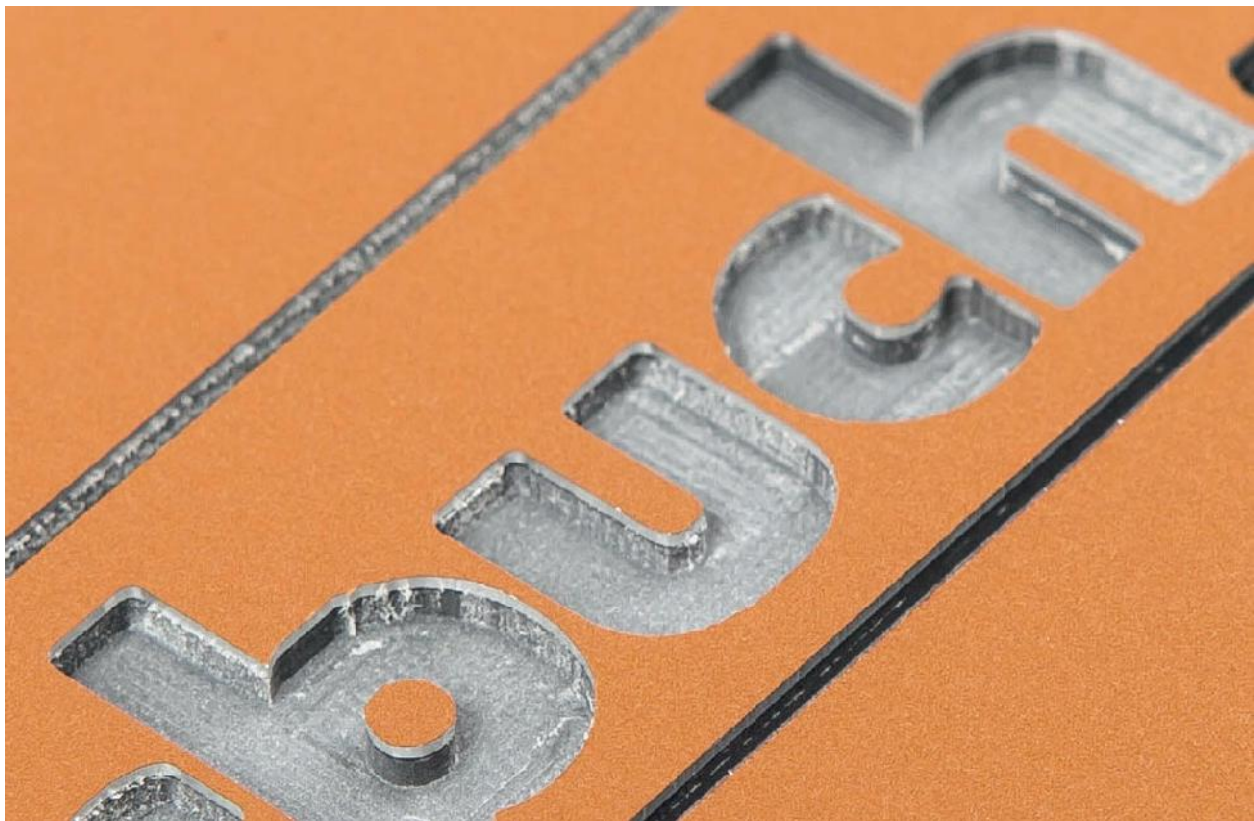


Fig. 4.37 Detail of the edges of the cut

The parameters we selected worked well and the mill remained largely undamaged.

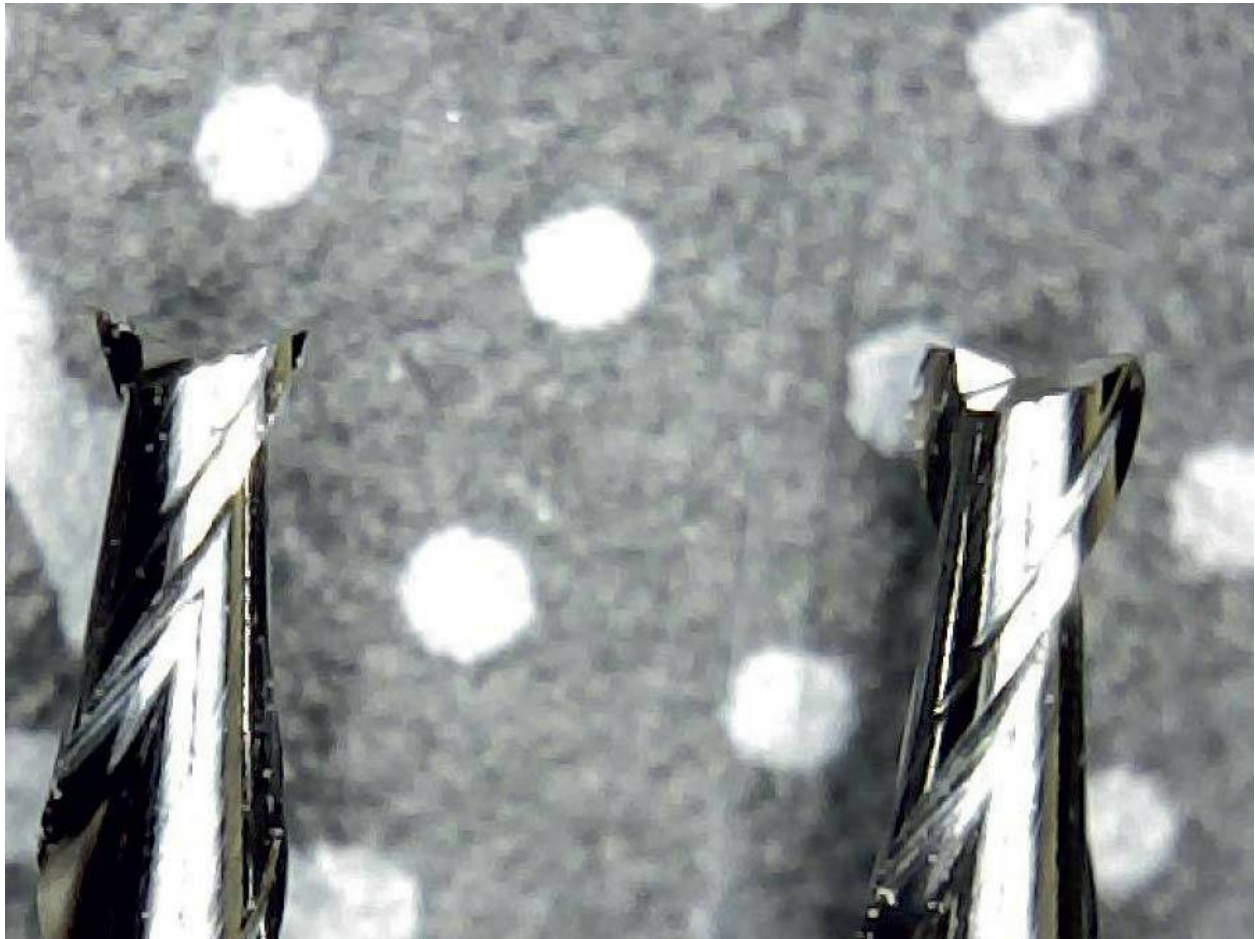


Fig. 4.38 The new mill (left) and following our Dibond job (right)

4.10 Glass

Popular opinion is divided on whether it is possible to mill glass and, if it is, whether the process involved is actually milling or more like grinding. Strictly speaking, you can't grind using a mill, as grinding involves using random abrasion rather than a geometrically defined cutting edge. However, grinding is still classed as a machining process.

In spite of all the debate, the simple answer is "Yes you can." Glass milling takes place every day in various industrial scenarios, especially in the production of optical glass. And yes, you can mill glass using a CNC

router.

Because glass is hard and brittle, any damage to its surface causes cracks that increase in size with every increase in applied force. The secret to successful glass milling is therefore to reduce tension in the material to a minimum by careful application of force from the tool and the clamping mechanism. This keeps cracks to a controllable minimum in an area directly surrounding the tool.

To avoid cracking, you will have to pre-drill your entry points or use a ramp to engage with the material. The low feed rate exerts very little force on the material, making it possible to mill right through thick glass in a single pass.

Cooling and chip evacuation are extremely important factors when milling glass. Glass chips have a similar effect to sandpaper and cause significant wear in the tool. Glass dust is bad for the machine, too, as it easily settles on the spindle and the bearings and causes lasting damage. And don't forget that glass particles represent a serious health risk. When working on glass, overflow cooling or underwater milling is the method of choice.

We set up our job using the Fusion 360 software package and programmed it to make the tool engage laterally with the material. Our sample data are aimed exclusively at helping you select appropriate parameters.

Machine Data	
Cut Depth a_p	Full depth
Cut Width a_e	0-100 per cent
Feed per Tooth f_z	0.0005-0.001 mm
Plunge Rate	Pre-drill or lateral (i.e., not vertical) plunge
Milling Type	Down- or upcut, depending on the material manufacturer's specifications
Tools	
Flute Type	Spiral or micro-tooth
Spiral Type	-
Material	Solid carbide
Coating	Diamond or other crystalline coating
Cooling	Overflow or underwater

Table 4.9 Basic cutting data for glass

As an example, we cut part of a rectangle with 10 mm radiused corners from a piece of ordinary 4 mm window glass. We used a 2 mm VHM micro-tooth mill with a polycrystalline diamond coating. We set our spindle to 25,000 rpm (setting #5) at a feed rate of 30 mm/min (0.5 mm/s).



Fig. 4.39 A glazier couldn't do any better . . .

We attached our rough blank to a sheet of plastic using tape and glued it to the floor of our water bath. In the top left-hand corner of [figure 4.39](#), you can see the angled ramp where the mill engaged with the workpiece. The edges are very clean and show hardly any signs of the milling process.

5 Machine Upgrades

This chapter talks about machine upgrades that make your router simpler and safer to use, and more versatile, too. Which upgrades you prefer and any modifications you may make will depend on your budget and your personal requirements. The steps you take will also depend on which parts are available. Using non-standard parts or building your own accessories are great ways to reduce the cost of sprucing up your machine.



Costs

The money we have spent so far differs from the original estimate we made (see [page 46](#)) and we have overstretched our total budget by about US\$270. This is due mainly to the enclosure and the vacuum system. However, safety is key, and we don't want to give up on our project or compromise safety for the sake of such a relatively small sum.

- Enclosure for reducing noise and dust levels (US\$959)
- Vacuum system (US\$650)
- Aluminum T-slot machine table (already budgeted)
- Machine vise (US\$90)
- Clamps for the T-slot machine table (US\$85)
- Tool length sensor (US\$115)
- Underwater milling bath (US\$270)

If you only want to mill wood, carbon fiber, aluminum, and brass, the smaller Stepcraft-2/300 router with a spindle, a few mills, and a simple plastic water bath will do nicely at a total cost of around US\$1,600.

5.1 Enclosure

On its own, a milling machine makes a lot of noise and produces masses of dirt and dust. It will distribute fine dust all over your workspace and throw chips several feet in all directions. A sealed enclosure reduces noise and dirt issues. However, an enclosure alone without a vacuum system, won't stop fine dust and waste particles getting out and posing serious health risks.

An enclosure has to be solid and stable enough to prevent chips and broken tools from exiting the work area. The best materials to use for an enclosure are extruded aluminum bars and 6 mm Plexiglass sheets.

There are various aluminum strut systems available that vary in stability and price. Some systems require you to provide your own connectors, and to drill your own holes and threads, while others offer a complete system straight out of the box. I used ones from a regional

supplier who cut them to size for me. The struts themselves have built-in threads and are easy to join using purpose-built connectors. Stepcraft offers its own enclosure for the 2/600 machine for \$959.



Fig. 5.1 *An extruded aluminum strut*

Selecting the Best Struts

Extruded aluminum is quite heavy, so I selected the smallest struts I could find with slots large enough to accommodate 6 mm Plexiglass sheets. The enclosure was relatively simple to make but weighs more than 66 lbs, making it impossible to transport without help.

The struts I ordered didn't have pre-drilled threads, so I ended up making my own (a total of 24!) using an M8 thread cutter and a can of cutting fluid from a home improvement store.

The enclosure has to be large enough to completely surround the machine at the extreme ends of all its movements, as well as offering sufficient space for setup and cleaning. The plan of my enclosure measures 90 × 65 cm (35.4 × 25.6"), which is plenty for my *Stepcraft-2/600* machine, and it also offers sufficient length to accommodate the wiring. It is 60 cm (23.6") tall. The manufacturer of the aluminum struts offers a range of ready-made connectors, and I ended up using the following components to build my enclosure:

- 4× 90 cm (35.43") extruded aluminum struts
- 4× 65 cm (25.59") extruded aluminum struts
- 4× 56 cm (22.05") extruded aluminum struts
- 8× 3D cube connectors

The enclosure is completely sealed except for one of the long sides, which opens upward. I built the flap the same way as the rest of the enclosure with dimensions designed to fit tightly into the frame (i.e., the Plexiglass sheet measures 89.8 × 64.6 cm). I used 8 mm right-angle connectors instead of the 3D cube connectors I used for the corners of the frame and added hinges to the finished flap. The additional parts required for the flap are as follows:

- 2× 89.8 cm (35.36") extruded aluminum struts
- 2× 56.6 cm (22.28") extruded aluminum struts. The size is derived from the 64.6 cm size of the Plexiglass sheet minus 2× the width of the strut.
- 4× 2D connectors for 8 mm slots
- 2× 40 × 40 mm (1.6") die-cast hinges and mounting materials

I ordered plastic caps for the remaining holes in the cube connectors to prevent dust and dirt collecting there, too. I also ordered a selection of T-slot nuts and bolts for attaching lights, tool holders, and so on.

The total budget for the enclosure is as follows:

- 11.4 m (37.4') heavy grade 8 mm aluminum strut: US\$215
- 8 cube connectors with mounting materials and dust caps: US\$57
- 4 2D connectors for the flap: US\$10
- 2 hinges with mounting materials: US\$15
- Various T-slot nuts and bolts: US\$27

This adds up to a total of US\$324. You can of course reduce these costs by using lighter struts and less complex connectors.

The Plexiglass sheets have to fit tightly into the slots in the struts, but need to have some play at the edges to leave space for rubber seals. I ordered mine cut to size from an online supplier. Make sure you order sheets with dimensions that precisely fit the depth of the slots in your struts (mine are 12 mm deep). My shopping list looked like this:

- 1 84 × 49.8 cm (33 × 19.6") Plexiglass sheet (front)
- 1 92 × 67 cm (36.2 × 26.4") Plexiglass sheet 92 × 67 cm (top)
- 2 67 × 58 cm (26.4 × 22.8") Plexiglass sheet 67 × 58 cm (left and right ends)
- 1 92 × 58 cm (36.2 × 22.8") Plexiglass sheet 92 × 58 cm (back)

Pre-cut Plexiglass isn't cheap, and mine cost a total of US\$200. The complete enclosure, including the frame, the flap, some handles and other odds and ends cost a total of about US\$530, which is US\$130 more than I planned.

Putting it all together took about 20 minutes and provided me with a solid enclosure with nice thick windows.



Fig. 5.2 The finished enclosure, showing the mounted Plexiglass sheets and the front hatch

I attached the front flap with the hinges at the top. You can attach it to hinge downward if you want, but either way you need to add some kind of safety mechanism to stop it from falling shut. The flap is heavy and can easily break if it flops down.



Fig. 5.3 *The finished enclosure with the front flap open and the additional T-slot nuts marked in red*

Figure 5.3 shows two ready-mounted T-slot nuts. Most T-slot nuts can be mounted post-assembly and are great for attaching the enclosure to a table, or adding handles and locks for the flap.

Tool holders like the one shown in [figure 5.4](#) attach directly to T-slot nuts and are available cheaply from home improvement stores. An accessory like this ensures that you always have the right tool handy when you need it.



Fig. 5.4 An add-on tool holder

5.2 Vacuum System

Milling produces huge amounts of dust and waste chips. Larger chips are easy to remove using a conventional vacuum cleaner, whereas fine dust particles simply float up into the air, requiring specialized vacuum equipment to keep them in check.

Even if they don't present a health risk, fine dust particles get in every nook and cranny and settle everywhere, which can cause problems in computers and other electronic devices, as well as in the milling machine

itself. If you don't continually remove dust, it will build up in your workplace and continually fill the air.

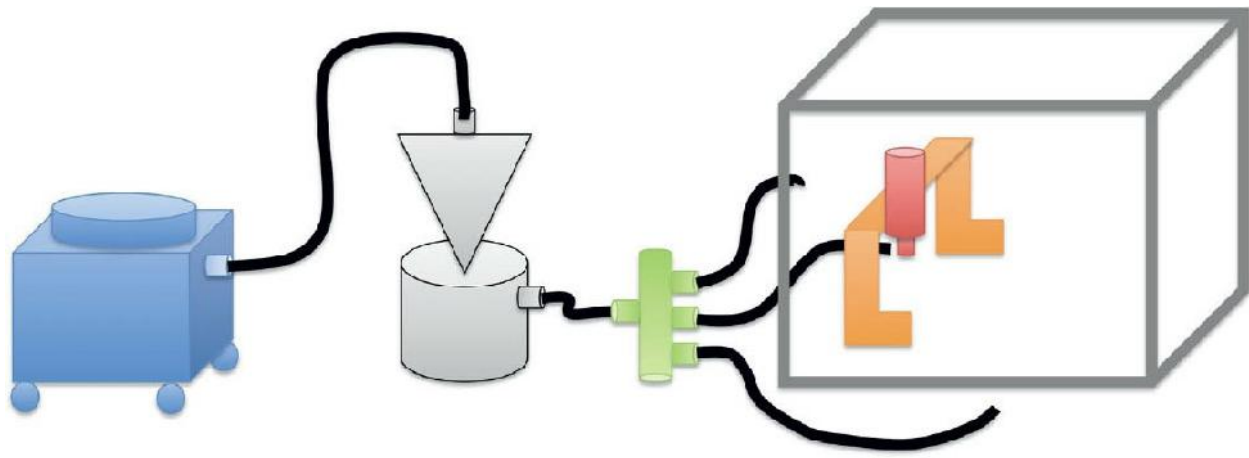


Fig. 5.5 *Schematic of a vacuum system*

Fire Hazards

Milling often produces high temperatures and sparks that can easily ignite the collected waste material. The oxygen in the vacuum's air stream works like a bellows on a bonfire and increases the risk of fire. It is essential to use a certified fireproof vacuum system.

A vacuum system that removes waste directly from the tool tip and from within the machine enclosure will significantly reduce dust issues. To remove dust from the enclosure, all you need is a simple tube. However, if you have to remove large amounts of waste, most vacuums quickly lose suction power. To prevent this from happening, we added a centrifugal (or “cyclone”) separator that diverts most of the waste material to a separate container.

Two tubes remove dust and chips from the tool tip and the enclosure, while a third tube with a suction nozzle removes additional waste where necessary. The system also requires some kind of valve to block any tubes not currently in use.

5.2.1 Centrifugal (Cyclone) Separator

Centrifugal (“cyclone”) separators produce a vortex that creates excess inertia in the dust particles contained in the airstream, causing them to

collect on the walls of the separator and fall into the attached container instead of getting sucked directly into the vacuum. This type of system can reduce the amount of dust reaching the vacuum by up to 99 per cent. You can purchase a ready-made separator such as the *Dust Deputy* (see [figure 5.6](#)) or build your own.



Fig. 5.6 *The main parts of a cyclonic separator*

The lower open end of the cyclone is placed on a simple bucket and lid purchased from a home improvement store. The lid I bought was too flexible, so I strengthened it with a sheet of plywood that I stuck in place using silicone. By the way, I cut the plywood to size in a couple of minutes using my CNC machine.

Suction is applied from the top and the dirty air enters the separator from the side. The setup shown here cost a total of about US\$130.

5.2.2 Vacuum and Connectors

Your vacuum needs to be certified for at least Class L dust extraction and should be designed to run continuously. Appropriate models cost US\$320–400. “Class L” means that a maximum of 1 per cent of the dust entering the system makes it through without being separated. Used with a cyclone separator, such a system is a really effective way to reduce the amount of dust and waste in the air. However, it is still not sufficient for jobs that involve flammable or carcinogenic materials, for which you require additional safety measures such as a specialized vacuum, a dedicated dust mask, or the use of an underwater milling setup.



Fig. 5.7 A Makita VC13010LX1 Class L vacuum

Alongside milling applications, a high-quality vacuum can also be used with keyhole and circular saws or grinding machines. If your vacuum is also suitable for use in wet environments, you can use it for all sorts of things, including carpet cleaning!

Further up the scale, vacuums certified for Class M or H dust extraction provide even better protection, but can easily cost two or three times as much as a lower-grade model. This is way outside our budget, so we stuck to the system described above.

Selecting a suitable vacuum with appropriate certification requires quite a bit of searching through product data sheets and manuals. Many industrial vacuums cannot be regulated and only work at full power, which is usually too much for use with a CNC router, and costs money while creating too much noise. If your vacuum cannot be regulated, you need an airflow regulator to keep the suction level in check.

We chose a variable-speed Makita VC1310LX1 wet/dry vacuum. This model is compact and is certified to Class L. It is supplied with a variety of accessories and has an automatic start/stop mechanism that switches the vacuum on and off in sync with an attached milling spindle. The price for all this convenience is around US\$400.

5.2.3 Dust Extraction in the Enclosure and at the Tool

Stepcraft sells a dedicated exhaust adapters for US\$99 (see [figure 5.8](#)), which attaches directly to a pre-drilled hole in the spindle holder and is ready for use in a couple of minutes. A washer fitted between the body of the adapter and the spindle ensures that there is no unwanted contact between the spindle and the fixing screw. The adapter is supplied with a 2-meter (6-foot) tube.

A magnetically attached brush ensures consistent dust removal but can only be used for a limited range of cut depths. At depths greater than 8–10 mm, it collides with the workpiece and causes the spindle motor to miss steps. For this type of cut, you have to remove the brush before you begin.



Fig. 5.8 The Stepcraft exhaust adapter for the Kress 800 FME spindle

The following connectors are required to set up the vacuum, the separator, and the distributor (shown in green in [figure 5.5](#)):

- The main hose has a conical connector with an outer diameter of 44–46 mm (1.73–1.81"). This has to be connected to the upper hole in the cyclone, which has a diameter of 48.5–54 mm (1.90–2.13"). To make the connection, we need a 50–45 mm (2-to-1¾") reducer.
- The dust-in side of the cyclone also has a diameter of 48.5–54 mm (1.90–2.13") and has to be connected to the distributor, too.
- The tube from the spindle exhaust adapter has an inside diameter of 26 mm (1.02") and has to be attached to the distributor.
- It makes sense to use 26 mm tubes for the other two outlets, too.



Fig. 5.9 *The finished distributor*

Two-inch (50 mm) plastic drain piping is ideal, and we connected ours to the cyclone using the rubber couplings shown in [figure 5.9](#). Adapters and reducers are available in a range of sizes if you need to use different-sized tubes.



Fig. 5.10 Airflow regulator

An unused outlet can be simply capped off or used with an adjustable airflow regulator. Our second vacuum outlet for the enclosure proved unnecessary, and hardly any dust collected in the enclosure without it. However, it is useful to have an extra outlet should you end up using a larger or more complex machine later on.

At a total cost of around US\$650, our vacuum system ended up well over our original US\$400 budget. At this point, we had only US\$330 left for all the other accessories, although the T-slot machine table mentioned in [section 5.3](#) was accounted for in the original purchase price of the machine.

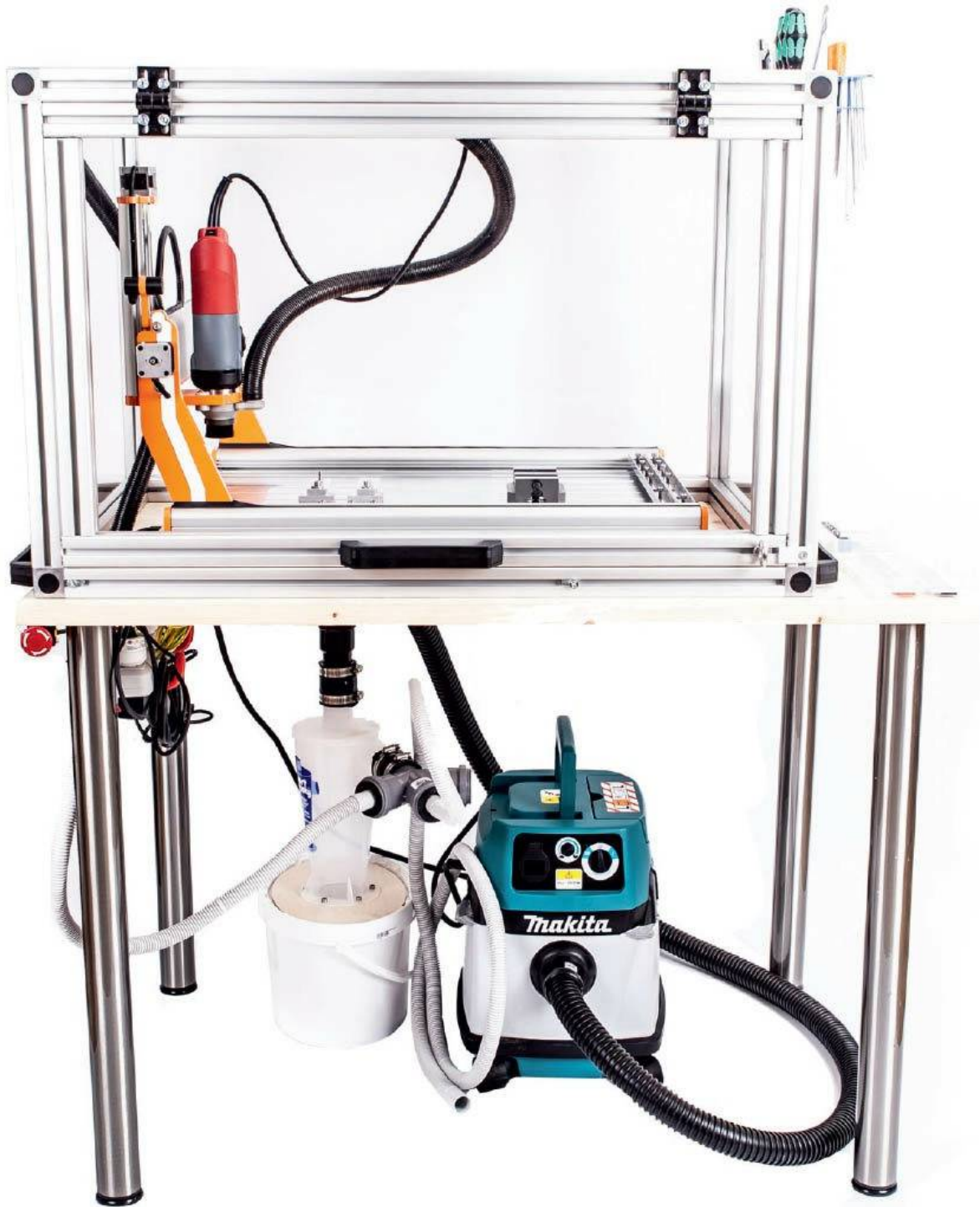


Fig. 5.11 The finished enclosure showing the vacuum system with its separate manual vacuum nozzle

5.3 Aluminum T-slot Machine table

The HPL (High Pressure Laminate) machine table we have used so far now needs to be replaced with a T-slot machine table that is more rigid and offers versatile clamping options using the hold-down clamp supplied with the machine.



Fig. 5.12 The mounted T-slot machine table

The T-slot machine table is mounted the same way as the original HPL table. To do this, loosen the two bolts circled in green in [figure 5.12](#), remove the front panel and slide out the HPL table. If this proves difficult, or if you have trouble inserting the new machine table, loosen the front and rear panel bolts (see [sections 2.11.3](#) and [2.11.5](#)) and realign the lead screws once you have finished mounting the new table.

The T-slot machine table is greased for protection during transport, so you need to clean it thoroughly before use to avoid getting grease on your skin or in your eyes. Once you are done, put the front panel back in place and tighten the bolts.

5.4 Clamps and Clamping Blocks

As soon as you begin milling, you will quickly find you need a wide variety of clamping devices to keep your workpieces still while you work. The clamps supplied with Stepcraft machines are best suited to clamping relatively flat blanks and work using vertical rather than lateral pressure.

Hold-down clamps and clamping blocks are a great combination to start out with. Hold-down clamps are fixed in place using square or hexagonal sliding nuts and, depending on the type of clamp, exert vertical and/or lateral pressure on the workpiece. Clamping blocks are used to exert additional lateral pressure and prevent a workpiece from slipping sideways while you work. Drilling slots in your clamps makes it easier to adjust them to get a perfect fit with your workpiece.

A separate screw at the end of the clamp determines the vertical distance between the clamp and the machine table (i.e., the height of the workpiece) and ensures that the clamp lies level on the workpiece without damaging it. A sliding nut is then used to fix the other end of the clamp in place above the workpiece.

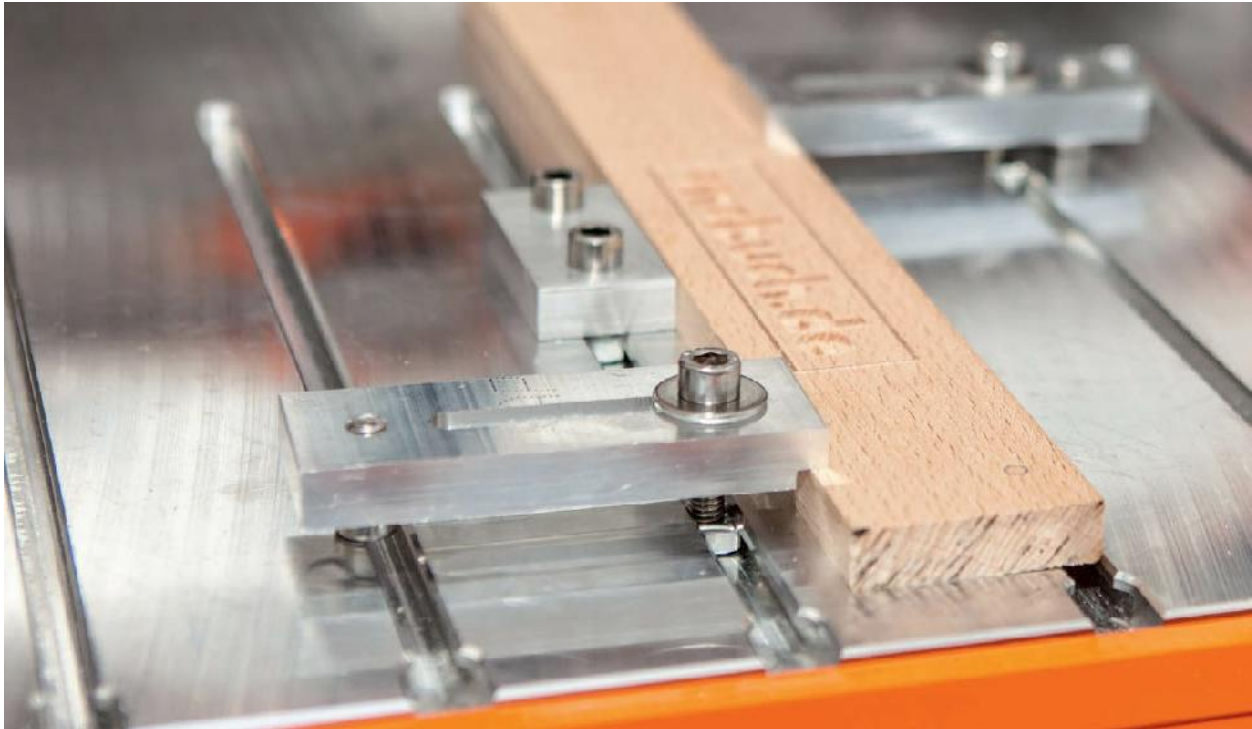


Fig. 5.13 A workpiece held in place using two clamps and an additional block

Our clamps and blocks are homemade from 80×10 mm (3.15×0.39 ") and 50×10 mm (2×0.39 ") AlCuMgPb aluminum blanks.

To begin, we cut our blanks to size (two $80 \times 10 \times 25$ mm for the clamps and four $50 \times 10 \times 20$ mm for the blocks). If you make your own clamps, the more accurately you prepare the blanks, the less effort it will be to finish them later.



Fig. 5.14 Facing the aluminum blanks

If you don't have a suitable power saw, you can either have the blanks made at a machine shop or saw them by hand and mill them smooth. The precise dimensions are not important, but it is essential that all the faces of a clamp or block are at precise right angles to one another. This enables you to use both as blocks and prevents damage to the workpiece due to uneven edges.

We milled 50×6.5 mm ($2 \times \frac{1}{4}$ ") slots in the clamps using a 6 mm two-flute right-hand spiral flat-nose Horex mill. We set the spindle to 21,000 rpm (Kress setting #4), cut depth per pass to 0.5 mm, and the feed rate to 1 mm/s. A slow plunge rate of 0.5–1.0 mm/s is important for this type of job.

You only need a 2D model to mill this kind of slot. We loaded our file in Fusion 360 and labeled the cut as a 2D pocket (see [section 6.3](#)). The best way to hold the blanks in place is using a machine vise like the one shown in [figure 5.14](#). For a job like this, hold-down clamps and a spoilboard don't provide sufficient grip.



Fig. 5.15 *Milling the slot in one of our clamps*

Positioning the tool can be tricky, although it isn't critical if the slot isn't located precisely in the center of the clamp. To determine the zero point, proceed as follows:

1. Load the model in WinPC-NC
2. Move the tool to the location where you want your slot to start and leave it about 2 mm above the workpiece
3. The zero point of the model lies at the intersection of the two border lines of the slot. Because the slot is 6.5 mm ($\frac{1}{4}$ ") wide, you need to move the tool 3.25 mm ($\frac{1}{8}$ ") to the left, which is easy to do using the software's *Manual movement* tool. Once the tool is in position, save the new location using the *Save To → Zero Point XYZ* command.
4. Use WinPC-NC to zoom right in to one end of the slot. Click the *Move machine to destination point* button, and then click on the middle of the slot.

5. Lower the tool to about 0.2 mm above the surface and measure the distance to both sides. Now correct your new zero point until the tool is at the center of the blank.
6. The start of the slot is defined the same way. Once you have defined the workpiece zero point, use *Move machine to destination point* to navigate to the center and the start of the slot, and re-check the position of the tool.
7. Set the Z zero point as usual and save it.

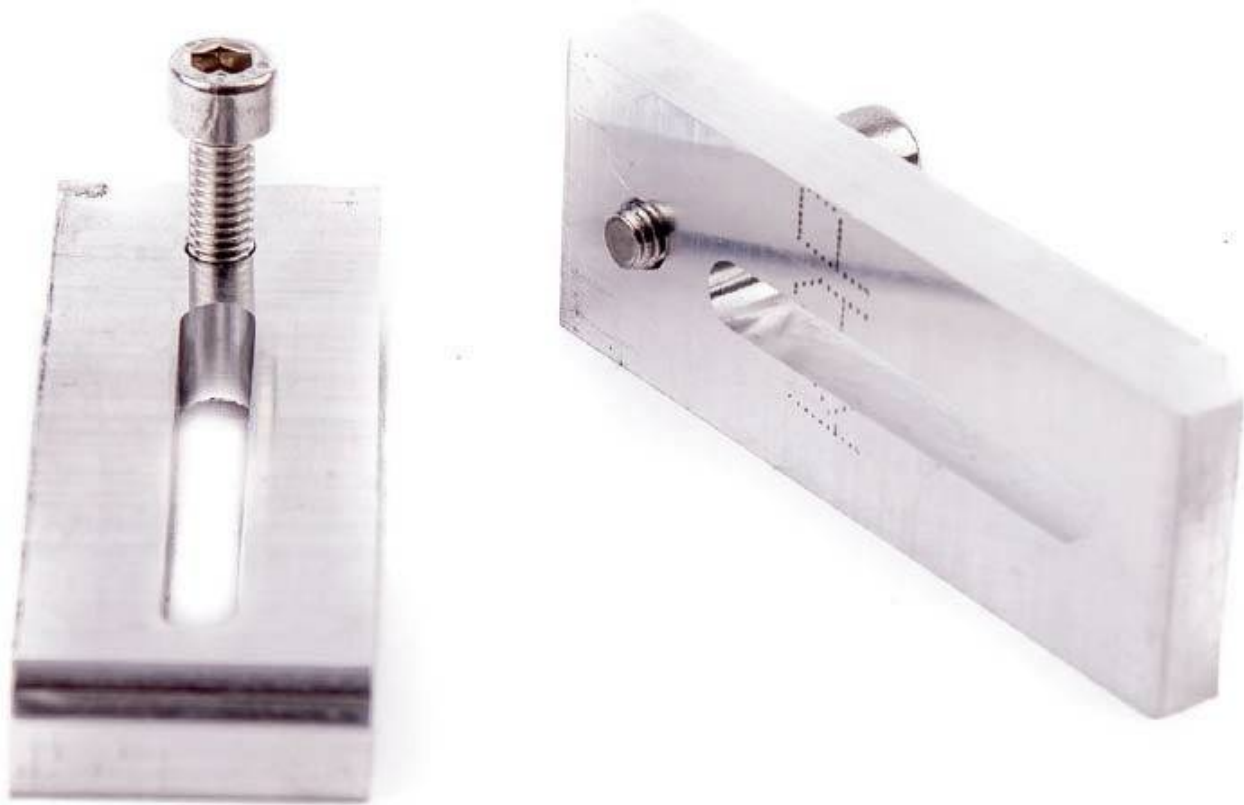


Fig. 5.16 *The finished clamps*

Once you have milled the slots, drill a 5 mm hole for the 6 mm adjusting bolt. To do this, insert a 3 mm drill into the spindle and set the spindle speed to setting #3. Set the center point for the hole the same way as you did for the slot. Once the drill is in position, use the single-arrow button in the *Manual movement* tool to lower the drill until it just touches the

workpiece. Drill the hole by lowering the spindle manually in 0.5–1 mm steps and bore it out with a 5 mm power drill.

Adding a step at one end of the clamp helps to exert downward and lateral pressure on the workpiece. Ours is 3 mm deep and 5 mm wide. You can mill the step using basic face-milling techniques, but you have to take care to set the workpiece zero point correctly. A 6 mm mill can cut a 5 mm step in a series of passes with 0.5 mm depth per pass. The other parameters are the same as those we used for cutting the slot. We made the M6 thread using a 6 mm manual thread cutter and deburred all our cuts to finish off. You can smooth sharp edges using sandpaper or a file, and you can finish the edges of the holes using a countersink.

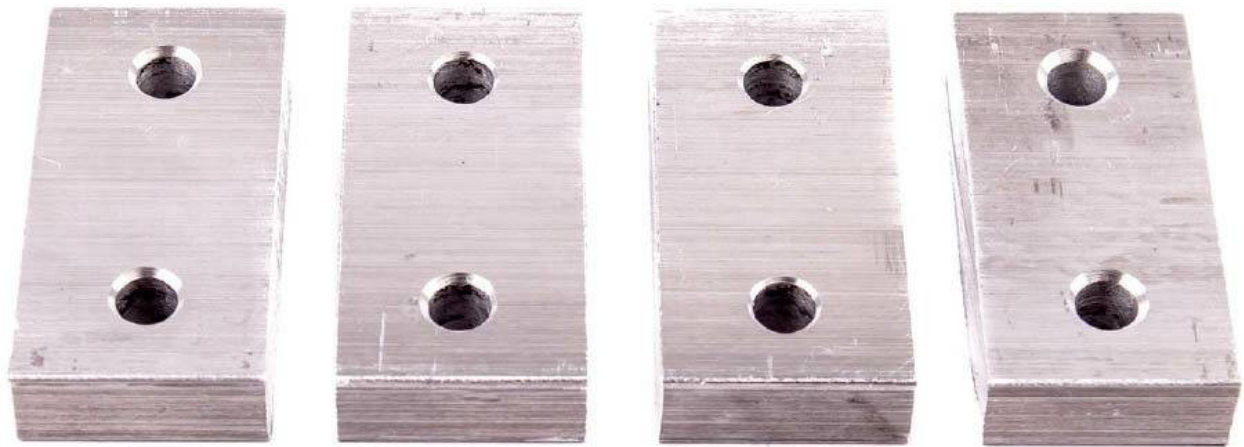


Fig. 5.17 Our homemade clamping blocks

The clamping blocks are simpler to make than the clamps. Once again, we pre-drilled the holes using a 3 mm mill and the *WinPC-NC Manual movement* tool, and finished them off using a power drill and a 6.5 mm bit. Drilling slots instead of holes gives you more positioning flexibility.

You can use different length bolts to clamp your workpieces, but note that the longer the bolt, the less rigid the clamp. For especially thick workpieces, it is better to support the end of the clamp using wood or metal blocks rather than a bolt as shown in [figure 5.13](#).



Fig. 5.18 An alternative clamping method

Clamps and blocks aren't always the best solution. [Figure 5.18](#) shows a workpiece bolted directly to the T-slot table, which is often a more effective approach for large workpieces or complex 2.5D jobs.

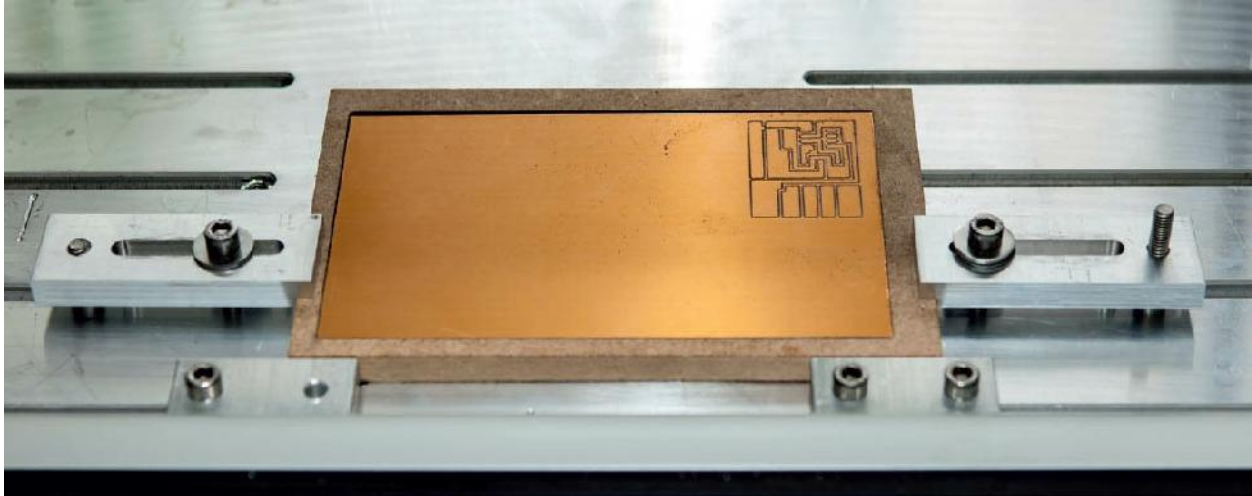


Fig. 5.19 A circuit board holder for isolation milling

The circuit board holder shown in [figure 5.19](#) is made from a piece of MDF with a 2 mm pocket. The circuit board itself is fixed in place in the pocket using double-sided tape. Because there is no direct force exerted on the board, it doesn't bend. The clamps and blocks only have to be screwed down lightly to stop the MDF from sliding around. A basic kit with eight clamps, eight drilled blocks, and a selection of aluminum offcuts will help you to master most situations.

5.5 Machine Vise

The simplest and quickest way to clamp small parts made of metal, brass, or other hard materials is to use a machine vise, which can save a lot of fumbling around with clamps and blocks. A vise is only effective when used with workpieces that have parallel sides, and it has to be firmly fixed in place on the machine table.

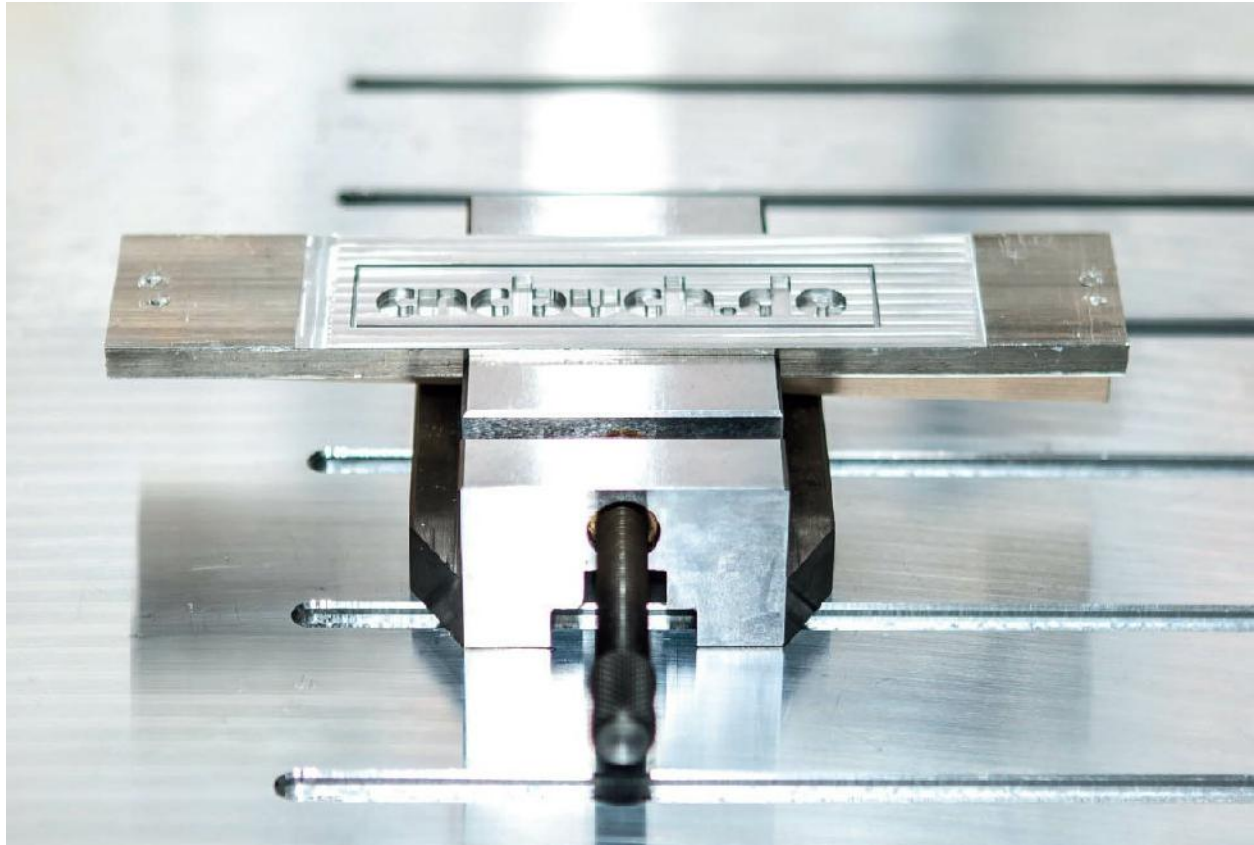


Fig. 5.20 A Proxxon PM 60 machine vise

A small machine vise like the one shown in [figure 5.20](#) costs around US\$80. Vises vary enormously in quality and precision. If the jaws are loose or not perfectly parallel, it can be tricky to clamp the workpiece parallel to the machine table. If the vise doesn't have grooves or built-in holders, you will have to support your workpiece separately.

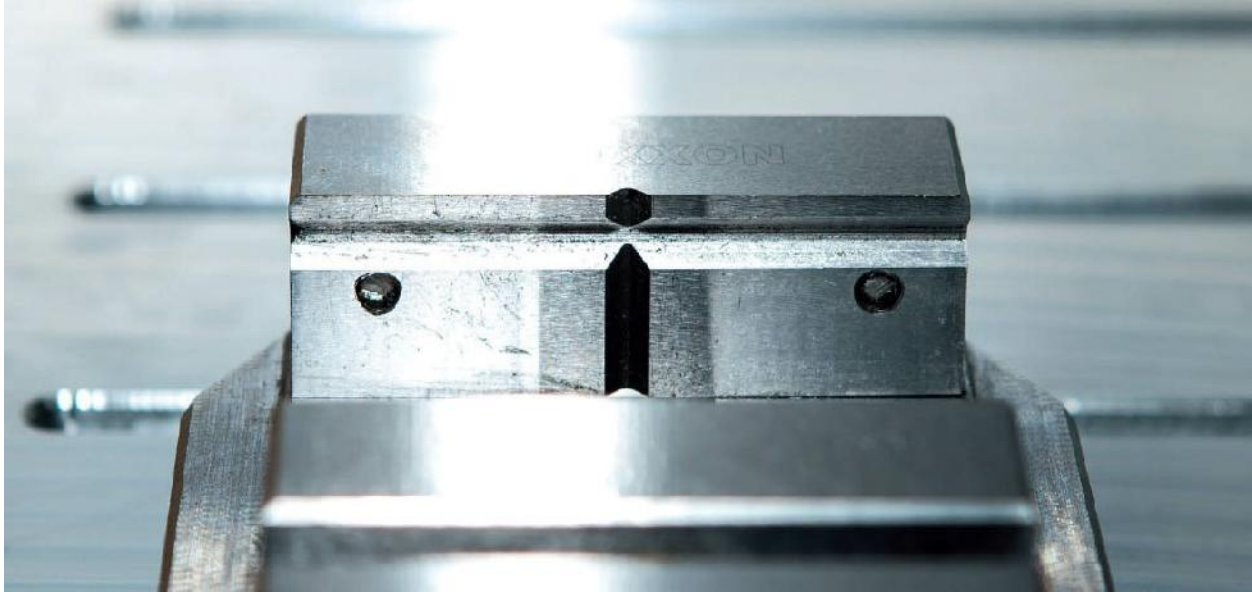


Fig. 5.21 Horizontal and vertical centering grooves for holding rounded workpieces

Round blanks cannot be held firmly using conventional clamps, especially if they need to be clamped vertically. The answer is to use a vise with centering grooves (often called V-grooves), which enable you to firmly fix your workpiece parallel to the direction of feed. An alternative approach for horizontal milling is to cut V-shaped grooves in a spoilboard and fix the workpiece in place using conventional clamps.

5.6 Tool Length Sensor

A tool length sensor consists of an on/off switch whose state can be read by the machine's control software. The software can use the sensor to determine:

- The absolute position of the tool's tip
- The Z position of the upper surface of the workpiece, or of a vise, or other objects on the machine table
- The absolute Z zero point—i.e., the height of the machine table or spoilboard

The distance between the base of the sensor and the switch in its

triggered position is known, so all you have to do to measure the position of an object is lower the tool onto the sensor until the switch triggers, and then read out the Z value.

Without using a tool length sensor, the only way to determine the workpiece zero point is to lower the tool in tiny increments onto the workpiece until it is just touching. This is time-consuming, and the resulting position cannot be accurately reproduced.



Fig. 5.22 A tool length sensor

Using a tool length sensor, all you have to do is move the tool to the XY zero point, place the sensor beneath the tool, and lower the tool until the sensor's switch is triggered. The software then automatically subtracts the height of the sensor and gives you an accurately reproducible value for the height of the workpiece surface. This makes determining the workpiece zero point a quick and reliable process.

A tool length sensor can also be used to determine the clamping

position for individual tools in the machine's collet. Changing the tool in mid-job usually leads to a change in the Z zero point.

To work around this issue, measure how far the current tool protrudes from the collet and save the value. Do the same for all your other tools. The software now “knows” the precise position of the every tool's tip and can automatically correct the zero point for each.

5.6.1 Fitting a Tool Length Sensor

To connect a tool length sensor, first unplug the USB and power cables and lay your machine on its side. Make sure it is level and isn't in danger of toppling. Remove the control unit cover.

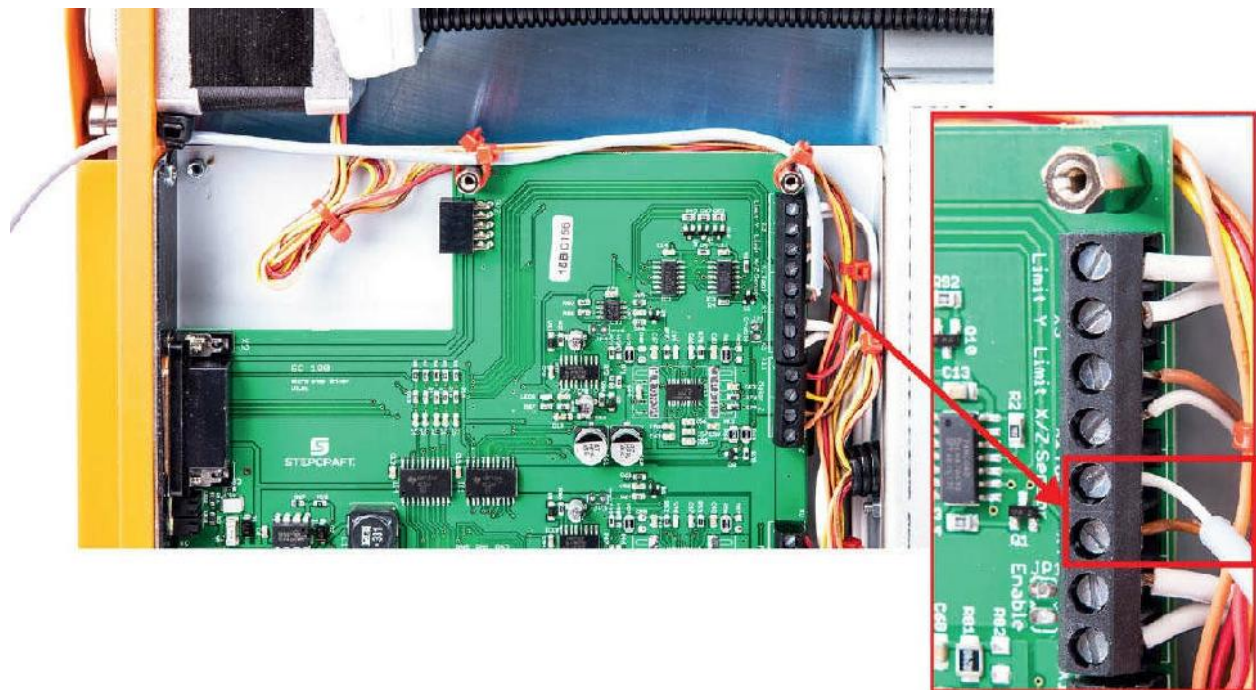


Fig. 5.23 *Connecting a tool length sensor*

1. Strip the ends of the sensor's cables and solder them to ensure a good connection in the screw terminals.
2. Thread the cable through the spare hole on the right of the machine's other connectors (the one currently covered with a rubber grommet). Thread the cable around the control unit as

shown in [figure 5.23](#).

3. Insert the soldered ends into the screw terminals and tighten them. Because we are attaching an on/off switch, it doesn't matter which core is attached to which terminal.
4. The cable is currently connected directly to the control unit so it needs to be secured against accidental pulling. We used two cable ties to fix it to the control unit spacers and added another behind the opening in the rear panel.
5. Refit the control unit cover and reconnect your machine.

5.6.2 Configuring the Tool Length Sensor in WinPC-NC

WinPC-NC has a tool length sensor feature as standard, so configuring it is simple.

1. Connect the machine to your computer and make sure it has power.
2. Open WinPC-NC and navigate to the *Parameters* → *Machine parameters* tab. Click the *Signals* button. Navigate to the *I221 Surface sensor* entry in the *Inputs* list. Make sure that *LPT1 Pin10* is selected in the drop-down list, as shown in [figure 5.24](#).

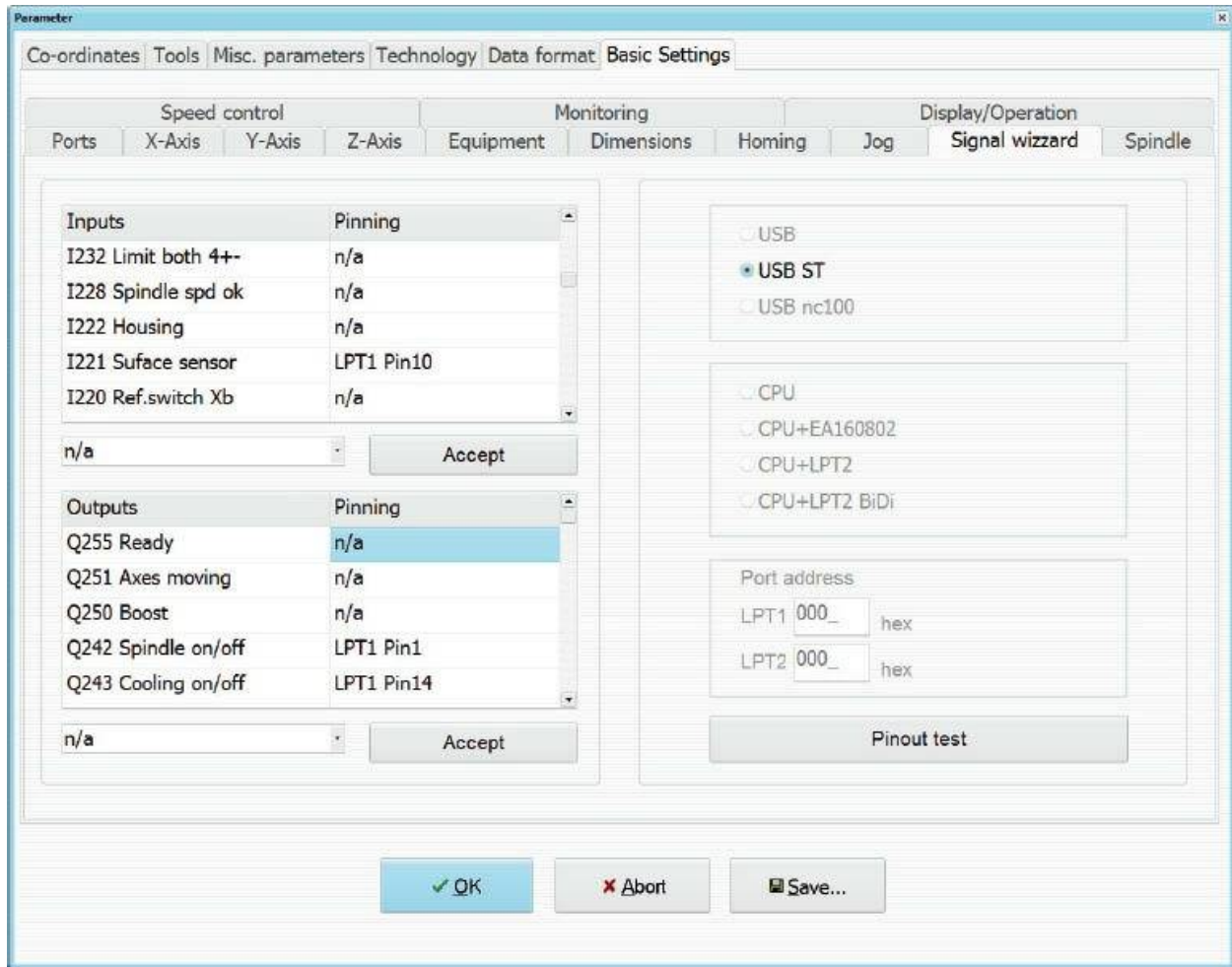


Fig. 5.24 Assigning the sensor signals

3. Click the *Signal test* button and manually trigger the tool length sensor's switch. If the LPT1 pin 10 indicator turns red, the sensor is connected correctly (see [figure 5.25](#)).



Fig. 5.25 The result of triggering the connected tool length sensor

If the Pin 10 indicator shows red and changes to black when you trigger the switch, you need to select *Pin 10 inv* instead of *Pin 10* in the drop-down list. Sometimes, you have to restart WinPC-NC for changes to take effect. If changes you make don't function the way you expect, the first thing to try is restarting the application.

4. Once the sensor is set up correctly, return to the *Parameters* → *Machine parameters* tab and click the *Functions* button. Activate the *Length measurement and compensation* and *Enable surface block* options.

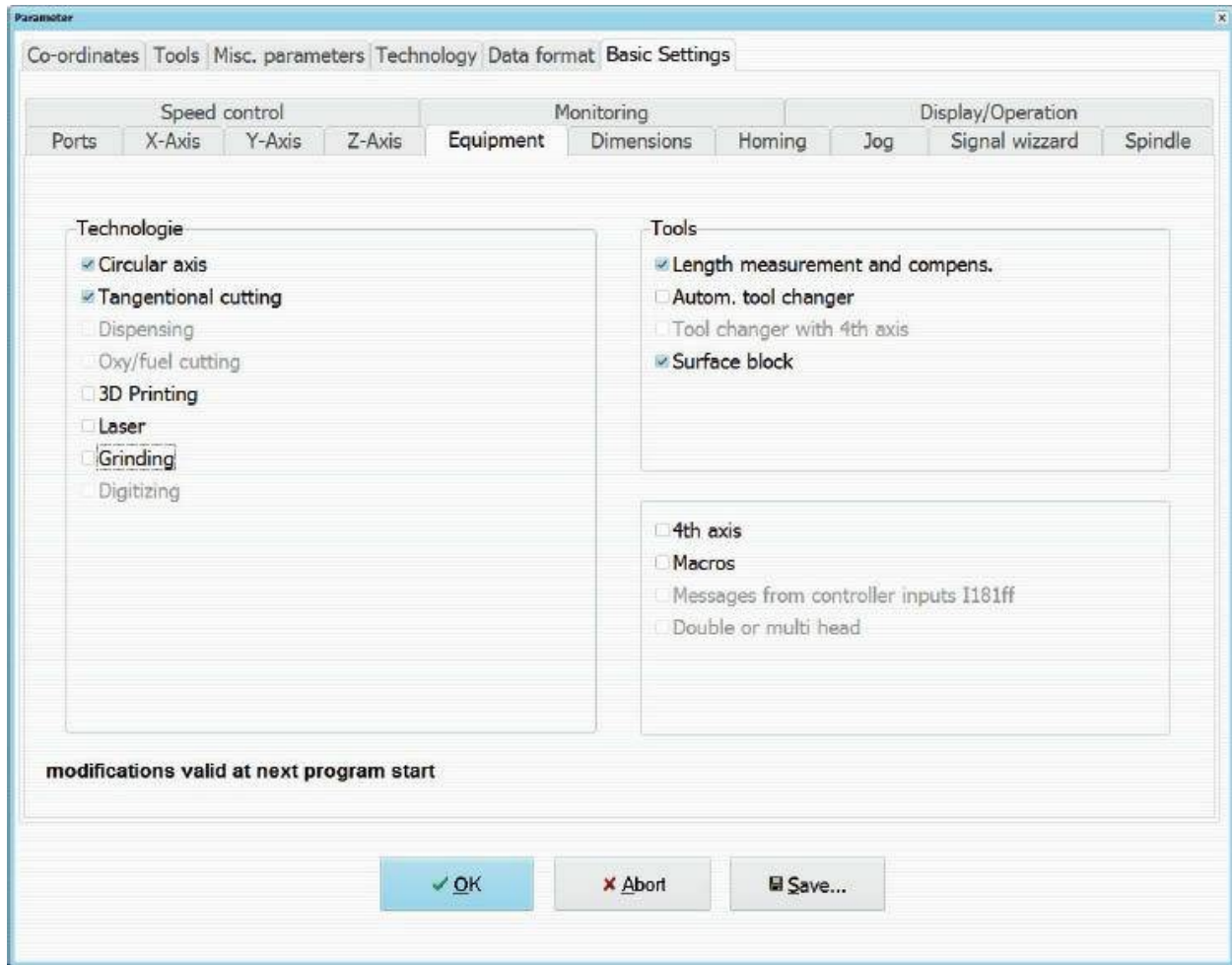


Fig. 5.26 Activating the tool length sensor functionality

To the measure the precise position in which the switch triggers, proceed as follows:

5. Use a caliper capable of measuring to an accuracy of 1/100 mm to measure the distance from the bottom of the sensor to the top of the switch in its triggered position. In our case, this measured 32.13 mm.
6. Navigate to the *Parameters* → *Coordinates* tab and enter the measurement you have just made in the *Surface block* box. WinPC-NC will now use this value to automatically correct any Z values you measure. Because the switch triggers before it actually reaches its final position, the correction will be greater than is actually

necessary. This means the tool will stop a little too high when you determine the Z zero point.

Parameter

Co-ordinates Tools Misc. parameters Technology Data format Basic Settings

	X	Y	Z	
Park position	+ 0.00	+ 598.00	+ 0.00	mm
Scaling factors	1.000	1.000	1.000	
Surface block			+ 32.13	mm
Toollift			+ 10.00	mm
Zero point	+ 126.44	+ 476.15	+ 173.29	mm
Zero point in file	bottom left			

☒ Define size of working piece
☐ Work piece area monitoring

	X	Y	Z	
Working piece from	+ 0.00	+ 0.00	+ 0.00	mm
to	+ 418.00	+ 598.00	+ 138.00	mm

OK Abort Save...

Fig. 5.27 Entering the sensor surface block value

5.6.3 Using and Adjusting the Sensor

You can now test the sensor and determine the correct height value for its switch:

1. Select a workpiece with parallel top and bottom surfaces and place the sensor on it.

Caution!

Use soft foam or Styrofoam blanks when conducting tests or checking altered parameters.

This way, if the tool unintentionally engages with the workpiece, you won't break anything or waste raw material. If in doubt, hit the emergency stop button.

2. Perform a reference movement.
3. Start WinPC-NC, hit the *Manual movement* button, and position the tool at the X-Y center of the tool length sensor and about 10 mm above it.
4. Hit the *Move to* button and select *block - zero point Z* from the list of options that pops up. The tool will descend onto the sensor and trigger the switch before returning to its Safe Z position. The control software records the position at which the switch is triggered.
5. Remove the sensor from the machine area and select the *Move to* → *Zero point Z* command to move the tool to the newly calculated zero point.
6. To determine the precise *surface block* value, you now have to measure the distance between the tool's tip and the workpiece. If you have followed the previous steps correctly, the *Manual movement* dialog will show that the tool is currently located at Z 0. You can now lower the tool manually and use a feeler gauge to ensure that the gap measures exactly 0.1 mm. You can now read off the distance by which you have moved the tool and use it to correct the *surface block* value. In our case, the gap was 0.39 mm, so the final surface block value came out at 32.52 mm.
7. This setting, made using a feeler gauge, ensures that the workpiece Z zero point is always precisely 0.1 mm above the workpiece. To achieve ultimate precision, you have to add this 0.1 mm to the *surface block* value, too. If necessary, you can adjust the zero point to any height you like above the workpiece. The configuration is now complete.



Fig. 5.28 *Using a tool length sensor to determine the Z zero point*

To determine the workpiece zero point, proceed as follows:

- Perform a reference movement
- Place the tool length sensor on the workpiece where you want the XY zero point to be
- Move the tool manually to the sensor position and select the *Move → Block zero point Z option*
- Remove the sensor and select *Move → Zero point Z*
- If desired, save the XY zero point using the *Save to → Zero point XY*. Don't use the *Zero point XYZ* command, as this overwrites the Z zero point you have just saved.

5.6.4 Using the tool length sensor with UCCNC

We have to make some changes to the configuration to get our tool length sensor working with UCCNC. UCCNC has to be told which

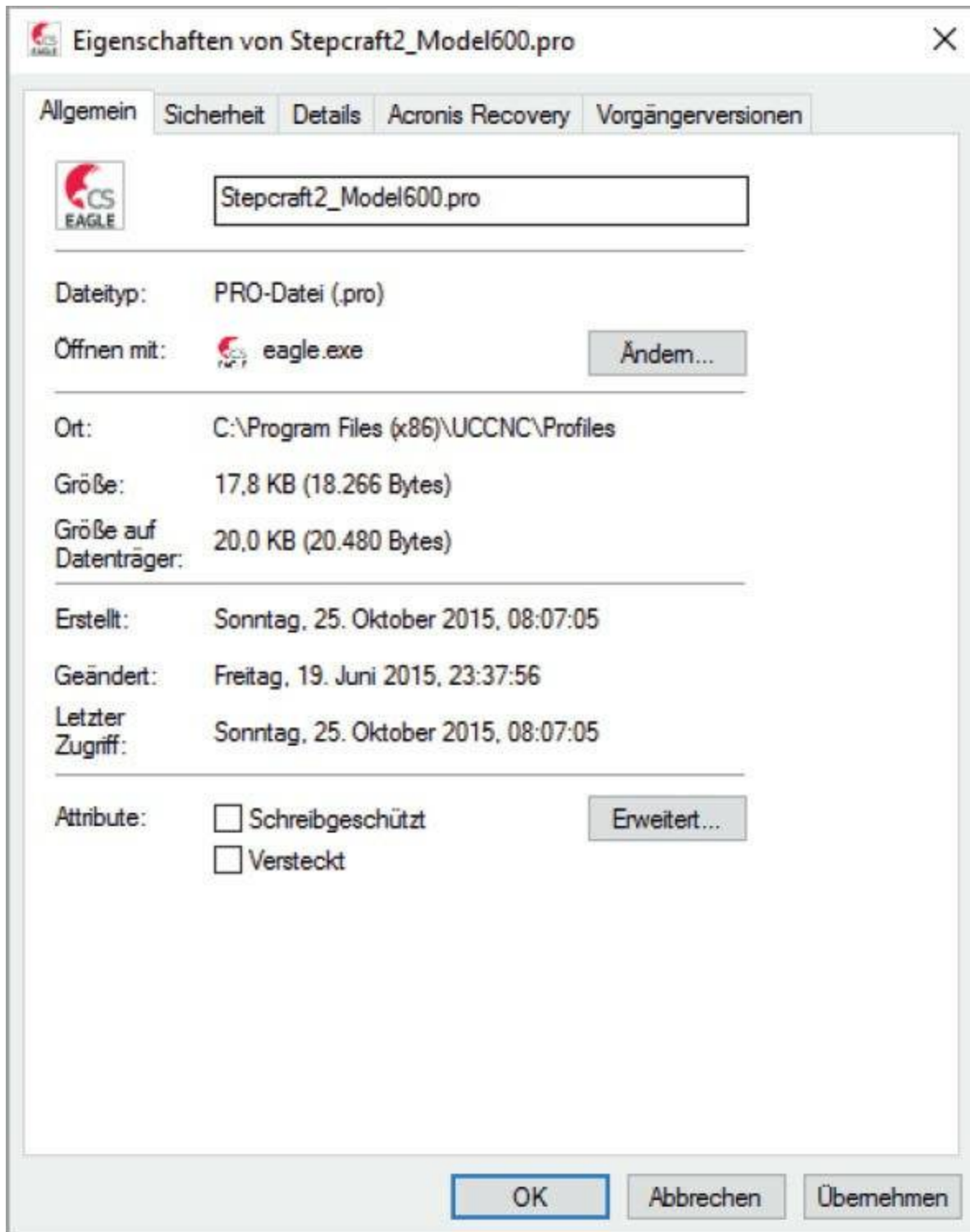


Fig. 5.30 Canceling the read-only setting

Start UCCNC using your machine's profile, go to the *CONFIGURATION* window, and select *I/O SETUP*. The *Probe pin*

setting is at top-left. The tool length sensor is connected by default to pin 10 on the machine, and you need to set *port* to 1. When you are done, hit the *Save settings* button.

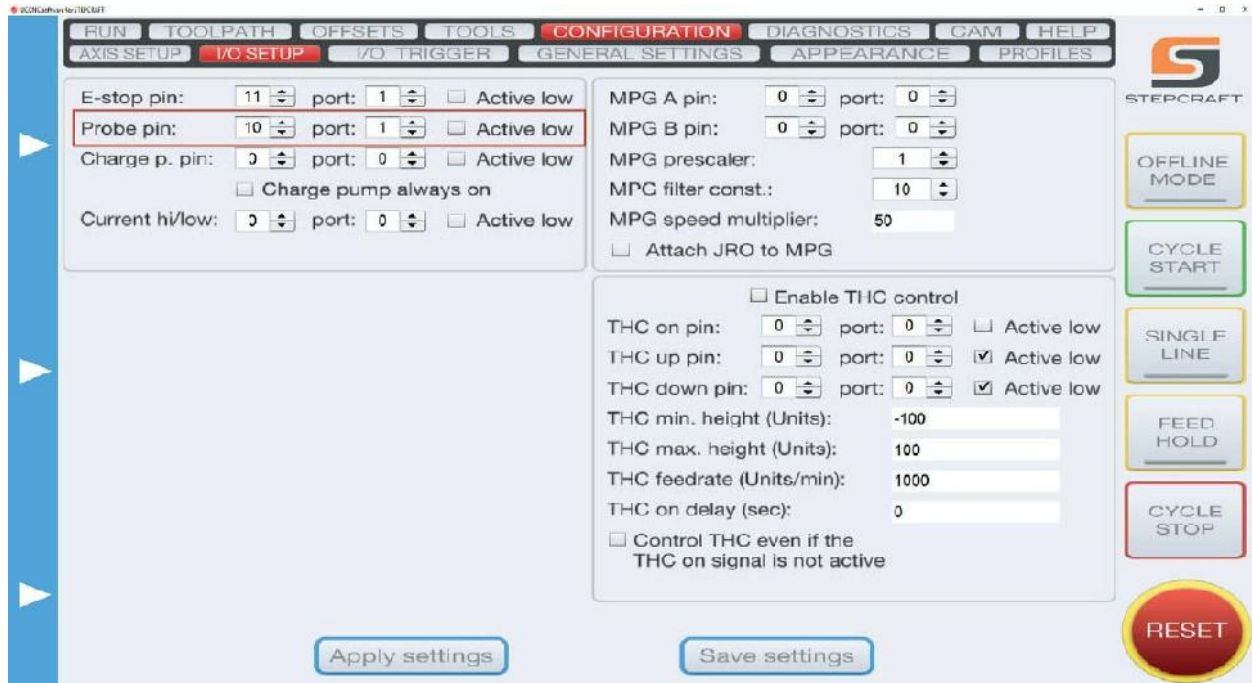


Fig. 5.31 Assigning the tool length sensor to pin 10

To check that the new settings have been saved, restart UCCNC, open the *DIAGNOSTICS* window and check the state of the I/O ports and their associated functions. The tool length sensor is connected to input I10. If you trigger the switch manually, the I10 box should turn green and then back to gray when you release the switch.

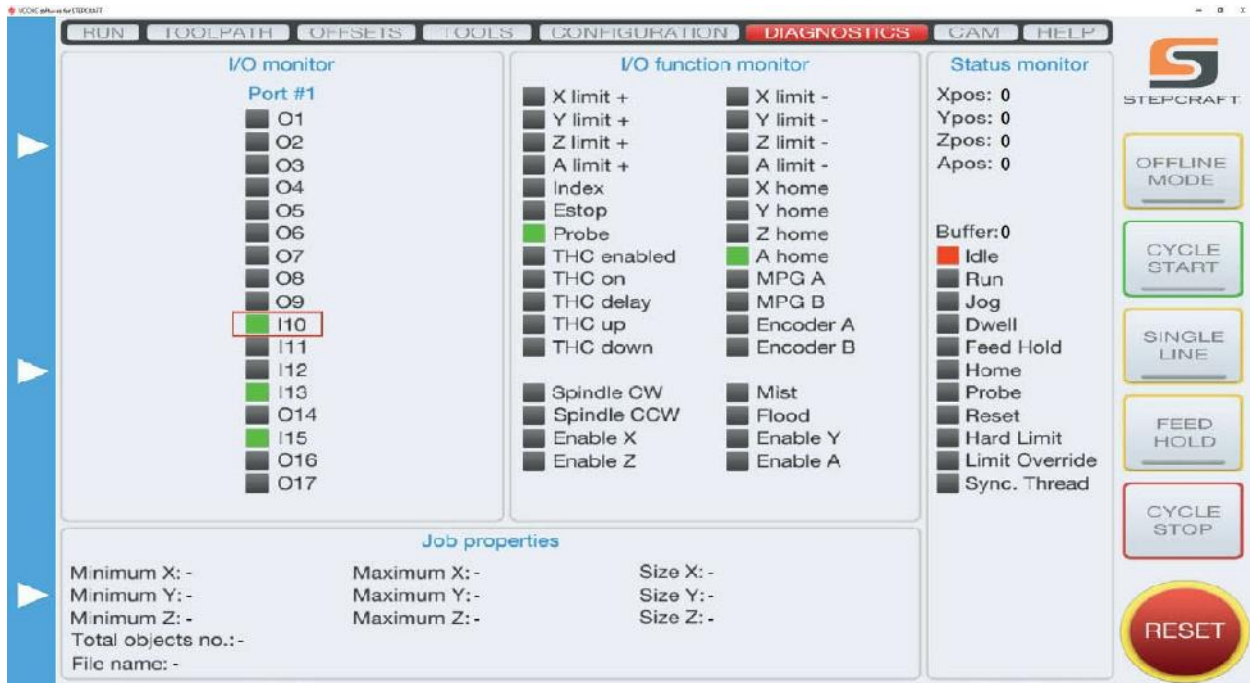


Fig. 5.32 The DIAGNOSTICS screen with the tool length sensor switch triggered

The sensor is now ready for operation. To trigger the sensor, go to the *RUN* screen and hit the probe button (the one in the rectangular red box in figure 5.33).

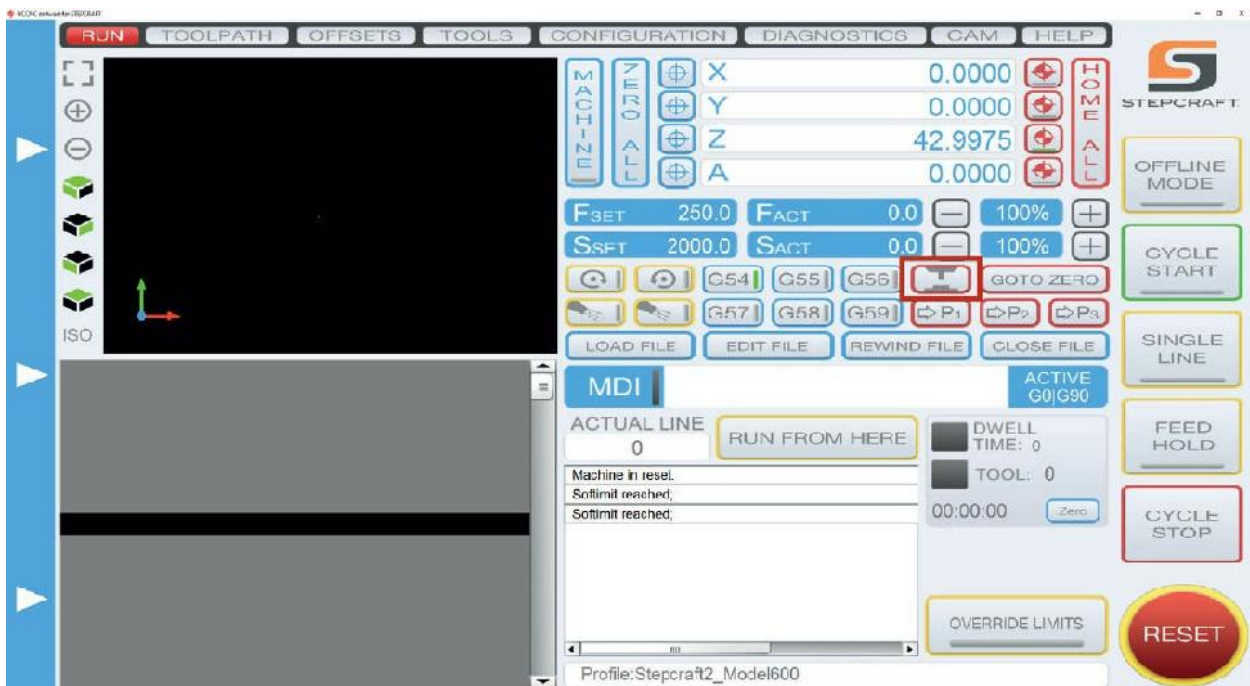


Fig. 5.33 Taking a reading for the tool length sensor

This slowly lowers the spindle until the switch is triggered. Check in advance whether triggering the switch stops the Z movement and, if it doesn't, hit RESET or the machine's emergency stop button. Once the switch has been triggered, the software moves the spindle to its *Safe Z* position.

To use the sensor to determine the Z workpiece zero point, you have to determine and enter the height at which the switch triggers, as described in [section 6.3](#). We used the same 32.52 mm value we measured earlier for WinPC-NC. The value you determine has to be entered in the *M31.txt* macro file.

The default value in the download file is 33 mm. The first four lines of the macro are reproduced below without additional comments. The variable *newZ* is the switch height value. In its default state, this should read 33, rather than the 32.52 shown here. Enter the value you have measured and make sure the line ends with a semicolon.

```
double Zmin = -100;  
double Feedrate = 250;  
double retractheight = 10;  
double newZ = 32.52;
```

The *retractheight* variable represents the 10 mm *Safe Z* setting. After using the tool length sensor to determine the zero point, the Z axis moves upward by this distance (i.e., the tip of the tool is located at the switch height plus 10 mm above the zero point). This value is displayed in the Z box in the *RUN* window. If you then remove the sensor and move the spindle manually to Z0, the tool's tip will be located precisely at the measured zero point.

An Alternative Way to Determine the Sensor Switch Height

Instead of using a caliper, you can use the Z coordinate display in UCCNC to determine the height at which the tool length sensor's switch is triggered. To do this, set an improbably low value (30 mm for example) in the macro and hit the probe button indicated in figure 7.15. At the end of the measuring process, the spindle will move upward by the *Safe Z* value (in this case, 10 mm). The software will assume that the bottom of the sensor is located 10+30 mm below the current position. Remove the sensor and lower the tool manually until it just touches the machine table. Use a feeler gauge to ensure the precision

of this movement. Because the original value was too low, the software will display a negative Z value (in this case, -2.52 mm). This means you have to add +2.52 mm to the 30 mm you entered previously, giving us the same 32.52 mm value as the one we measured earlier.

5.7 Underwater Milling Using a Water Bath

Fine dust and waste particles—especially those produced when cutting carbon fiber, glass fiber, and other fiber-reinforced plastics—present a health hazard and, depending on the size of the fibers, can be carcinogenic. However, these types of materials are all waterproof and can thus be safely milled under water. Using this method, the dust produced remains bound in the water and cannot escape into the surrounding air. A water bath also acts as an active cooling system for the cutting tool and the workpiece. If the material is lighter than water, chips and dust will float away from the cut while you work.

In principle, all you need to do is fix a plastic container full of water to the machine table and clamp the workpiece in place in the water so that it lies about ½" under the surface. The challenge lies in keeping the water bath and the workpiece stable so that the water doesn't slosh about and make contact with the machine's electrics.

It is all too easy to accidentally mill through a plastic water bath and kill your machine in the process. To prevent this kind of mishap, the following sections show you how to make a water bath from solid aluminum. The basic components are a 10 mm smooth-cast aluminum plate, solid 3 mm aluminum brackets, and silicone. We used a thick base plate to prevent unwanted lifting while milling thin (1-3 mm) carbon-fiber workpieces. We selected the largest possible base plate to give ourselves space to mill larger parts.

For use with a *Stepcraft-2/600*, this equates to a base plate that measures 670 × 420 mm (26.4 × 16.5"). We ordered the parts online and paid about US\$150 for the fine-finish base plate and the brackets.

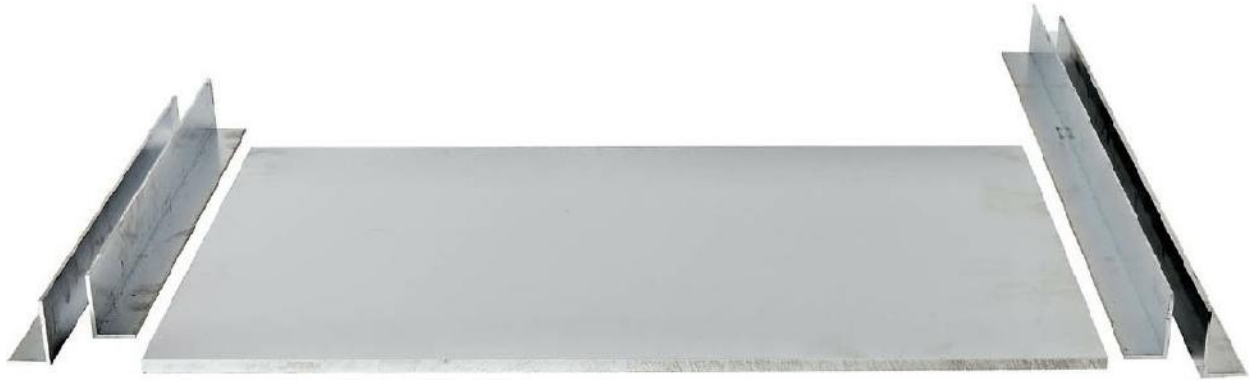


Fig. 5.34 *The raw materials for our water bath*

The holes for fixing the base plate to the T-slot machine table need to be precisely drilled. The distance between the centers of each pair of slots is 50 mm, each slot is 6.5 mm wide, and the gaps between the slots measure 43.5 mm. The distance between the centers of the outermost slots is 400 mm, and these are the ones we used to fix the corners of the water bath in place.

The 6.5 mm slots mean the largest bolts we can use are 6 mm in diameter. The width of the T-slot table is around 434 mm (17.1"), which gives us 6–7 mm of space between the edges of the plate and the sides of the machine table. The centers of the corner holes in the base plate have to be precisely 10 mm from the edges, and we drilled them using a 6.5 mm bit to give us some play.

To fix the brackets (i.e., the sides of the water bath) in place, we drilled 4.5 mm holes every 100 mm (4") starting in the center of each bracket, and countersunk them from below. This gave us three holes per short side and seven per long side. Later on, we will fix the brackets to the base plate using 4 mm countersink bolts inserted from below. We finished our preparations by drilling four 6.5 mm holes in the corners (see [figure 5.36](#)).

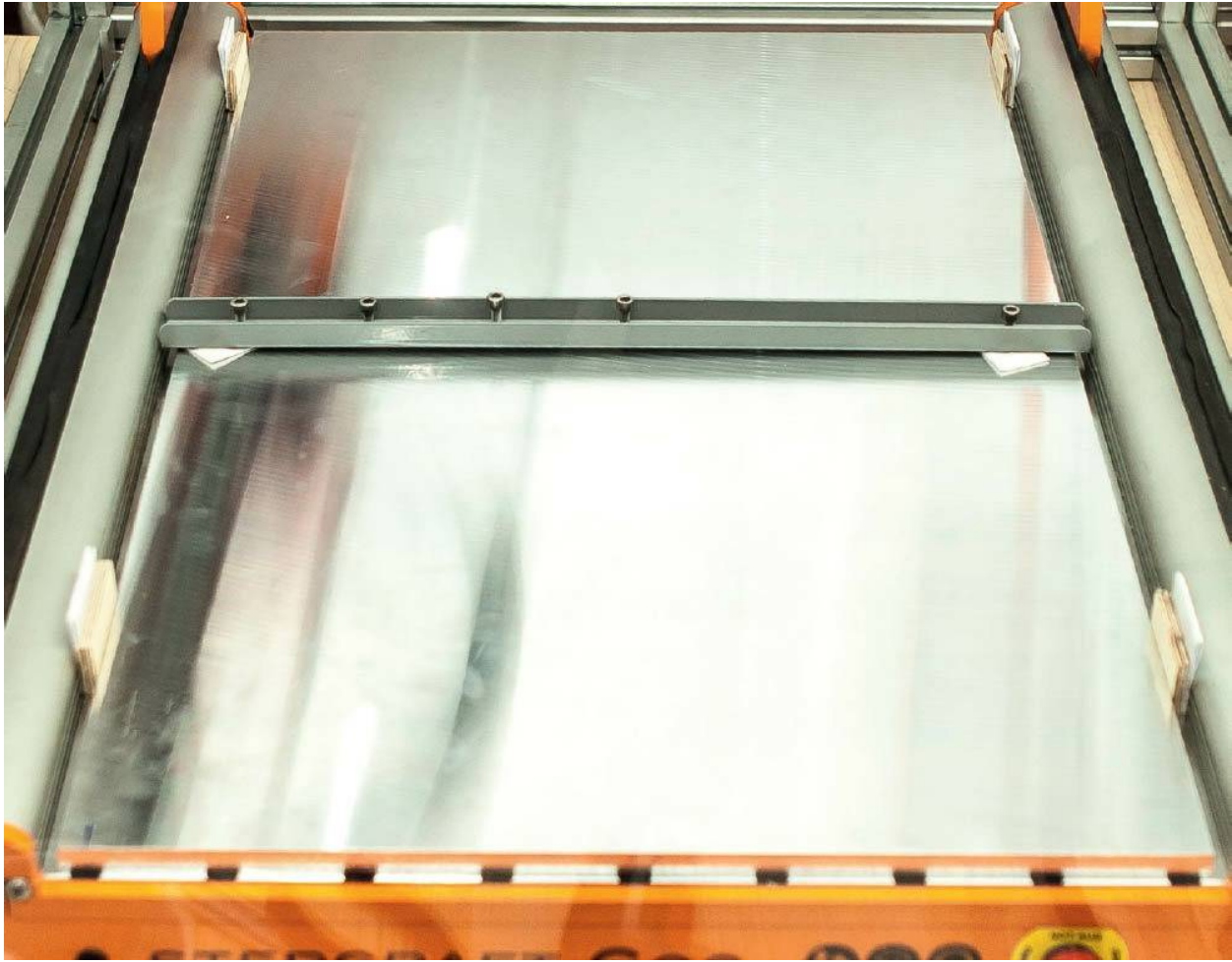


Fig. 5.35 *Aligning and clamping the base plate*

Pre-drilling the holes with a router ensures precise placement, and you can drill them to full bore and countersink them later using a power drill. To pre-drill the base plate, we fixed it in place on the machine table using a clamp, and then used 6 mm wooden wedges and folded paper to prevent it from becoming decentralized.

We used a 2 mm HSS drill bit and, because we were drilling directly above the slots in the table, set a final cut depth of 11–12 mm. However, the machine table wasn't large enough to pre-drill all the holes in one job, and we also had to take care not to drill all the way through the base plate for the two holes in the center of the long sides, as the T-slots aren't continuous over the entire length of the table. To pre-drill these, we first moved the base plate a couple of centimeters so that the holes were aligned with a slot in the machine table.

To drill manually using your router, attach the drill bit to the spindle using an appropriate chuck. Just like when you are milling, you have to stick to appropriate cutting speeds and feed rates when drilling, too. The minimum 10,000 rpm speed of our Kress spindle is just about OK for use with a 2 mm HSS drill. Begin by marking the first hole on the base plate as close as possible to the machine zero point (bottom left) and move the spindle manually to this point before lowering the spindle slowly until it just scratches the surface. Save this position as the Z zero point, raise the spindle, and manually check the position of the hole. If necessary, correct any discrepancies by altering the X or Y positions (the *Manual movement* dialog allows adjustments as fine as a few thousandths of a millimeter).

Once you have completed your settings, save the resulting position as the XYZ zero point. Go to the *Parameters* → *Speeds* tab in WinPC-NC and set the *Manual speed, slow* value to 1 mm/s. You can then use the single-arrow button to lower the tool onto the base plate and begin drilling. Stop the movement every 2 or 3 millimeters to allow the chips to break off. Because we previously set the Z zero point to coincide with the surface of the workpiece, the drill exits the lower surface at a depth of 10 mm. Raise the tool after you have drilled each hole and move it using X or Y movements to the next location.

Once you have used your router to pre-drill the holes, it is a simple matter to bore them out using a conventional power drill. The corner holes for fixing the water bath to the machine table are the only ones that have to be precisely positioned, while the holes for the brackets can deviate by ± 1 mm without causing problems.

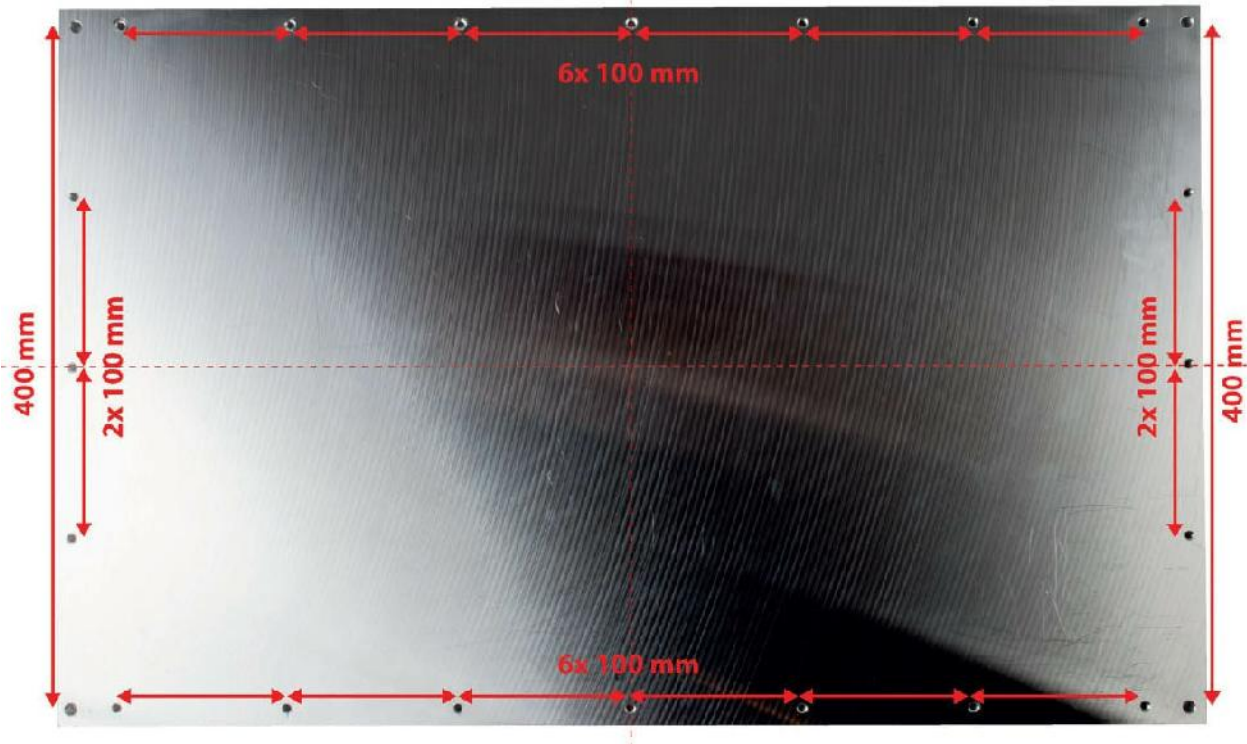


Fig. 5.36 The drilled and countersunk base plate

Once you have successfully drilled the base plate, nothing else can really go wrong. The next step involves clipping the brackets in place and drilling them through to match the holes in the base plate. Make sure you place the brackets with their long side facing upward to form 50 mm (2") walls, as shown in [figure 5.37](#).



Fig. 5.37 The finished water bath

To make sure the water bath remains waterproof, add a thin line of silicone to the bottom of each bracket before you bolt it in place. You also need to seal the inside corner and edge joints of all four brackets. To make sure the silicone adheres cleanly to the aluminum, clean the parts with grease remover and dry them thoroughly before adding sealant. Be sure to wait for the silicone to dry before using your water bath.

Check that the water bath is fully waterproof by filling it half full and placing it on kitchen towels or newspaper. Leave it to stand for a few hours, check for leaks, and add more silicone if necessary.

Caution!

Take great care with your machine when using a full water bath. Only fill your water bath as far as necessary and never fill it to the brim. Depending on the size of your workpiece, the bath described here will hold 3-5 liters (0.8-1.3 gallons) of water. Jogging the water bath can easily slop water over the edges and onto your machine. If water comes into contact with the control unit, it will probably fry. Empty the water bath with a scoop or a pump before you remove it from the machine, or add a drainpipe below the water level in one of the side walls.

5.8 Additional Emergency Stop Button

If you can't reach the emergency stop button fast enough through your machine's enclosure, or if simply pulling the plug or using a switchable power cable isn't safe enough, you can add one or more additional emergency stop buttons wherever you feel they are necessary. For security reasons, the standard emergency stop switch is a circuit breaker that is still active if a cable comes loose. This means you can add multiple additional emergency stop switches wired in series with the standard one.

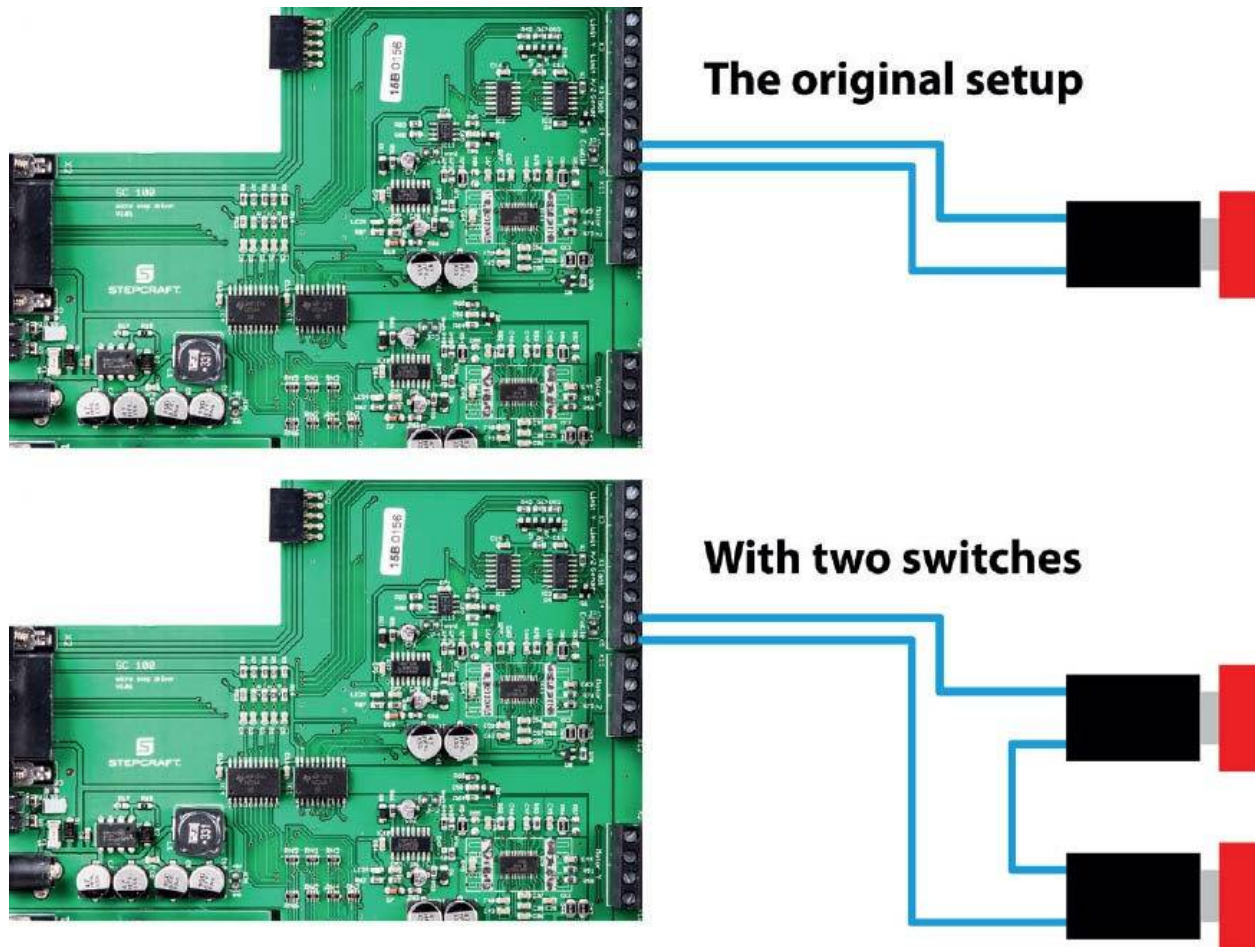


Fig. 5.38 Schematic of the extended emergency stop system

Emergency stop buttons with single or multiple circuit breakers are available in many shapes and sizes and, because they run on a very low voltage, they usually come without a protective enclosure. We built our emergency stop button using separate switching and button modules costing a total of about US\$10. The switch module has two built-in circuit breakers, making it suitable for wiring to other components, too.

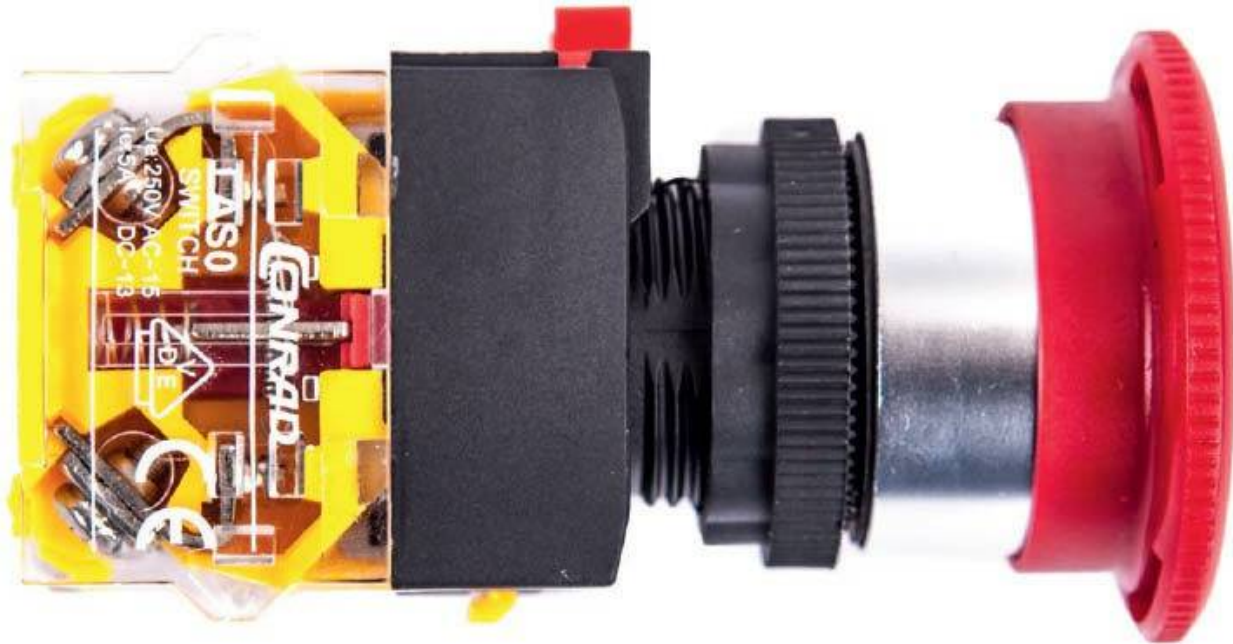


Fig. 5.39 *An accessory emergency stop button*

Figure 5.38 shows how the second switch is wired, requiring only one of the original two wires to be cut.

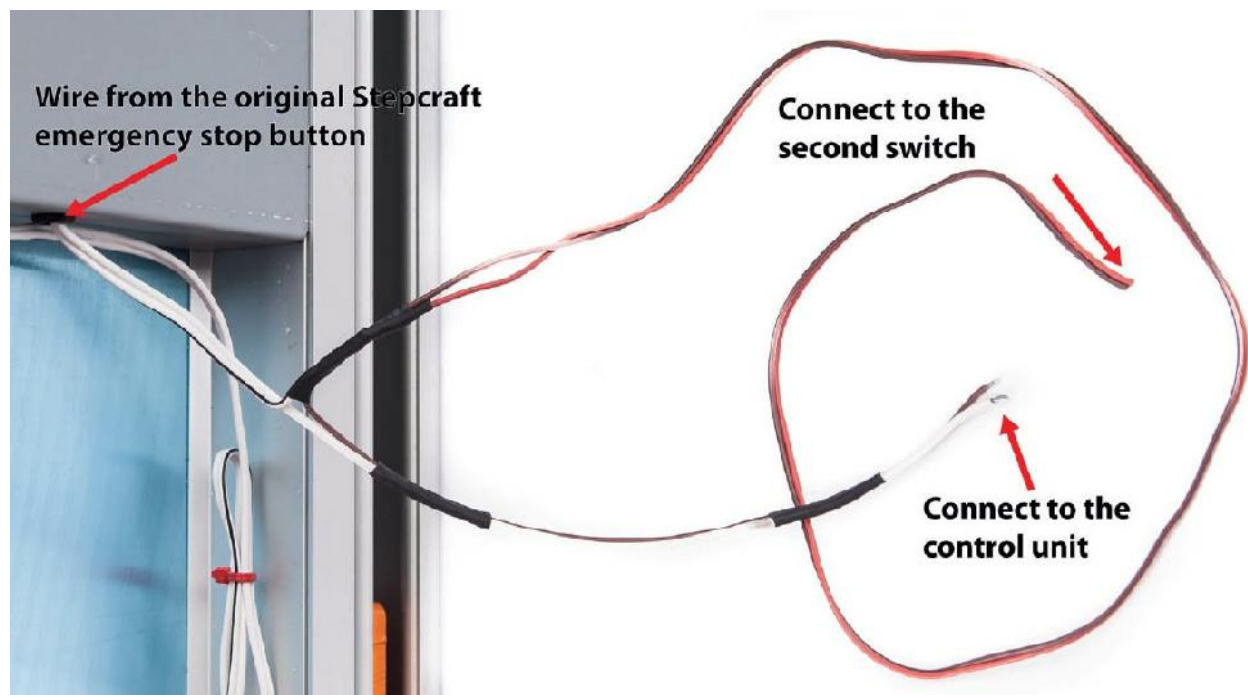


Fig. 5.40 *The wiring junction for our second emergency stop button*

It doesn't matter where you add the extra wire. Figure 5.40 shows how we

released the cable from its clips on the control unit and soldered an additional wire into the loop. We insulated the new wire using heat-shrink tubing. Alternatively, you can remove one of the wires from the original switch and use additional cables with the same type of connector to complete the circuit. [Figure 5.36](#) shows how we reconnected the white/brown cable to the control unit and threaded the red/brown cable through the hole in the front panel (the one the tool length sensor cable is already threaded through). As with all the other wires, use a cable tie to fix it securely in place.

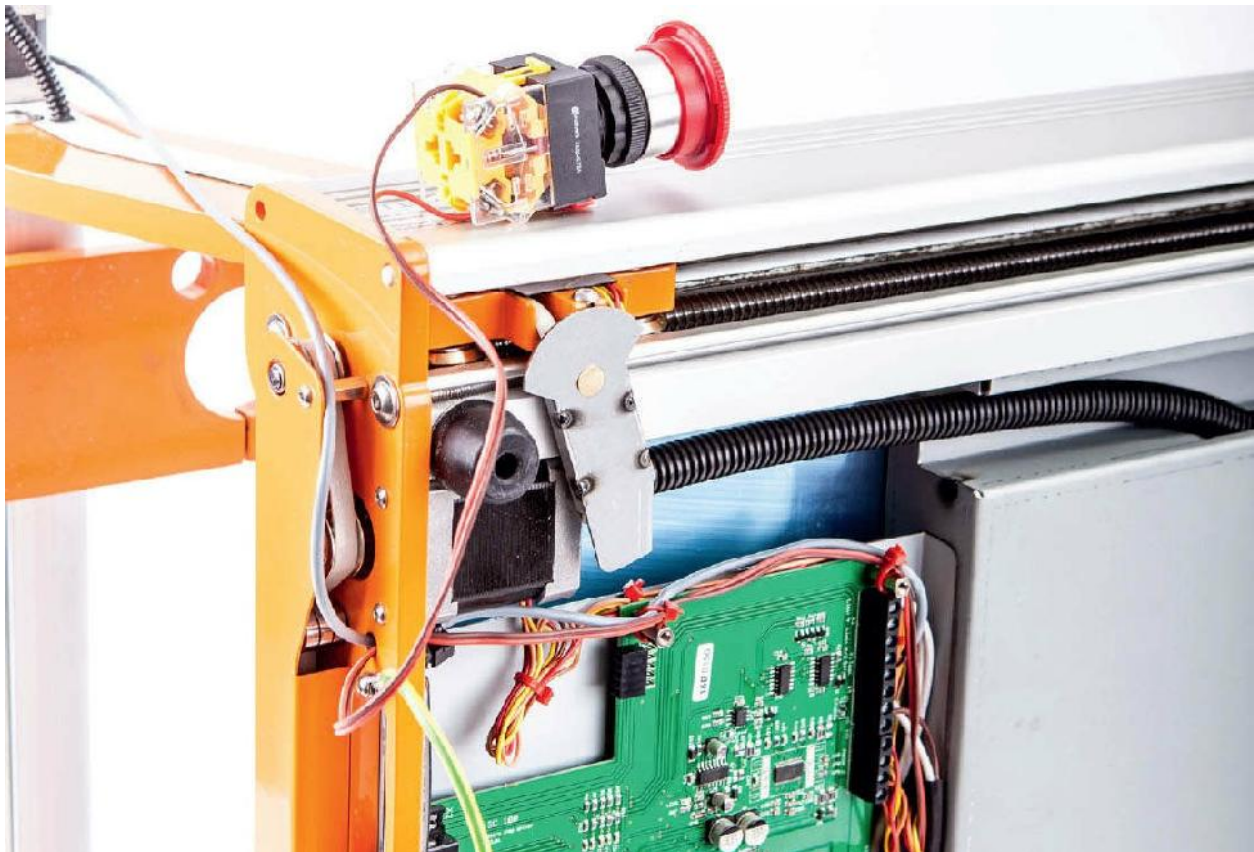


Fig. 5.41 *Our test setup*

Before you reassemble your machine, connect the new switch, connect the machine to your computer, and check that both switches are correctly identified by the software. Pressing one of the buttons (or both) should trigger the emergency stop indicator, and the application should only restart once both switches have been reset.

Mount the emergency stop button where you have easy access to it but not where you can trigger it unintentionally. Once you have set up an external emergency stop wire, it is simple to add further switches wired in series.

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.

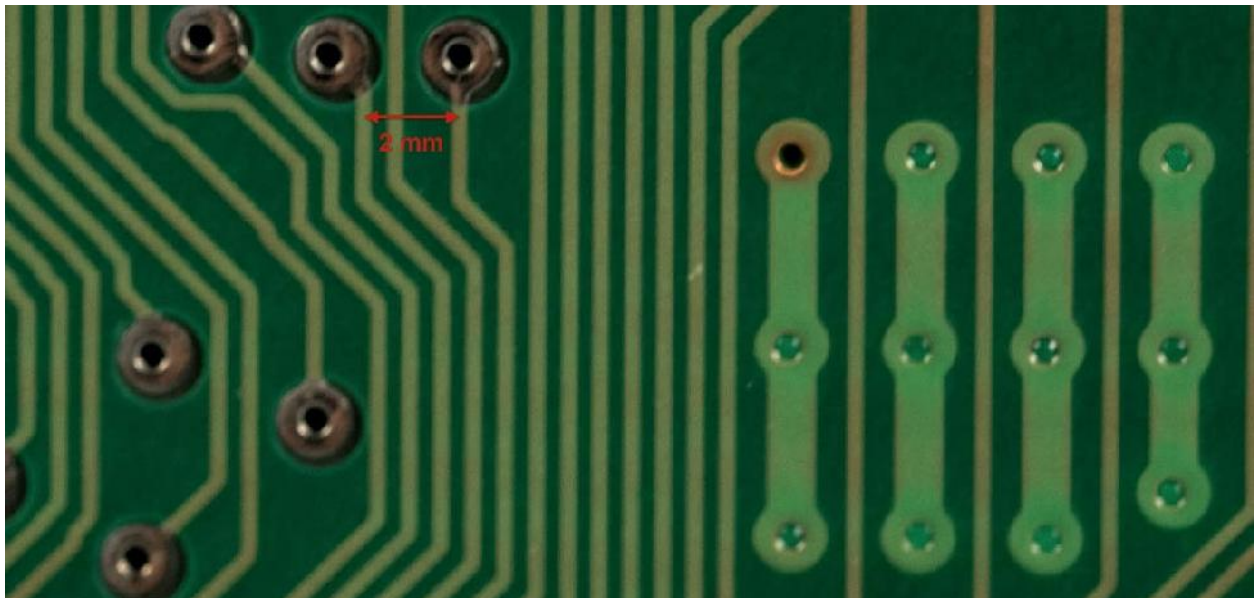


Fig. 6.1 *One of the challenges involved in milling a circuit board*

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.



Fig. 6.2 A 60° 0.2 mm graver

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.

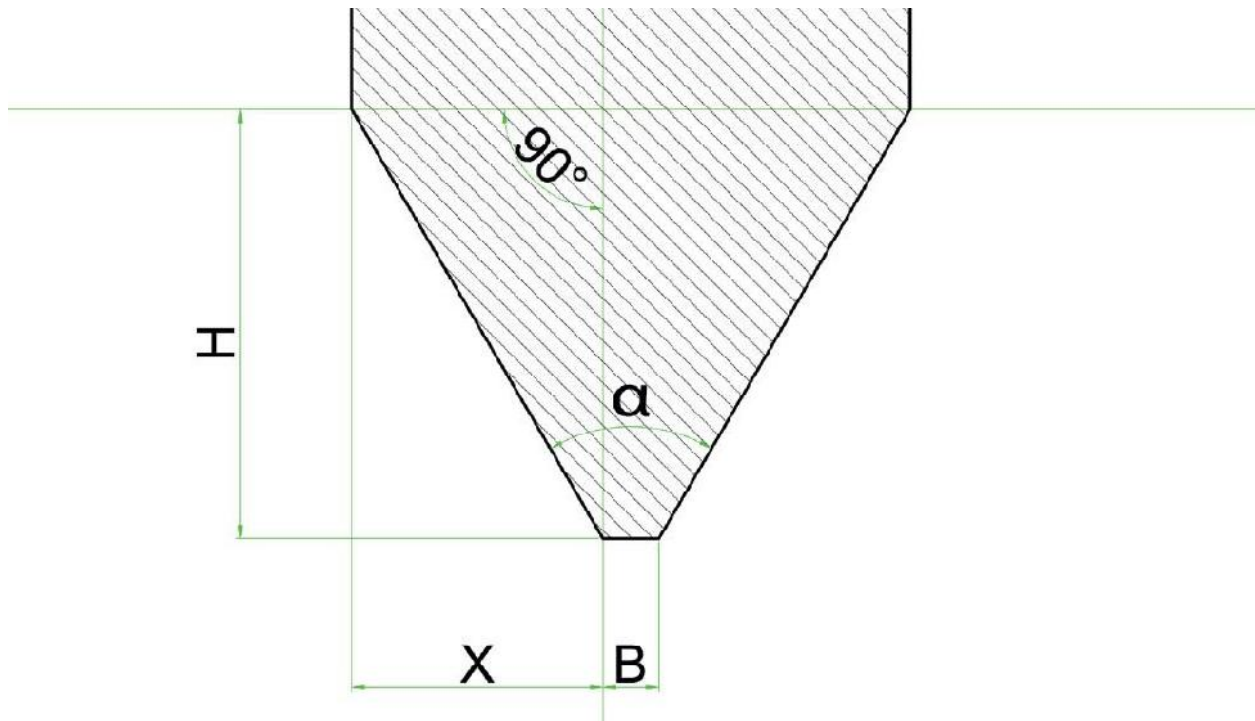


Fig. 6.3 The geometry of the tip of an engraving bit

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:

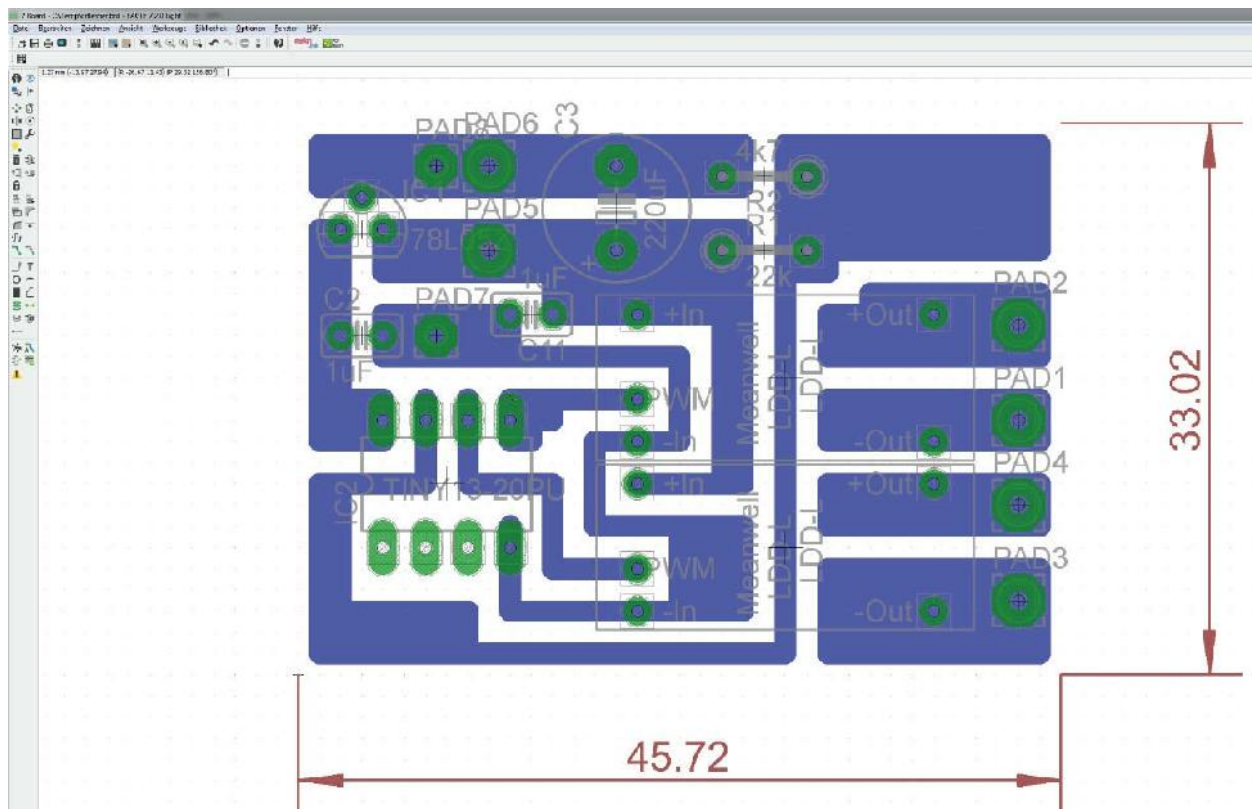
$$B_{\text{Fräsbahn}} = 0,1 \text{ mm} + 2 \cdot 0,15 \text{ mm} \cdot \tan \frac{\frac{60^\circ}{2} \cdot \pi}{180} = 0,27 \text{ mm}$$

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.

6.1.1 How to Configure Eagle

The follow sections assume that you have Eagle installed and that you know how to use it to create a circuit board. The screenshots show version 7.2. The sample project is based on a circuit that makes high-power LEDs flash. The project files are available for download from the book's website at <https://cncbuch.de/en>.



sub-directory in your Eagle installation.



Fig. 6.5 Starting the PCB-GCode ULP module setup process

Once you have copied the files switch back to Eagle, type `run pcb-gcode --setup` into the command line (see figure 6.5), and press return. This opens a setup dialog where you need to set the *Generation Options* shown in figure 6.6 before proceeding.

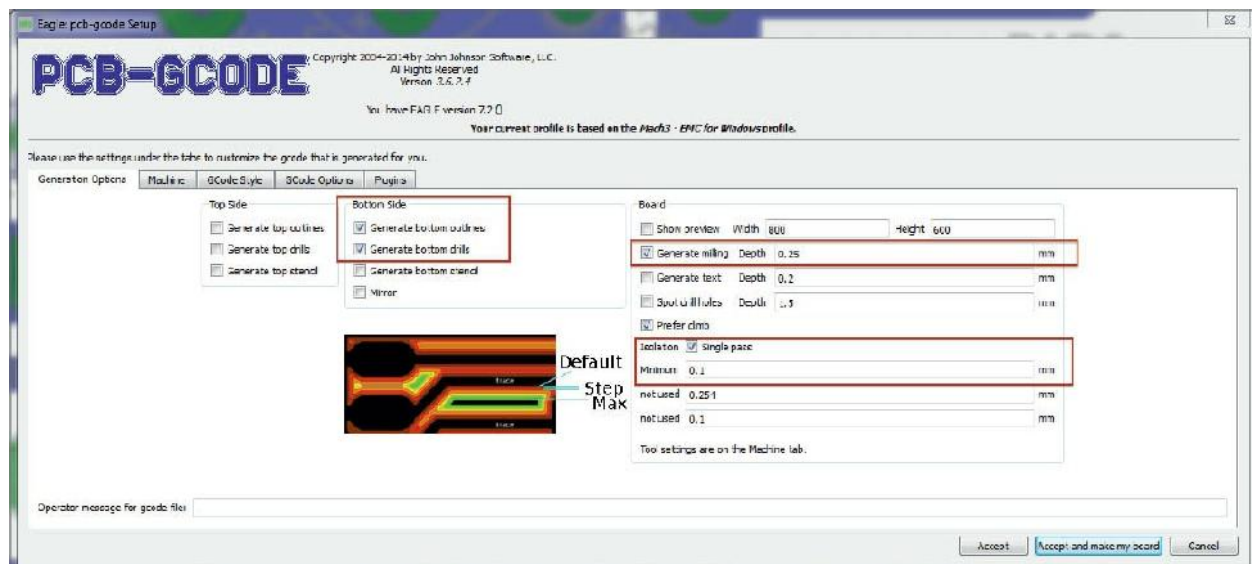


Fig. 6.6 The G-code generation options

The next step involves entering the *Machine* parameters. Make sure you set the units to millimeters to match those used by WinPC-NC. To begin, set low X,Y, and Z feed rates so you can react quickly if one of your settings doesn't work (more on this later). As long as you haven't

inverted the Z axis in WinPC-NC, negative values are above the X axis and positive values are below it. A *Z High* value of -30 (as shown in [figure 6.7](#)) means the tool's park position is 30 mm above the board.

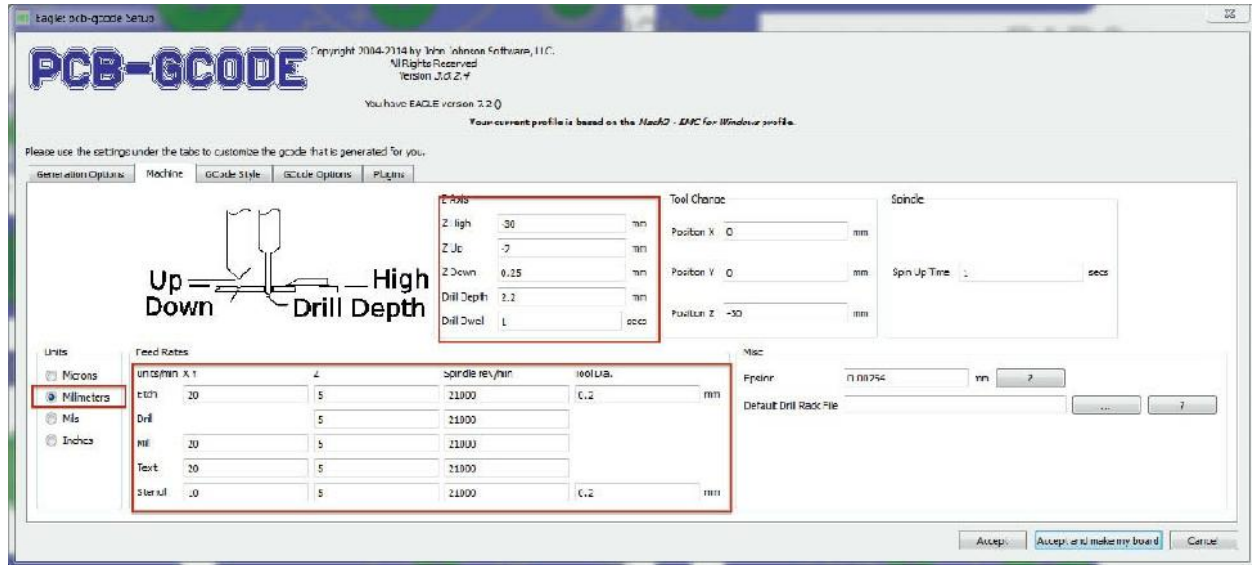


Fig. 6.7 The Units, Withdrawal, Feed Rates, and Drill Depth settings

In the *GCode Style* tab, select the appropriate post-processor profile (see [figure 6.8](#)). The *Mach 3* profile is basically the one you need, but it includes the wrong G-codes (G20 and G21) for switching between metric and imperial units, which have to be altered to G70 and G71. To do this, download the *winpcnc.pp* file from the book's website and copy it to the *ULP\profiles* sub-directory in your Eagle installation. You can then select the new profile in the PCB-GCode setup dialog. Hit the *Accept* button to apply the new profile, which will then be displayed as active at the top of the dialog window.

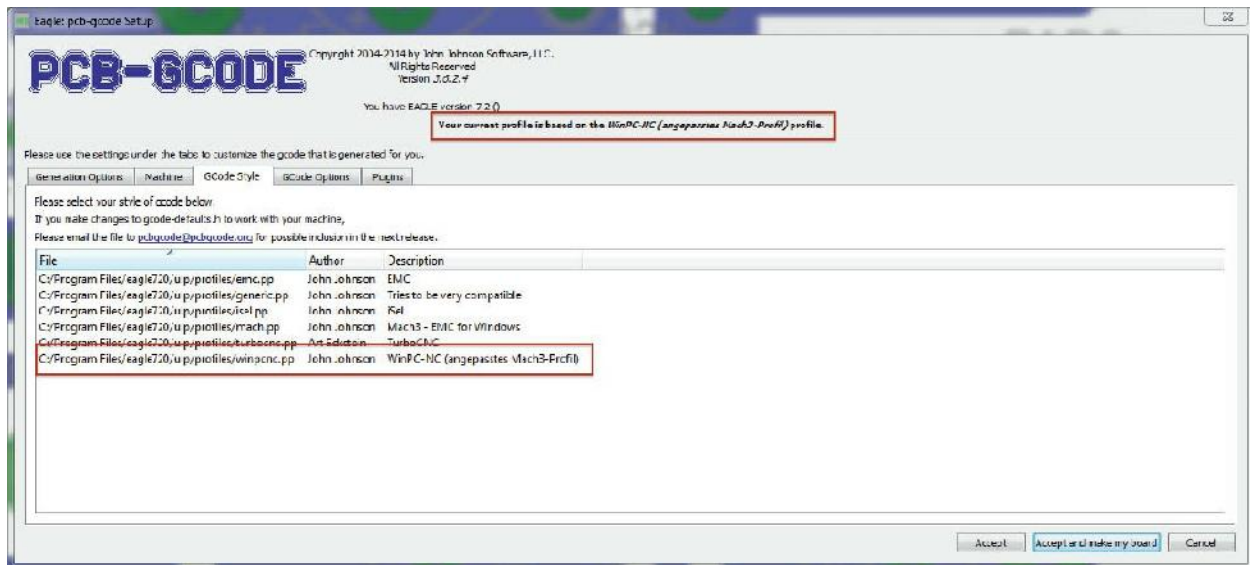


Fig. 6.8 Selecting an appropriate G-code profile

The only setting you need to make in the *GCode Options* tab is switching the filename extension to *.nc*.

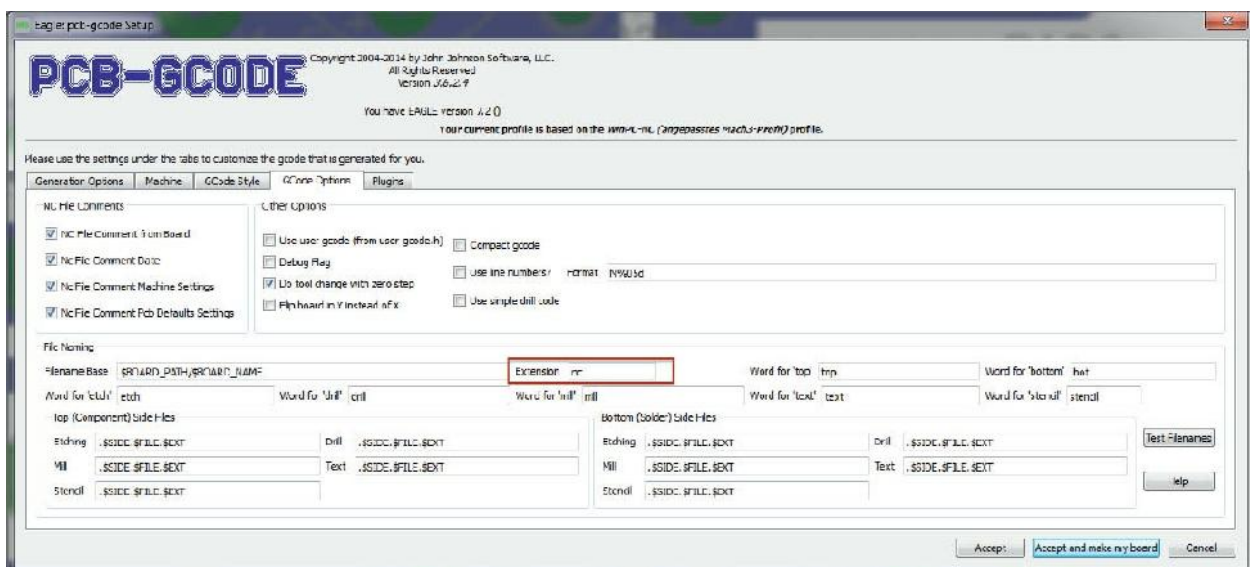


Fig. 6.9 Changing the filename extension

Click the *Accept* button to close the dialog and open the *DRC* dialog in the *Tools* menu. Now make the settings shown in figures, 6.10, 6.11, and 6.12.

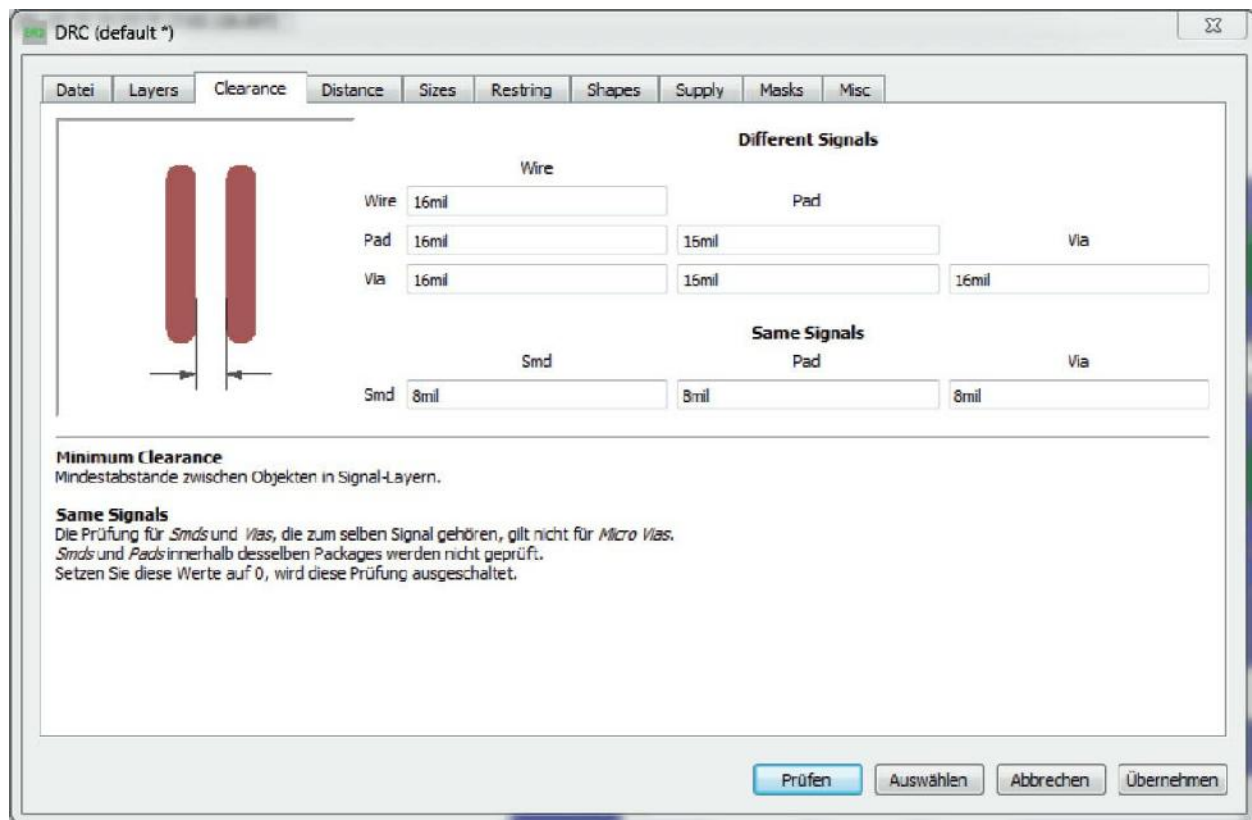


Fig. 6.10 The Clearance settings

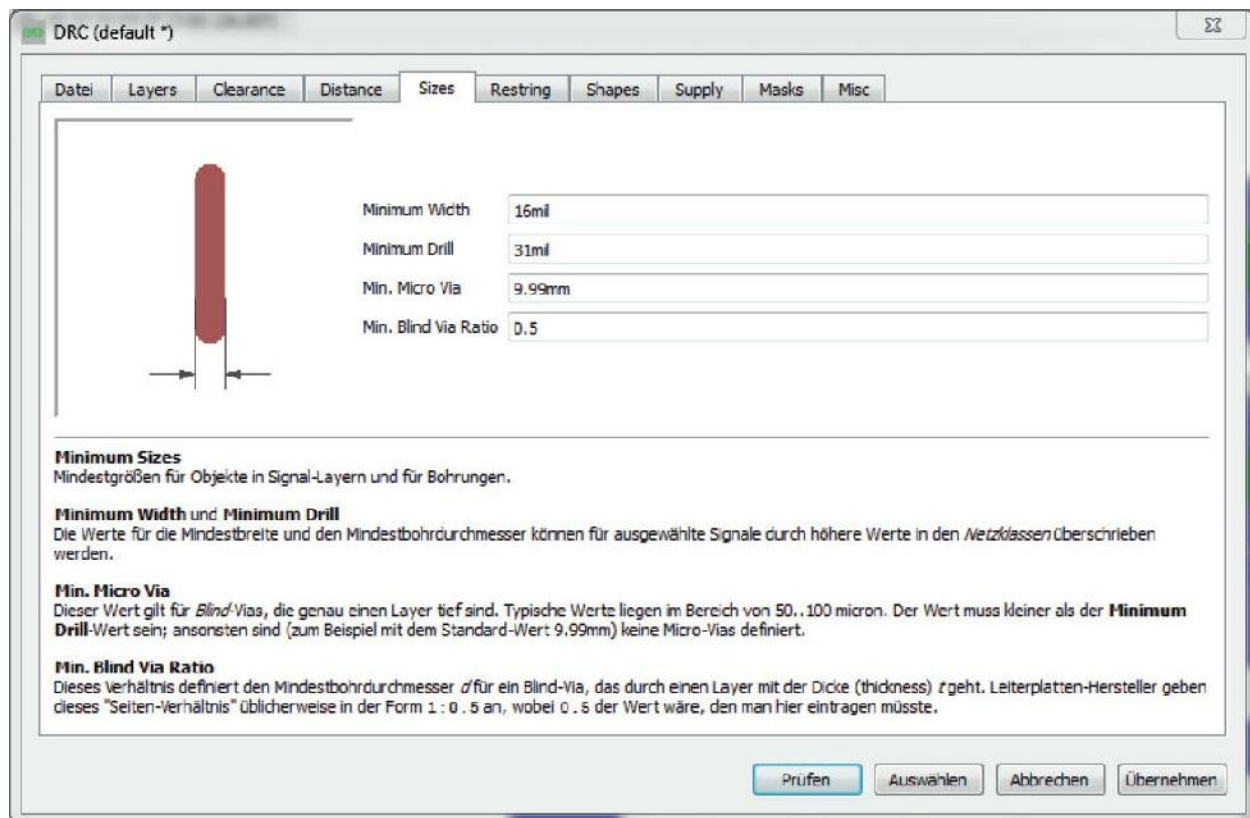


Fig. 6.11 The Sizes settings

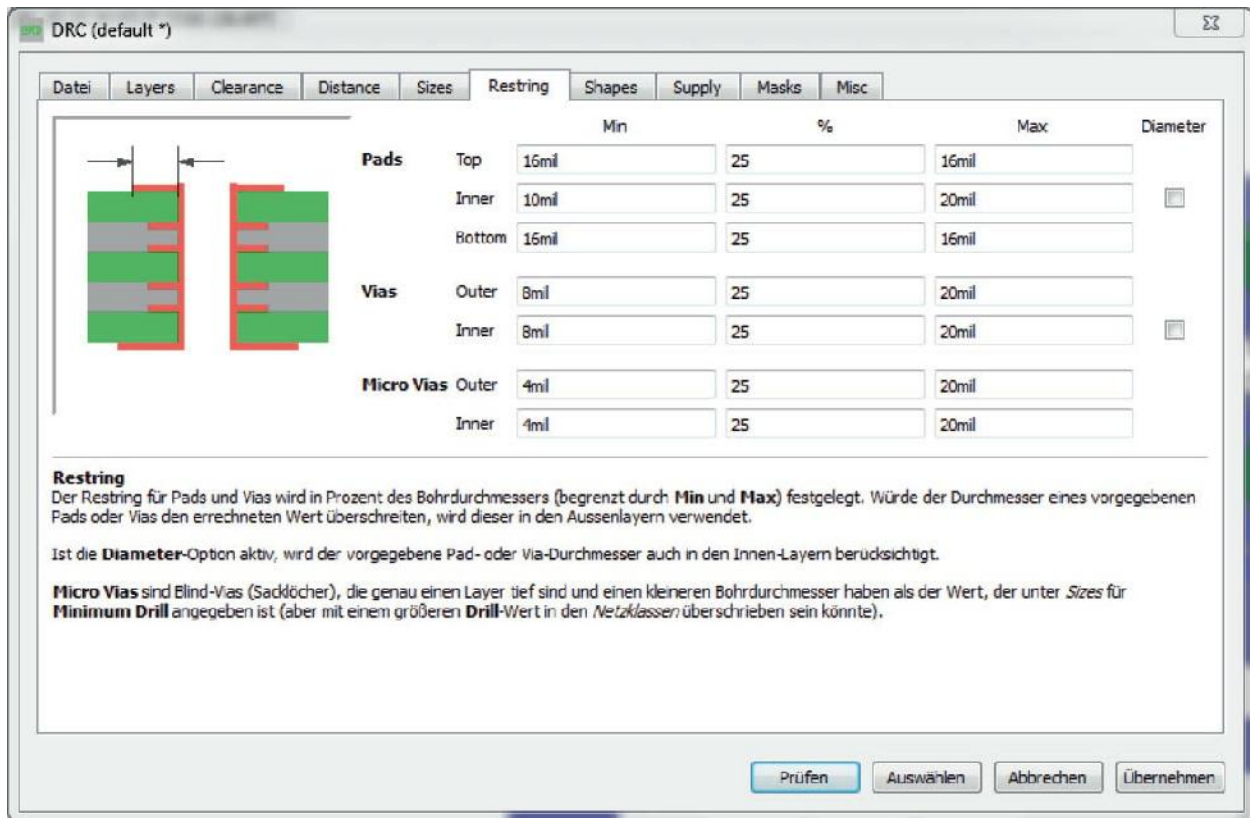


Fig. 6.12 The Restring settings

Once you have made your settings, click the *Check* button. The software will now check whether the loaded board matches the new settings. Any discrepancies are displayed in a separate *DRC Errors* window, and clicking an error takes you directly to the affected location on the board. Correct your layout as necessary.

Once your layout is error-free, you can go ahead and generate your G-code files. To do this, type *run pcb-gcode* into the Eagle command-line and press return. This produces a number of new files in the same directory as the loaded Eagle file, the names of which begin with the same Eagle filename but with *.bot.etch.nc* and *.bot.drill.nc* extensions appended. The program may generate other files, too, depending on the options you selected during setup. The *bot* extension stands for the bottom of the board, while *etch* contains the G-code for the traces and *drill* the G-code for the holes.

6.1.2 Milling a PCB Using WinPC-NC USB

Start WinPC-NC and perform a reference movement, then open the *Parameter* dialog and make sure the settings match the ones shown in figure 6.13.

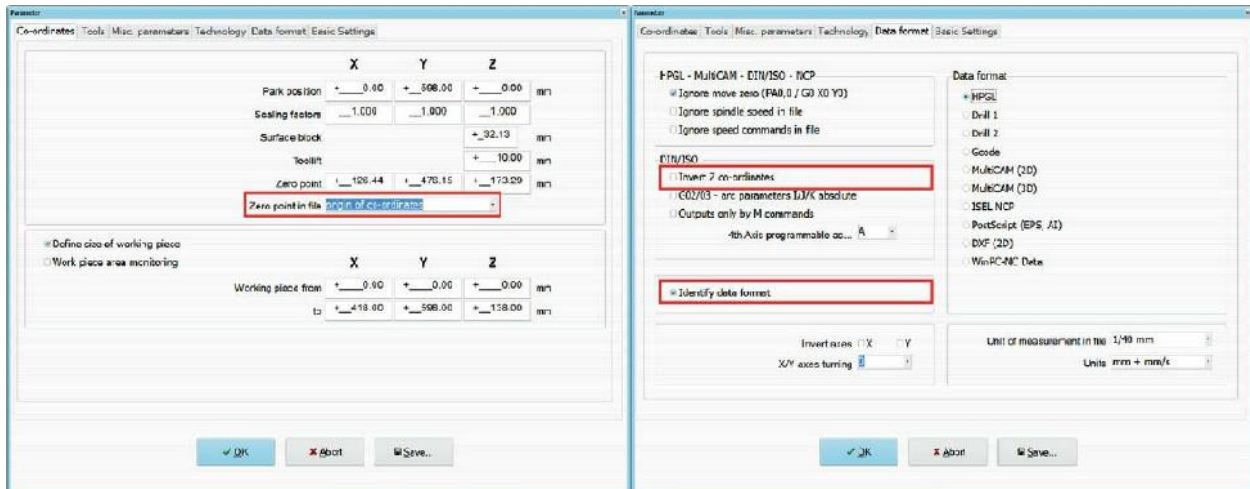


Fig. 6.13 The required settings in WinPC-NC USB

To make sure the *drill* file is congruent with the *etch* file, select the *origin of co-ordinates* option in the *Zero point in file* drop-down list. Make sure the *Invert Z co-ordinates* option is unchecked. If you invert the Z axis, you have to alter the appropriate PCB-GCode setting shown in figure 6.7 to switch the +/- sign for all Z values.

Caution!

If the workpiece Z coordinates don't match the machine coordinates, the software will attempt to move the tool downward by the saved amount—i.e., it will ram the tool into the machine table. To prevent this from happening, enter a Z zero point that is 2 or 3 inches above the table, switch off the spindle, and start a test run. You can then check whether the tool movements are correct before starting the real job.

Normal hold-down clamps are too bulky for clamping a small circuit board. A better solution is to clamp a large spoilboard to the machine table and stick the circuit board to it using double-sided tape. Take care not to use too much tape, as you might not be able to unstick the board when you are done. Use small (1") strips and check with the addition of each strip whether the board is securely fixed.

Now load the *.bot.etch.nc* file and move the tool to the zero point. As you have already selected the *origin of co-ordinates* option for the file zero point, you have to make sure that you have enough space surrounding the zero point (in our case, bottom-right—see [figure 6.14](#)). Use the *Move machine to destination point* feature in WinPC-NC to check that you can reach all points on the workpiece. Especially when you are starting out, make sure you leave plenty of room for maneuver in all directions.

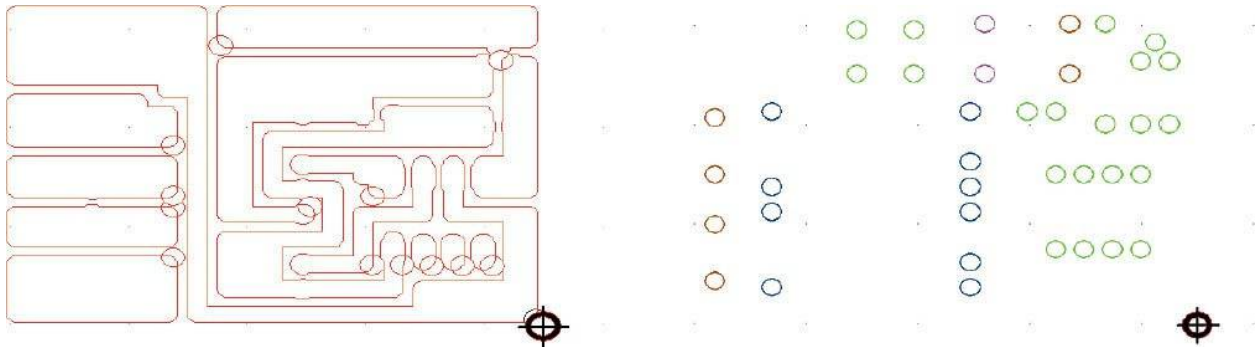


Fig. 6.14 The Etch file (left) and the Drill file (right)

Because the settings dictate a cut depth of just 0.25 mm, when setting the workpiece zero point it is essential that the distance between the tip of the tool and the board is zero. If the tool tip is too far from the workpiece, it won't cut completely through the copper layer. If you find this happening, move the tool to the XYZ zero point, lower it by 0.1 mm along the Z axis, and resave the XYZ zero point before repeating the job.

Switch off the spindle and load the *drill* file. Make doubly sure that the workpiece zero point in the *etch* file precisely matches the one in the *drill* file. The engraving bit is too broad for drilling the holes, so you need to fit a suitable 0.8 mm circuit board drill bit. Raise the spindle to switch Tools, then lower the drill bit until it just touches the board and save this position as the new Z zero point. You can now restart the spindle and execute the drill job.

If everything went according to plan, the result should look like [figure 6.15](#). On the basis of the steps outlined here, you can now experiment with the settings, try out other engraving tools, and generally optimize your approach.

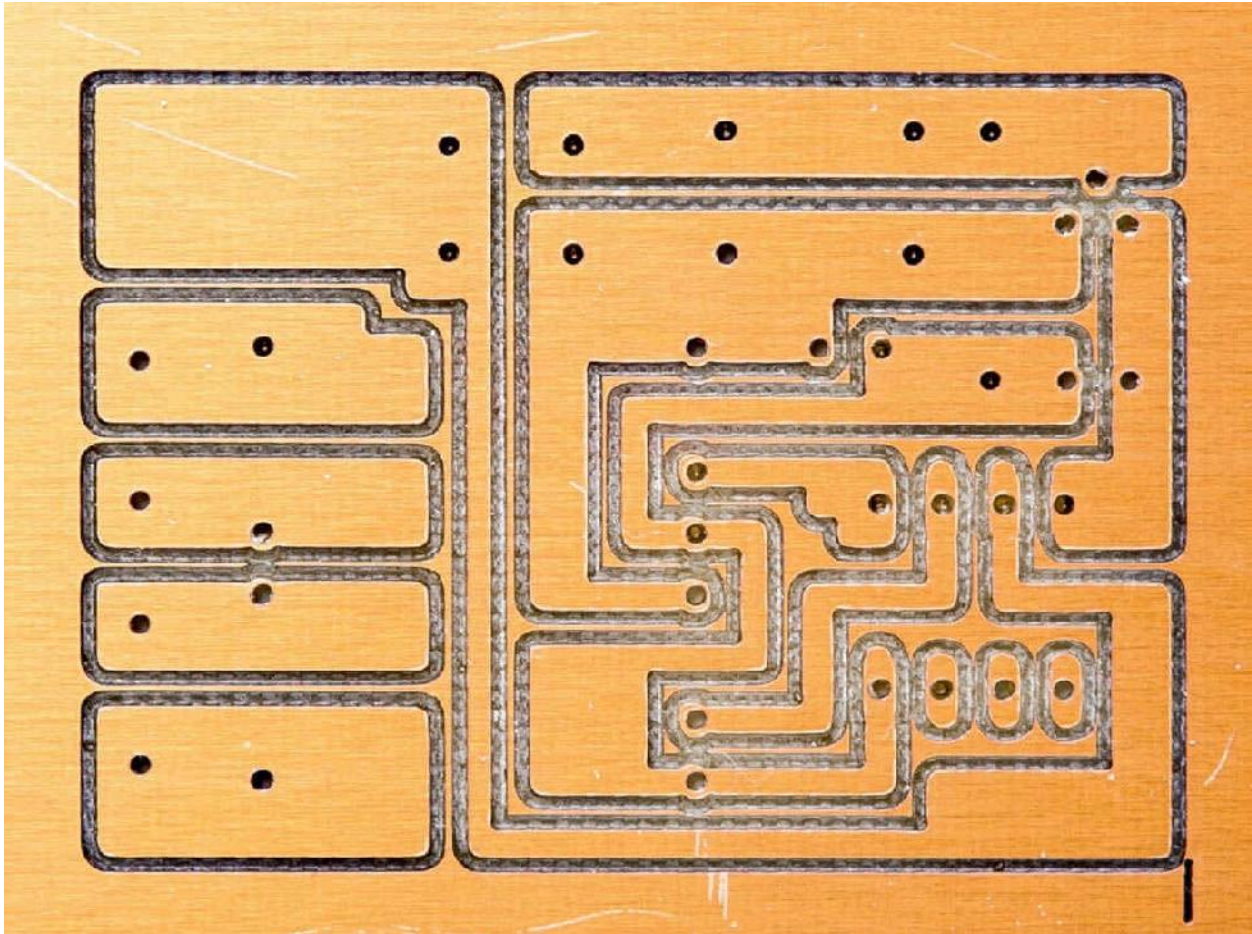


Fig. 6.15 A circuit board cut using a 60-degree 0.2 mm engraving bit and drilled using a 0.8 mm drill bit

6.2 Underwater Milling

Milling in a water bath works exactly the same way as conventional milling, and you have to select your parameters the same way for both methods. However, there are a couple of things you have to watch out for when milling under water:

- The tool mustn't touch the sides or the bottom of the water bath. The sides of the water bath limit the maneuvering space of the spindle, and the machine can only reach all points on the machine table when it is positioned above the level of the sides of the water bath. When positioning the tool, always adjust the Z position last.

- Make sure that there is sufficient space between the surface of the water and the spindle so that the water can't splash onto the machine. This makes it less advisable to insert the tool shank as far into the collet as possible, as you would in a conventional milling situation. The total plunge depth is increased by the depth of the water above the workpiece plus your safety margin. This inevitably causes a loss of rigidity in the spindle/tool assembly and leads to increased vibration.

Caution!

When milling under water, use only a machine with appropriate IP protection certification and a surge protector, or use a low-voltage machine. The illustrations in the following sections are for demonstration purposes only, as our machine doesn't have an appropriate HF spindle. Never use a normal AC110V hobby-grade spindle for underwater milling.

Our underwater test pattern creates a hexagonal piece of carbon fiber sheet with a large hole in the center and six further holes close to its sides. We used a 2 mm 2-flute micro-tooth fish-tail PKD mill. Cutting speeds of 100-200 m/min are appropriate for carbon fiber, so we set the spindle to 29,000 rpm (setting #6), which equates to about 180 m/min. Due to the very small cutting edges, we selected a feed-per-tooth value of 0.01 mm to allow time to evacuate the relatively small chips. The resulting feed rate came out at 10 mm/s. Cut-depth-per-pass values of up to 2 mm are no problem for composite materials like this, but we only needed to cut to a depth of 1.5 mm anyway.

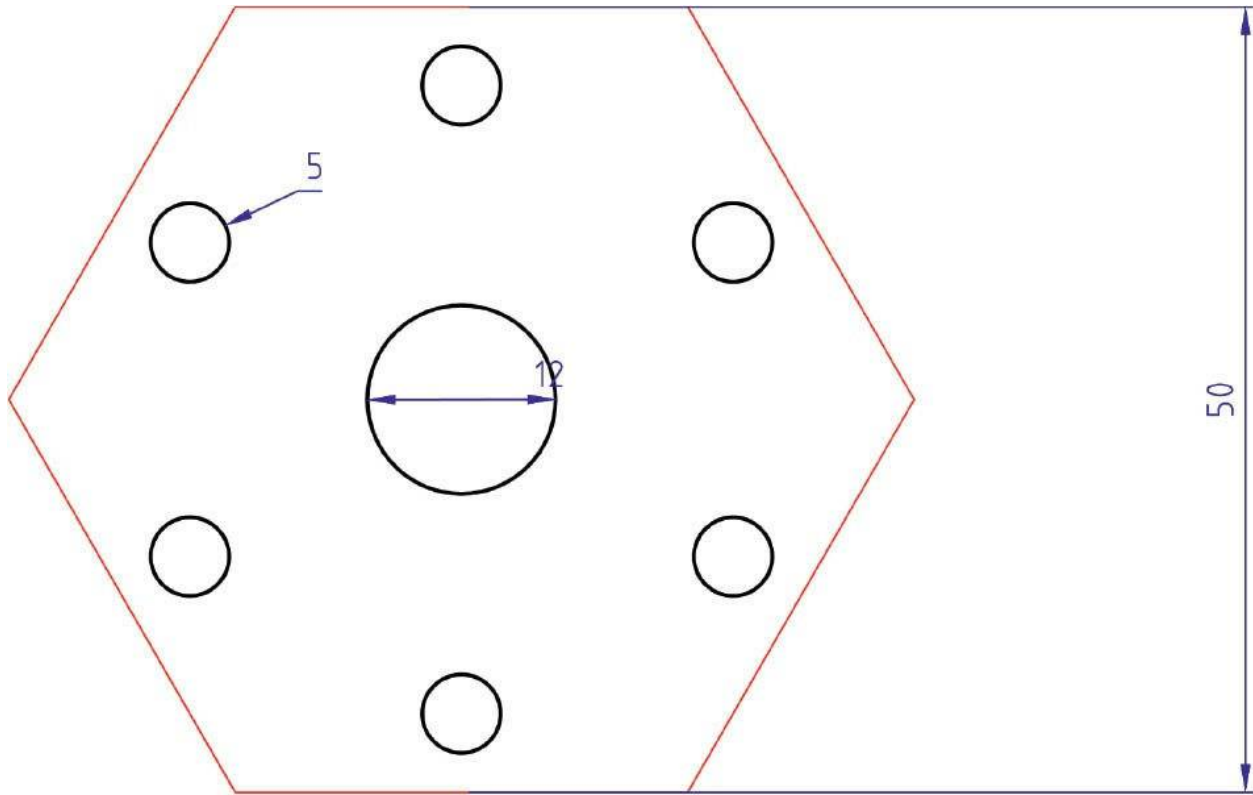


Fig. 6.16 *A 2D model of our sample carbon-fiber workpiece*

- Mount the water bath on the machine table but don't yet add water.
- For a job like this, attach the workpiece to a plastic spoilboard with a thickness of at least 3 mm using double-sided tape. Once the workpiece is mounted, you can stick the spoilboard to the bottom of the water bath. If necessary, mount the spoilboard first and mill it smooth. This degree of precision wasn't necessary for our example, and we were able to iron out any irregularities by increasing the cut depth by a couple of tenths of a millimeter.



Fig. 6.17 The spoilboard with the workpiece attached

- Mount the tool with at least 20 mm protruding from the collet and perform a reference movement.
- Load the milling file and prepare to start the job. If, for example, you want to mill using a plot file and WinPC-NC, don't forget to use the 2D CAM process to determine where you want to mill inside and outside edges, and double-check your parameters accordingly. Because our sample piece is firmly taped in place, the cutting sequence for the edges was of no particular relevance.
- Position the tool at the workpiece zero point and save the coordinates.
- Fill the water bath to about ½ inch above the workpiece and start the job. Adding a couple of drops of detergent reduces the surface tension and prevents the carbon-fiber waste particles from collecting on the surface of the water.



Fig. 6.18 Underwater milling in progress

The tool we used ensured that the chips stayed short and couldn't wrap around the bit. A tool clogged with chips tends to splash water around, which makes a real mess, especially when the water is already full of carbon dust!

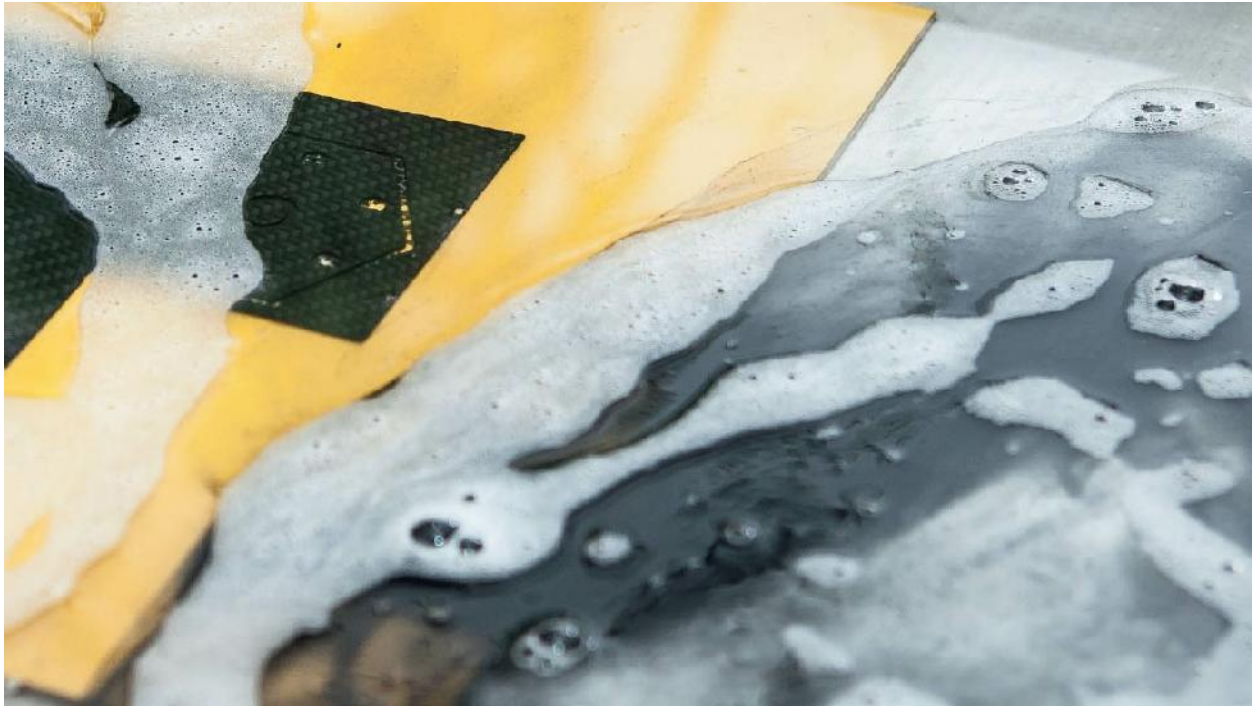


Fig. 6.19 The finished workpiece and the carbon-fiber dust bound in the water

You can see in [figure 6.19](#) that our workpiece wasn't completely cut through. This was because we mistakenly entered a cut depth of 1.2 mm instead of the required 1.5 mm. However, we were able to break it out by hand and finish the edge using a file. [Figure 6.20](#) shows the excellent end result.

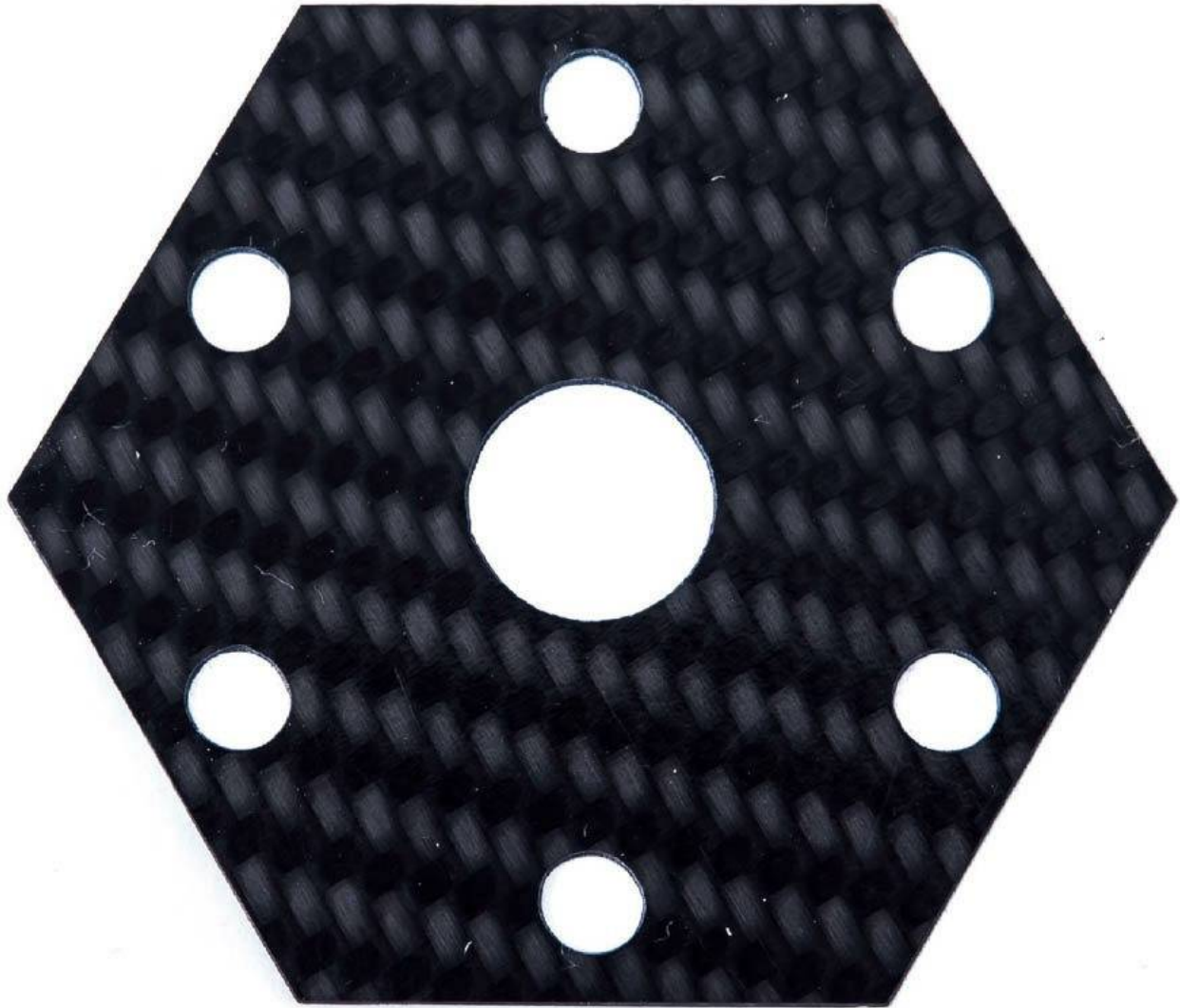


Fig. 6.20 The finished workpiece

6.3 2.5D Milling

All the examples we have looked at so far have been created using 2D milling techniques. 2D techniques cover a broad range of applications using differing cut depths, feed rates, and cut-depth-per-pass values, but don't allow you to alter the cut depth during a pass. Because cut-depth data has to be saved with the model, you need a 3D model to create complex cuts. A 3D model includes data for every point on the model, whether or not it is actually part of the workpiece.

Although a 3D model is used for this type of work, it is still referred

to as 2.5D rather than 3D milling. This is because a 3-axis machine can only engage with the upper surface and exposed sides of a workpiece, but not with its underside or any other hidden areas. In order to access the entire surface, the workpiece itself has to be rotated in two dimensions. In other words, to produce truly three-dimensional components, you need a 5-axis milling machine. The complexity of the machine and the required CAM process make true 3D milling an unrealistic goal for beginners.

On the other hand, 2.5D milling is a useful tool, even when you are making 2D components. For example, if you create a 2D workpiece using a three-dimensional CAD model, the cut depths are an integral part of the file and don't have to be entered separately. Used in conjunction with a CAM application, this approach allows you to control much more precisely how a piece is formed. A 3D model also defines precisely where inside or outside edges and surfaces are located, allowing the CAM application to select a more efficient cutting path and reduce potential sources of errors.

2.5D milling complicates the overall process and, for beginners, even selecting appropriate software becomes a much trickier task. There are huge ranges of CAD and CAM applications available, with specifications that are not always easy to distinguish from one another and prices that range from free to tens of thousands of dollars. 3D modeling isn't intuitive and you will need appropriate training to get started. If you really want to use 2.5D milling techniques, you will first have to consider how to get appropriate 3D models made.

The following sections detail the steps involved in creating a simple 2.5D workpiece. Our sample piece is a medallion with rounded edges and slanted surfaces that cannot be created using 2D processes. We created our model using Autodesk Inventor 2016, but most other 3D CAD applications are just as suitable. We used Fusion 360 for the CAM process and WinPC-NC to mill the result from a sheet of 8 mm MDF.

6.3.1 CAD

Our model has a diameter of 150 mm and is 10 mm thick. Such a model

can either be built up additively (i.e., layer by layer) from the bottom or subtractively from a cylindrical blank. We used the latter approach.

To create 3D objects using Inventor, you have to start by creating an orthographic 2D sketch and then adding a third dimension using a process called extrusion. Once you have created a 3D model, you can add further 2D components in any direction anywhere on its surface, which you can then turn into 3D objects that are added to or subtracted from the base model.

Figure 6.21 shows a 3D cuboid with a square and a circle that have been subtractively extruded. We added an additional layer (shown here in blue) to help us orient the circular cut at 45 degrees to the cuboid's sides. These relatively simple tools can be used to create extremely complex objects.

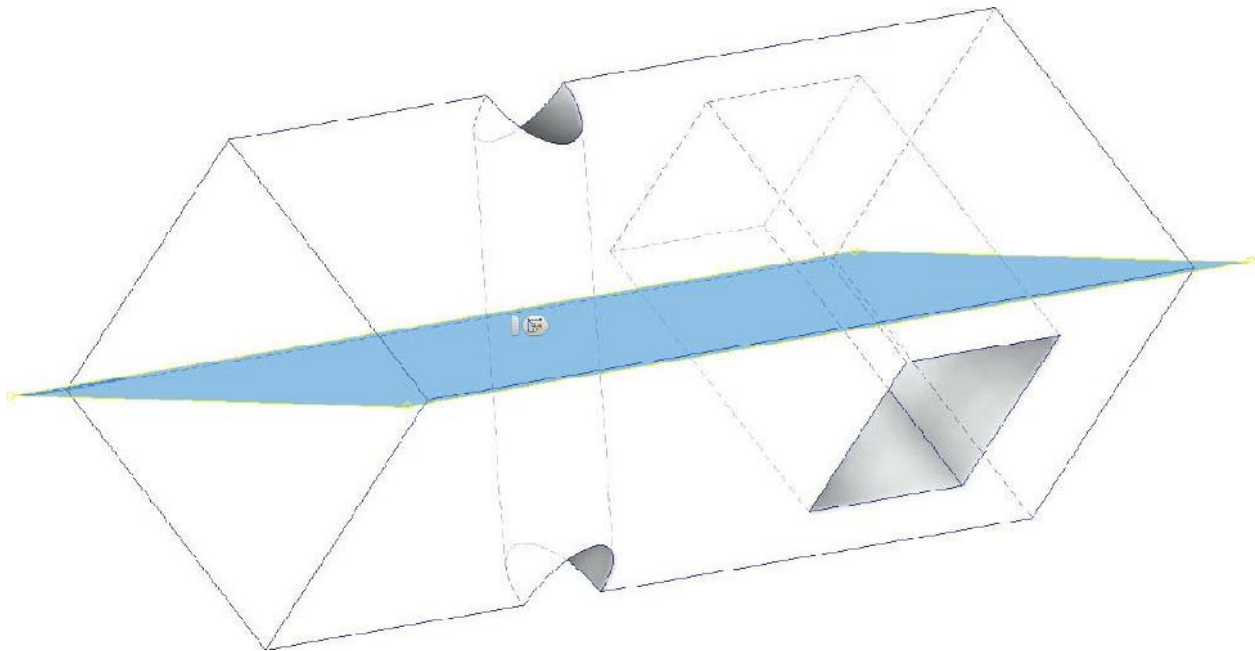


Fig. 6.21 A 3D drawing of a cuboid with two cuts

To create the basic form of our medallion, we drew a 150 mm circle, and then used the *Extrude* tool to give it 10 mm of depth. We then used the *Chamfer* tool to bevel the edge of the resulting cylinder and sketched an additional circle to make a 6 mm deep indentation in the surface. We used the *Taper Angle* option in the *Extrude* tool to taper the hole toward

the bottom, which means the edge of the medallion is sloped on both sides. The result is the basic shape shown on the right in [figure 6.22](#).

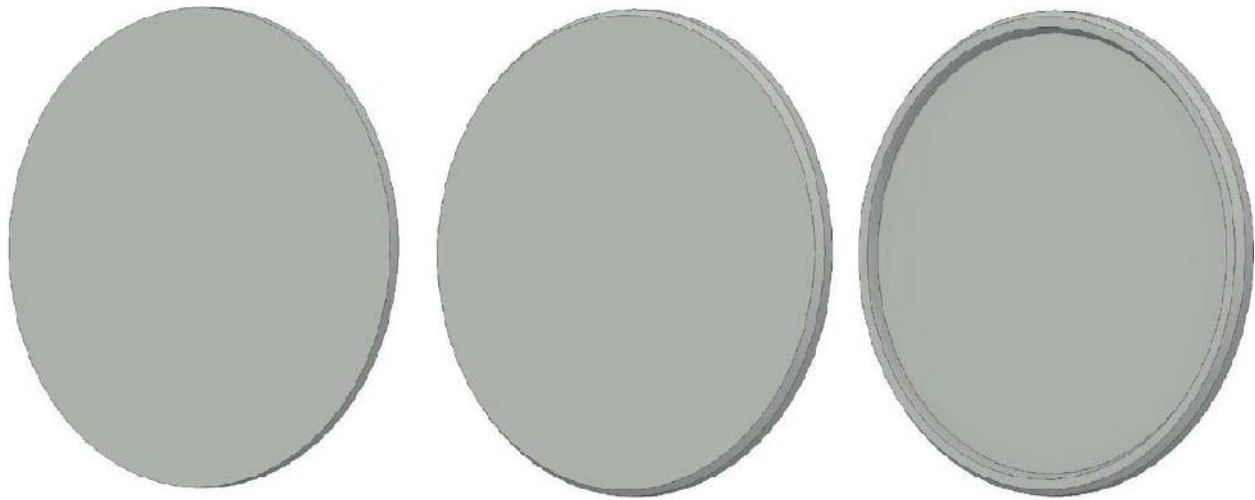


Fig. 6.22 *The basic shape of our 3D medallion*

The next step involved creating some three-dimensional shapes that can be milled using 2.5D techniques (i.e., shapes that can be milled from above).

We added a 6 mm cuboid by extruding a square, and we tapered its surface using the *Face Draft* dialog. We then added some curves using the *Fillet* tool.



Fig. 6.23 *Adding the first object to the medallion*

We added a cylinder by extruding a 2D sketch and added some curves along its Z axis, followed by a 5 mm radius on its outside edge (again,

made using the *Fillet* function). We used the *Circular Pattern* function to rotate it around its own axis using polar coordinates, and then subtractively extruded five 6 mm holes with radiused edges.



Fig. 6.24 *Adding a second object*

We added a second cuboid that bisects the first one, again with radiused edges. Our final object is a rounded extruded rectangle that is tapered toward the top and acts as the base for our milled text.



Fig. 6.25 *The finished 3D model*

The finished medallion has a whole bunch of faces and edges that can only be created using 2.5D milling techniques. In order to proceed, the model now has to be converted to a format that CAM software can interpret.

STL (STereoLithography) is a popular format that works by

disassembling the model into a surface model made up of a huge number of tessellated triangles. Such a model reduces complex 3D objects to a series of identical shapes, making it relatively simple to process using CAM software.

6.3.2 CAM

We loaded our 3D model into *Fusion 360* and used the CAM view to define the tools and parameters used to create our workpiece. Fusion 360 is ideal for this, as it has a built-in CAM processor for UCCNC. With Fusion 360, Autodesk offers a free CAD/CAM solution that is more than adequate for many of the jobs you can do using a Series 2 Stepcraft machine. In fact, it includes many of the features that are usually only available in the more expensive pro versions of AutoCAD and Inventor.

To get the CAM process and UCCNC to talk to each other, we had to define a tool and create a Setup that contained the basic settings for our job. We used a sheet of 10 mm MDF as a blank and, due to the fine text the model includes, a 1 mm end mill. We used settings like those described in [section 4.2](#) for the MDF.

In Fusion, go to *Setup* → *New Setup* and set the operation type to *Milling*. To reset the origin for the workpiece coordinates, select the *Box Point* option in the *Stock Point* drop-down list and select the lower left-hand corner of the workpiece. It is important that the X and Y axes in the application equate precisely to those in the machine.

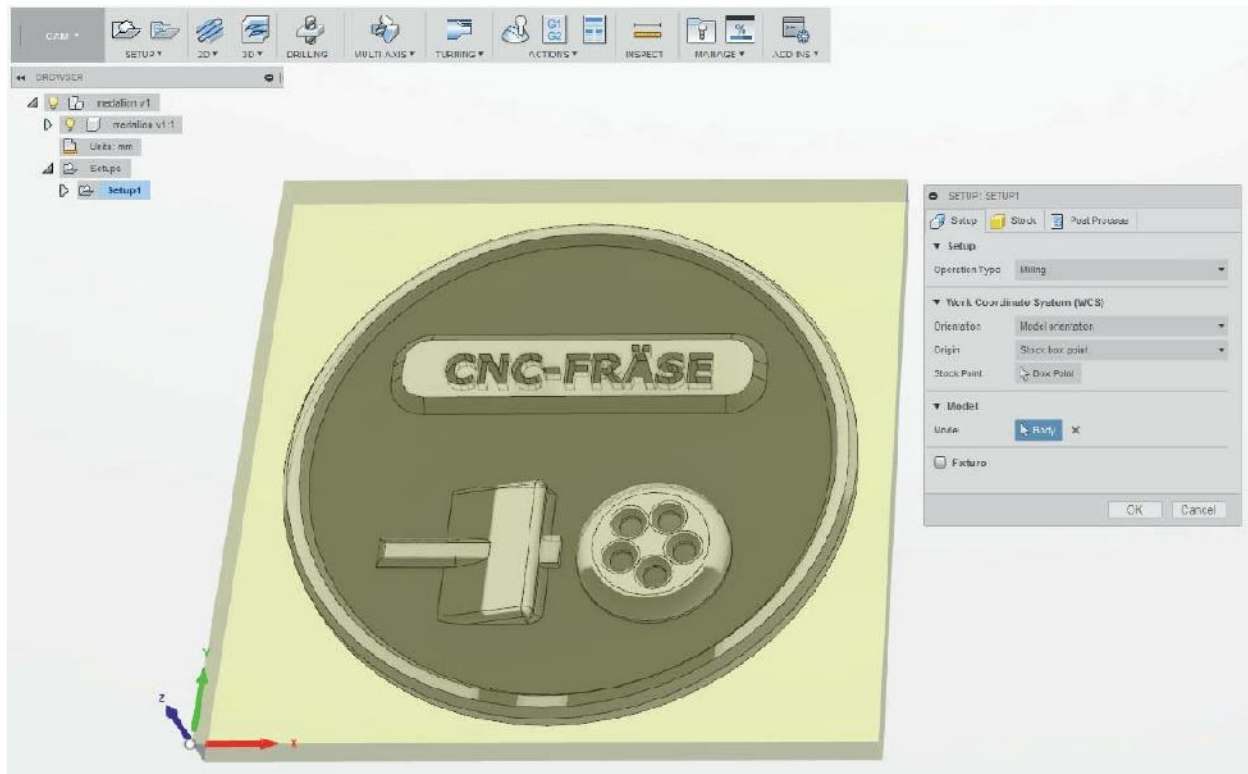


Fig. 6.26 *Creating a new Setup*

The settings in the *Stock* tab define the dimensions for the blank. We defined an offset of 10 mm on all sides to ensure that there was sufficient material available to successfully mill around the circular medallion.

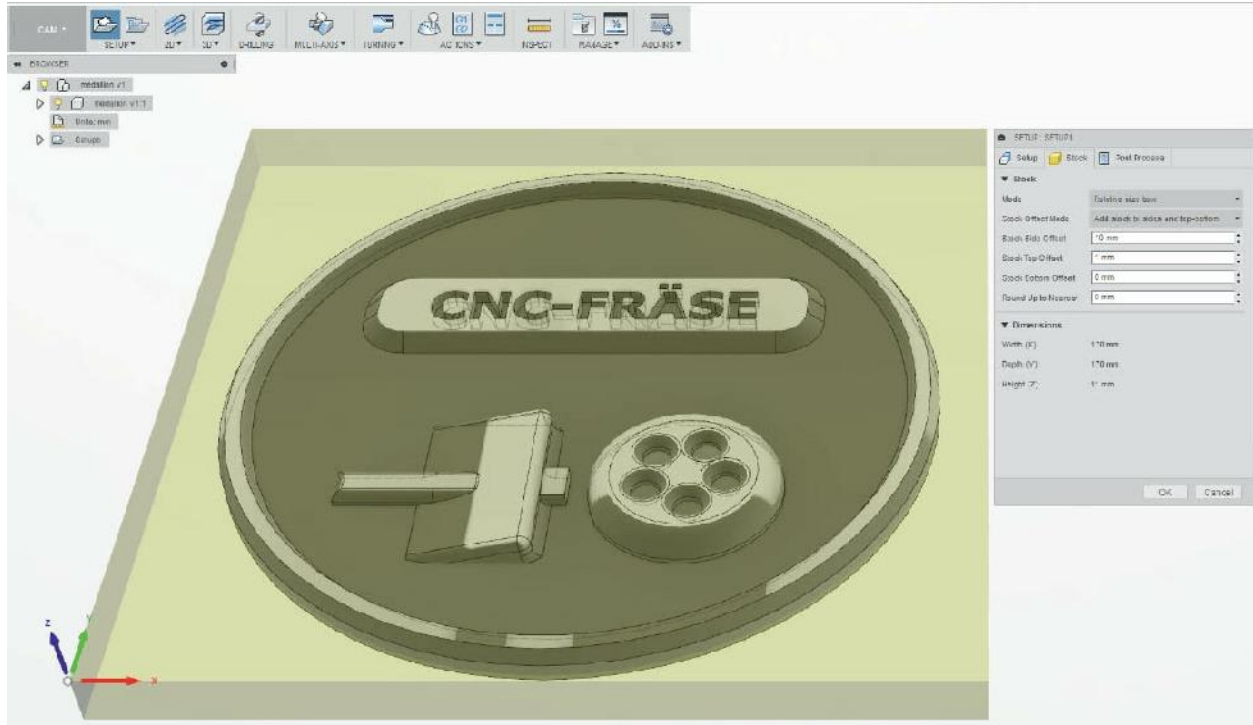


Fig. 6.27 Defining the blank for the job

Enter a name and description for the job in the *Post Process* tab. These labels help you identify precisely which job you are working on, especially if you have multiple versions of a single workpiece with similar names.

The next step involves defining a *New Tool* for use with all subsequent operations. To do this in Fusion 360, go to *Actions* → *Tools Library* and enter appropriate data. In our case, we selected a *Flat end mill*. Enter as much data as possible for your tool (either known or calculated), as Fusion 360 uses complex algorithms based on a broad range of parameters to calculate the toolpath.

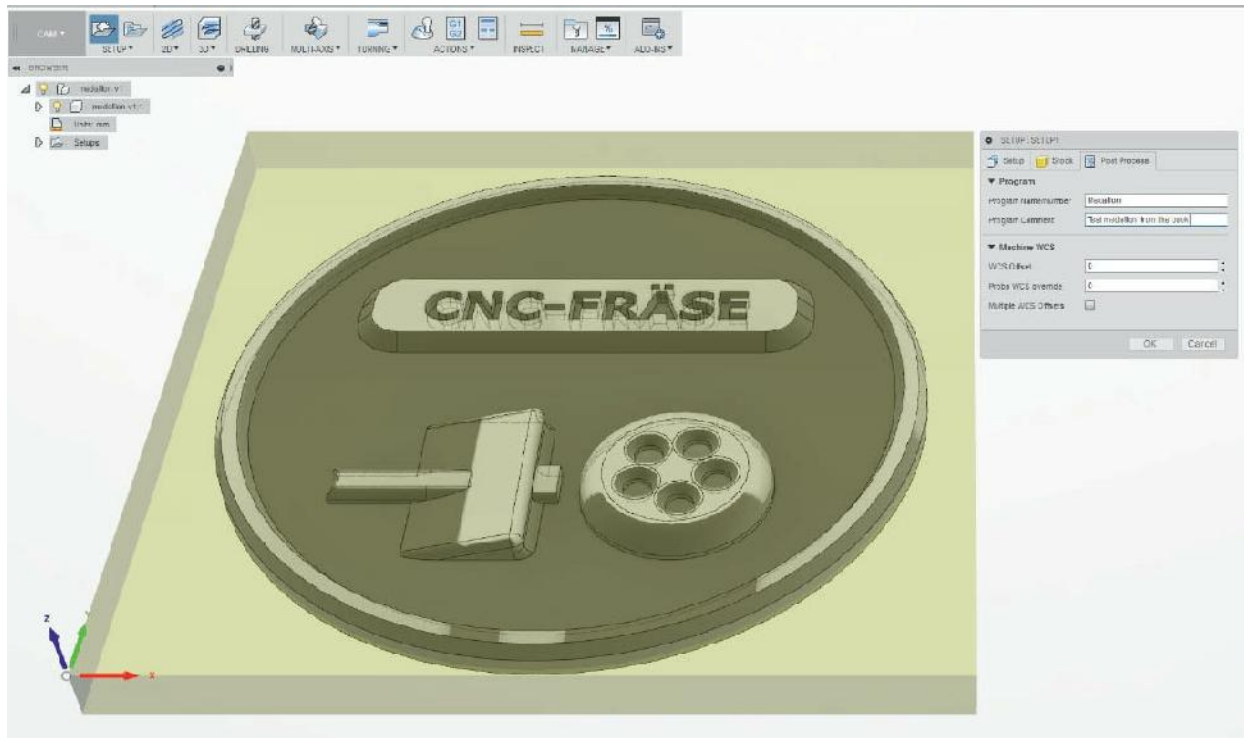


Fig. 6.28 Defining the program name and description for the Post Process

Now the basic settings are complete, the software can go ahead and calculate the toolpath. Compared with other applications, this aspect of Fusion 360 is highly automated and extremely user-friendly. For our workpiece, it involves a single *Adaptive Clearing* operation.



Fig. 6.29 Creating a new tool

In the first tab in the *Adaptive Clearing* dialog, select the tool (see [figure 6.29](#)) and check the *Disabled* option in the *Coolant* drop-down list. Because we already entered the tool data, there is nothing else to set in this tab.

In the next tab, select the model you want to mill. Because our model consists of a single piece, the application automatically selected the option *Body*. If the software displays additional parts next to the *Body* box for a single-part model, click *x* to cancel your selection and start again.

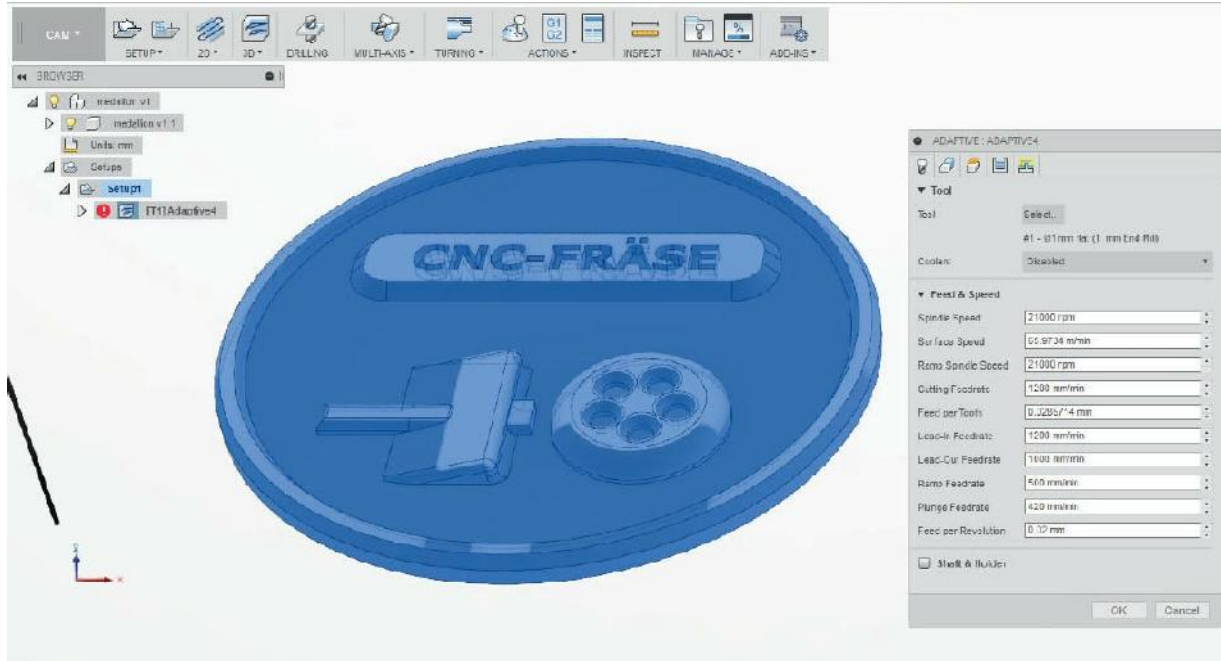


Fig. 6.30 Selecting the milling tool

In the next tab (the fourth from the left), we defined the cuts the tool was to make. The cut depth per pass, or “stepdown,” is especially important and is treated by Fusion 360’s *Adaptive Clearing* function not as an absolute but rather as a maximum value. The software uses varying cut depths according to which part of the model it is milling. Most of the values are set automatically and don’t have to be altered by the user. For the parts milled in roughing mode, the cut depth is set using the *Maximum Roughing Stepdown* setting, while the parts milled in finishing mode use the *Fine Stepdown* setting. The software decides which mode is used for which parts.

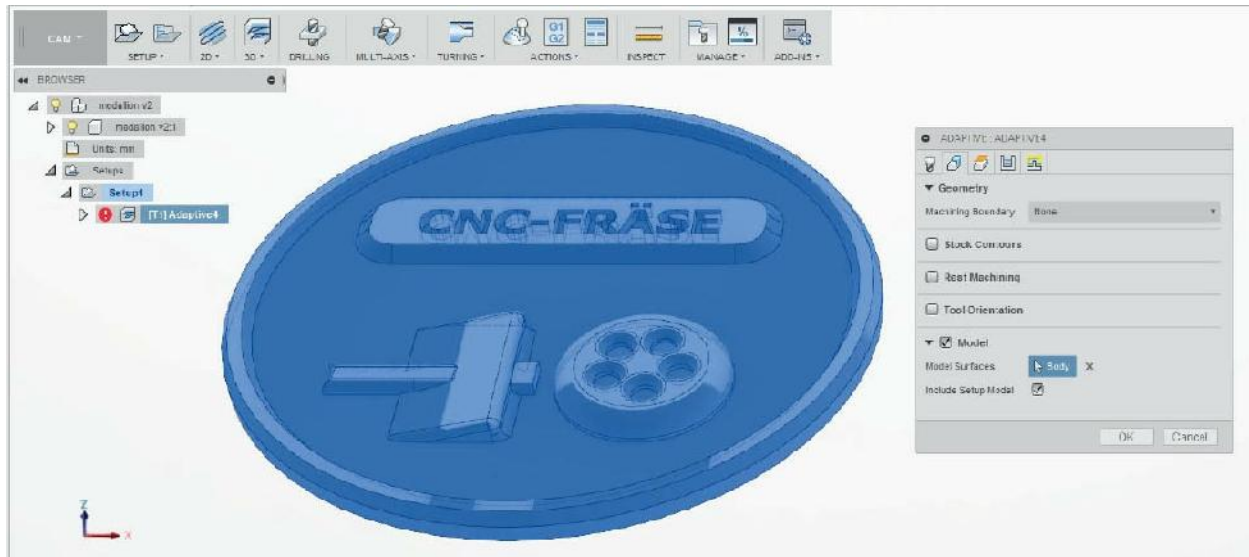


Fig. 6.31 *Selecting the model to mill*

You can decide whether you want to mill the entire model in layers from top to bottom or selected features in their entirety. We used the familiar layer method, selected by checking the *Order by Depth* option. The *Use Slot Clearing* option helps to cut clean surfaces in pockets and slots—in our case in the letters of the text.

Because we were cutting a complete outline around the medallion, we had to define tabs to hold the workpiece in place. To do this, activate the *Fillets* option. Because MDF is relatively soft, we defined broad 5 mm fillets to prevent them breaking due to the pressure produced by the milling process.

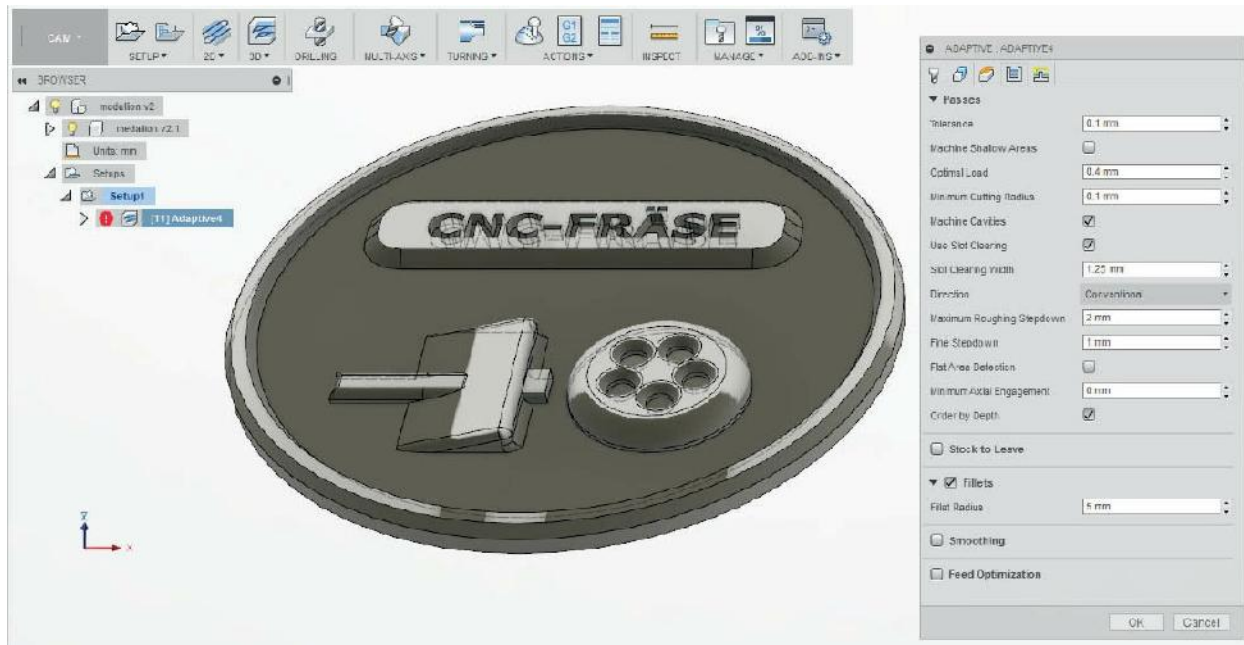


Fig. 6.32 Setting the milling parameters

No further settings were necessary in the two remaining tabs.

Once you have confirmed your settings, the software calculates the toolpath. In our case, this took a while due to the large number of passes required by the narrow end mill. Progress is displayed as a percentage next to the name of the job (in our case *Adaptive4*, as shown in [figure 6.32](#)). If your workpiece comprises multiple parts, it makes sense to rename each to help you keep track. Roughing toolpaths are displayed in blue and finishing toolpaths in yellow.

Note that you can change your *Adaptive Clearing* settings by right-clicking the Setup and selecting the *Edit* command.

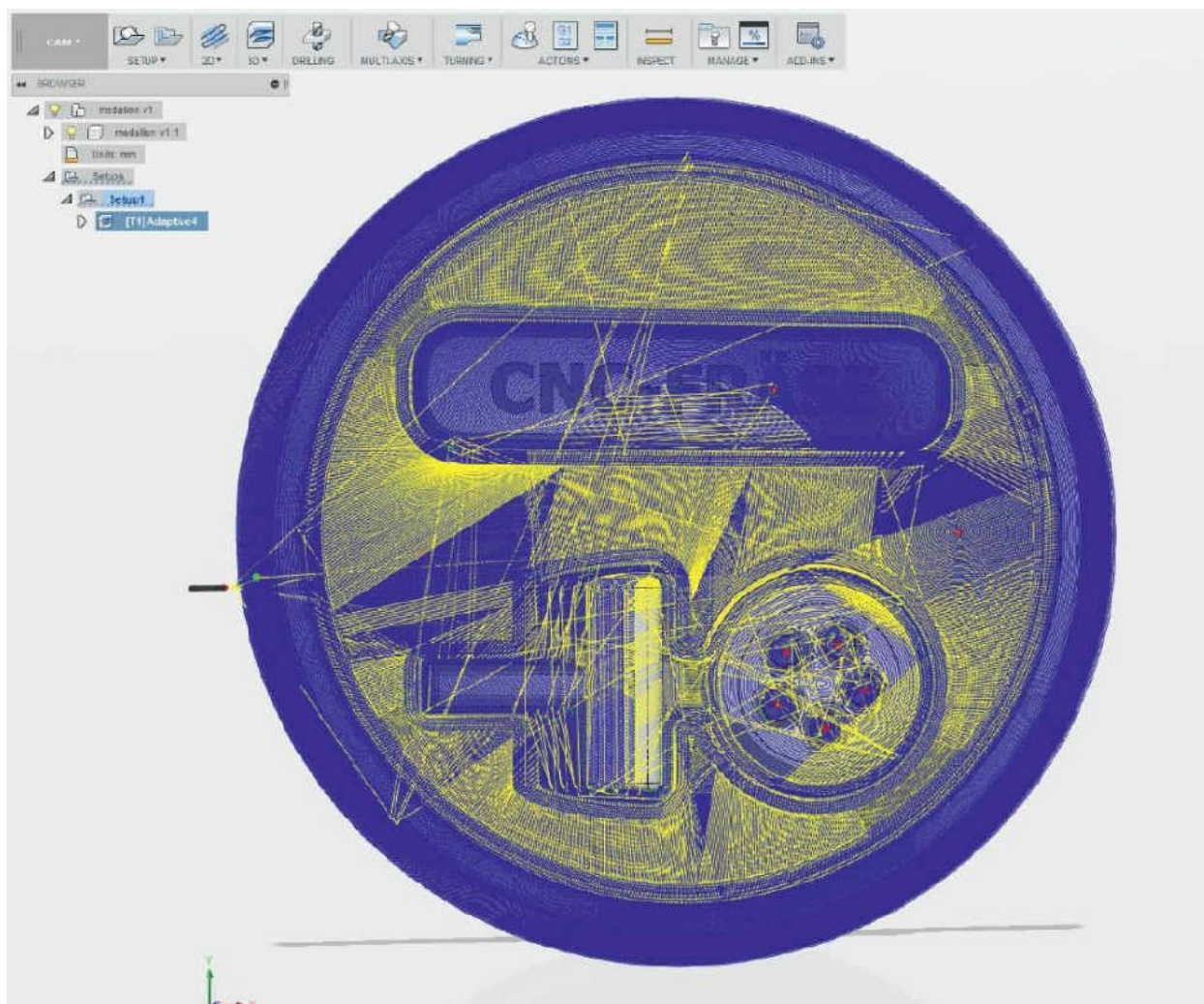


Fig. 6.33 The toolpath displayed in Fusion 360

All that now remains is to output the G-code using the settings in the *Actions* $\rightarrow$ *Post Process* dialog. Set the *Post Configuration* filter to *Milling* to reduce the number of options, then select the Stepcraft UCCNC profile.

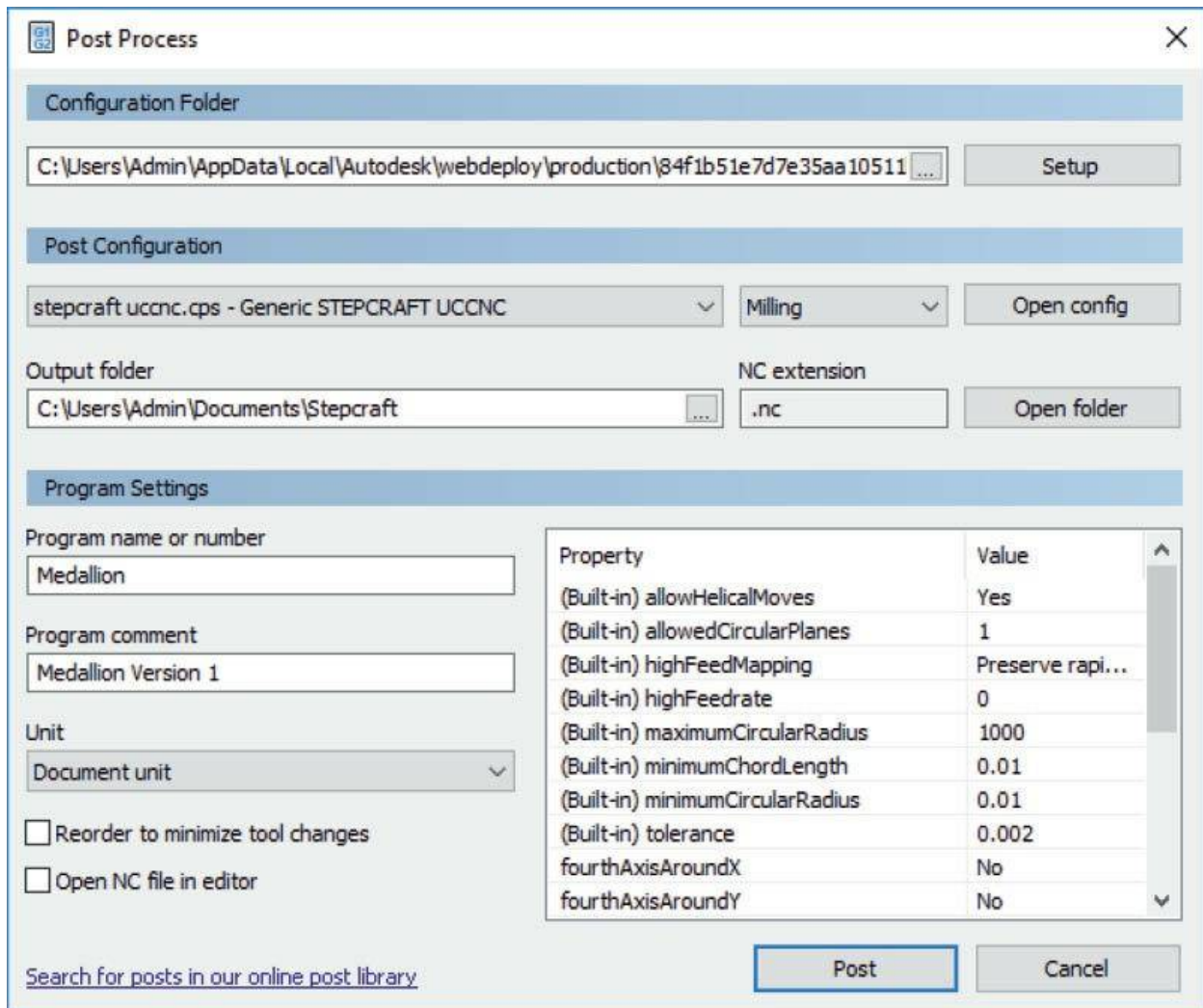


Fig. 6.34 Configuring the Post Process

If the Stepcraft UCCNC processor isn't available, you will have to install it separately. To do this, click the link at bottom-left in the dialog window and download the *uccnc.cps* file stored under *Generic STEPCRAFT UCCNC* on the Web page that follows. Downloading the file automatically copies it to the configuration directory and makes it available to the *Post Process* dialog.

6.3.3 Milling

Because we were using a manually selected spindle speed, we didn't have to set the corresponding value. However, if you are using a spindle that is controlled directly by the machine, UCCNC will select the spindle speed

automatically. Before you load the work file, you have to clamp the workpiece in place and set up the machine.



Fig. 6.35 Fixing multiple blanks to the machine table without using hold-down clamps

- Mount the mill in the spindle.
- Fix the MDF sheet to the machine table. We fixed three 50 × 40 cm (19.69 × 15.75") sheets directly to the slots in the table using bolts and square nuts. To ensure that the medallion didn't move while we cut its outline, we added double-sided tape to the cutting area between the first two sheets.
- Perform a reference movement and load the .din file that contains the medallion data.

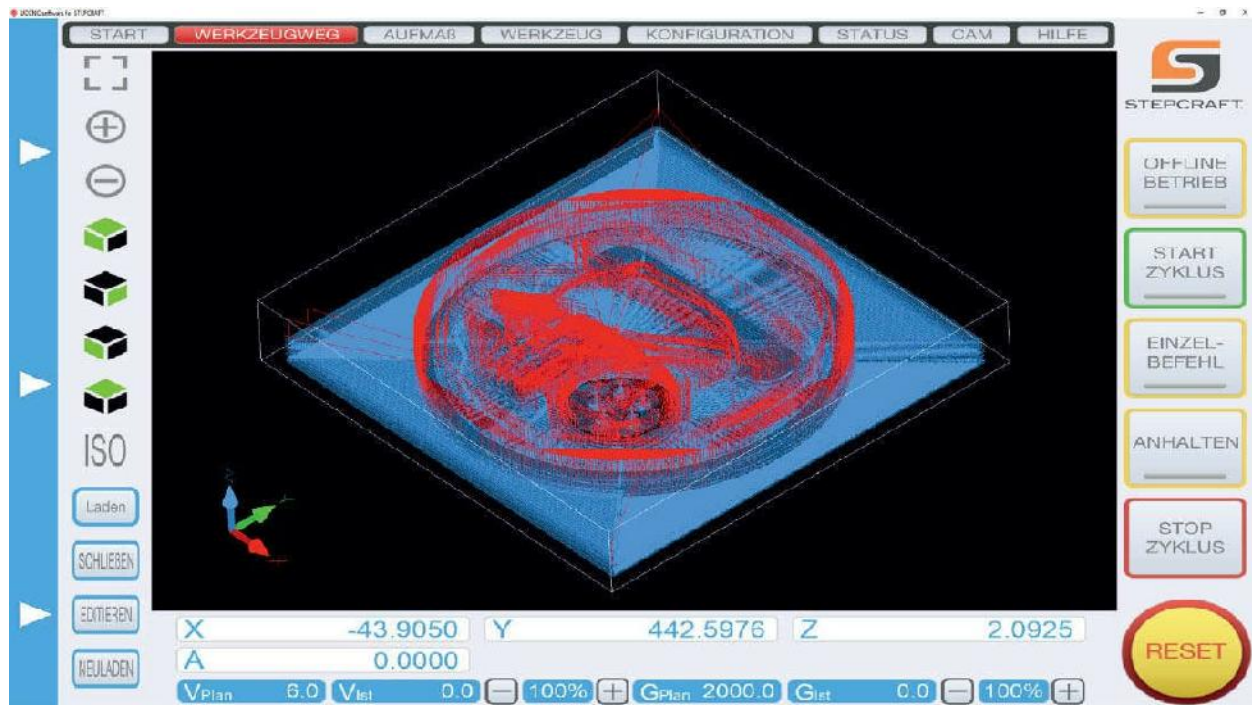


Fig. 6.36 The loaded 3D model displayed in UCCNC

Open the UCCNC G-code file that you saved in Fusion 360. Because the 2.5D process includes toolpath data and data for each separate pass, you don't have to make any additional settings in UCCNC. Loading the *.nc* file instantly displays the finished model ready for milling (see [figure 6.35](#)).

Before you start the job, set the workpiece zero point to match the lower-left corner set earlier on.



Fig. 6.37 *Milling our 2.5D medallion*

It took about two hours to mill the medallion, which was of excellent quality without any extra finishing. The chamfers and curves show some slight stepping, which we could have avoided by tweaking the *Fine Stepdown* option in the *Adaptive Clearing* setting. You can use the procedures described in the previous sections to produce 3D models using any 2.5D-capable machine.

However, this doesn't mean you can't use your 3-axis machine to make 3D models. If you mill a workpiece in a series of steps from different angles, you can easily make undercuts and mill otherwise hidden surfaces. If you take this approach, you need precise reference points so that you can successfully locate the zero point once you have changed the position of the workpiece. You can create reference points either by drilling reference holes or using an edge-finder (or "wiggler"). The workpiece has to be of a shape that can easily be clamped at various angles and has to be small enough to fit beneath your machine's gantry in different positions. Our medallion is too wide to be clamped vertically in our Stepcraft machine.



Fig. 6.38 *The finished medallion*

6.3.4 Slots, Pockets, and Facing

The CAD/CAM process described in the previous sections can be used to cut slots and pockets, and for facing jobs, too. The basic procedure involves making a 2D drawing that includes the edges and precise dimensions of the cut. For facing jobs (i.e., smoothing the entire surface of a workpiece), drawing edges that are oversized by half the diameter of the tool ensures that the cutting edge protrudes from the workpiece while milling and produces clean edges as a result. Of course, this doesn't apply if you are facing a surface that lies between two other raised parts of a workpiece. In this case, you have to enter the precise contours of the milled surface.

The following example explains how to cut a 50×6.5 mm slot in a sheet of 10 mm AlCuMgPb. To simplify the CAM process, we drew the slot as a cut-out in a 60×20 mm cuboid.



Fig. 6.39 *The drawing of our slot showing the workpiece dimensions*

Switching to the CAM module shows the workpiece with the slot already milled. As with our medallion, we had to create a new Setup and define a tool. In this case, we used a 6 mm 2-flute mill with a cut-depth-per-pass value of 0.5 mm, a feed rate of 0.5 mm/s, and a horizontal feed of 1 mm/s.

We used the *2D Pocket* function to define the slot and deselected the cooling option when selecting the tool. We selected the lower edge of the slot in the *Pocket Selections* section of the function's second tab, and set an offset value of -0.5 mm for *Top Height* and *Bottom Height* to ensure that the slot was milled all the way through. Most of the important feed settings are automatically taken from the tool settings, but we still had to activate and configure the *Multiple Depths* option. We set *Maximum Roughing Stepdown* to 0.5 mm. Because the feed rate for aluminum is so low, the whole piece was effectively milled in finishing mode, so we also deactivated the *Stock to Leave* option.

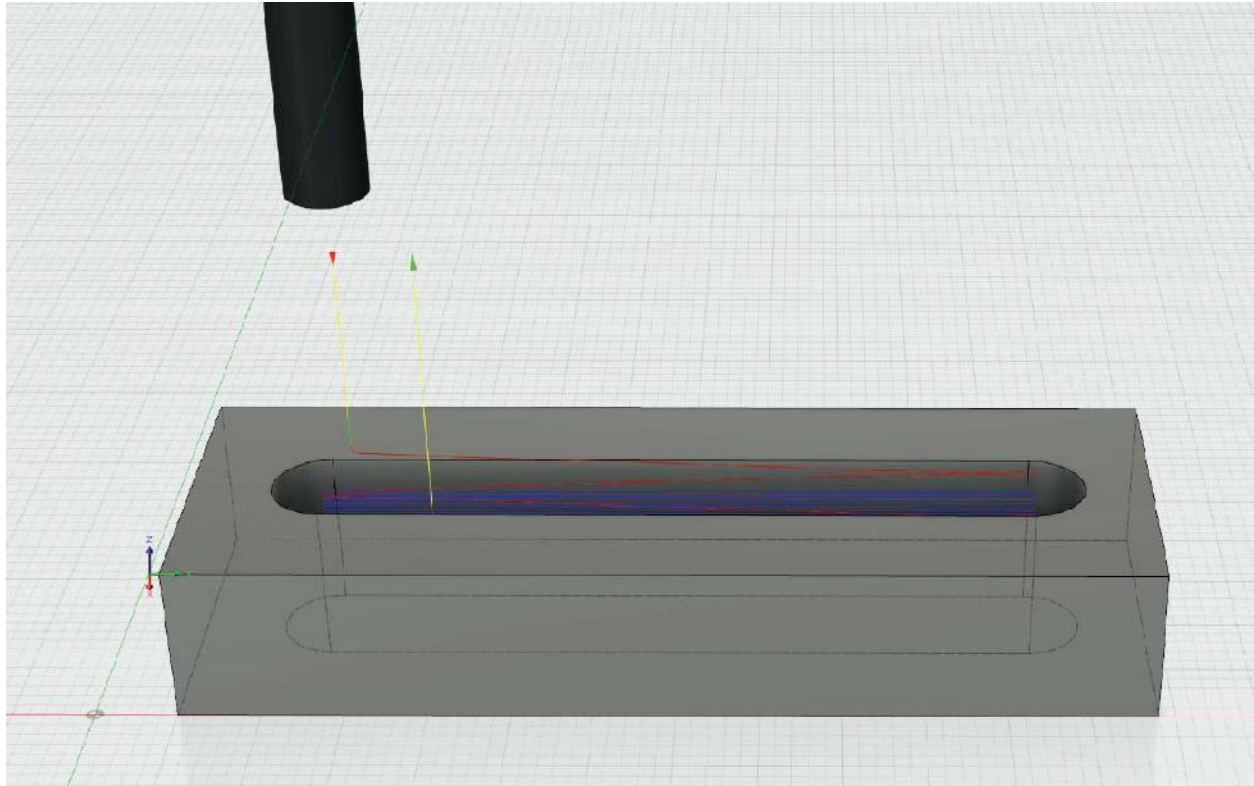


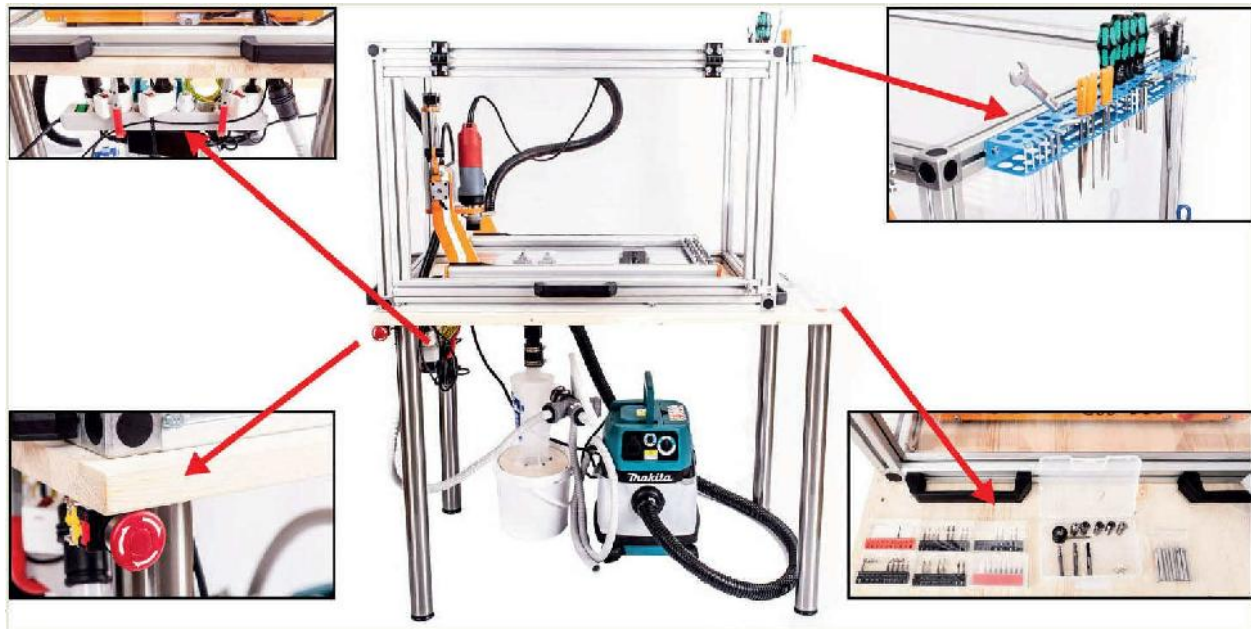
Fig. 6.40 *The toolpath generated by Fusion 360 for our slot*

Always clamp this type of workpiece so that you can mill right through. We used a piece of wood as a spoilboard. Once you have clamped your workpiece in place, perform a reference movement. You can then load the file into UCCNC, set the workpiece zero point, and start the job. See [section 5.4](#) for more on clamps and clamping.

We set the spindle speed to 21,000 rpm (Kress setting #4), giving us a cutting speed of about 400 m/min and feed per tooth of about 0.015 mm/revolution.

7 Looking Ahead

The Stepcraft-2/600 proved to be an amazingly worry-free, all-around package, and was definitely the right machine for me. It fulfilled and even surpassed some of the requirements I outlined in [chapter 2](#), especially when it comes to accuracy. Set up and used correctly, it consistently produced tolerances within a range of ± 0.02 mm, which is pretty incredible and just as good as machines that cost a lot more.



Used together with the add-ons described on this book, the Stepcraft-2/600 makes a potent and versatile machine. But there are other things you can do with a machine like this, too. As well as conventional milling, you can quickly convert your machine for use as a 3D printer or a heat cutter, or you can mount a drag knife for cutting vinyl. There is also plenty of potential for adding homemade upgrades such as a self-cleaning water bath, underwater clamps, accessory lights, and much more besides.

The total cost of the machine with all the upgrades and accessories described in these pages is about US\$4,200, which is quite a lot for a hobby-grade machine. However, I reckon you could reduce this sum by about 60 percent if you use a smaller machine and less complex (or cheaper) add-ons. You may find that you need only a dust-filtering face mask instead of a dedicated vacuum system, or that a simple plastic tray will do instead of a dedicated water bath.

Whether you customize your machine or use it straight out of the box, it is capable of creating much more complex and sophisticated workpieces than the ones demonstrated in this book. The real challenge in raising your milling game lies not with the machine, but in learning to use CAD/CAM software and manipulate the machine's software. The basic skills we have discussed give you the knowledge you need to get up and running and start experimenting with your own ideas.

7.1 Replacement Parts

As time goes on, some parts of your machine will begin to wear out and will require replacement. All the relevant parts are available from Stepcraft, although the ones that break quickest are the ones that are subjected to mechanical loads, such as the bearings, lead screws, and rails. Motors and electronics are less prone to wear and tear.

To help you estimate future running costs, [table 7.1](#) lists the prices of the more common spare parts available for the Stepcraft-2/600 machine. The prices for spares for other models are very similar.

Part	Price (US\$)
X axis linear rail	49.99
Y axis linear rail	86.62
Z axis linear rail	38.71
X axis lead screw	36.11
Y axis lead screw	71.15
Z axis lead screw	27.60
Bearing 604_ZZ	1.55
Bearing F688_ZZ	1.55
Bearing F695_ZZ	1.55
Roller	6.37
Roller holder (8 mm)	9.22
Roller holder (12 mm)	9.49
Lead screw nut	13.18
Drive belt	28.88
Control unit	165.28
Stepper motor ST4118M1404	48.58
Stepper motor ST4118L1804	54.77

Table 7.1 *The prices for a selection of replacement parts for the Stepcraft-2/600 CNC router (prices correct as of February 2017)*

8 Glossary

AFRP

Aramid-Fiber Reinforced Plastic

Backlash

Deviation from the intended position caused by play in the system when changing cutting direction

CAD

Computer-Aided Design

CAM

Computer-Aided Manufacturing

CBN

Cubic Boron Nitride. A refractory boron/nitrogen compound with a cubic crystalline structure. Often used to coat the cutting surfaces of milling tools.

CFRP

Carbon-Fiber Reinforced Plastic

Clamp

Device that fixes a workpiece to the machine table by exerting downward pressure

Claw Clamp

Clamp with one or more claws for extra grip when clamping

Clearance Angle

The angle between the back of a tool's cutting edge and the surface of the workpiece

CNC

Computerized Numerical Control. Method for controlling machine tools using computers

Collet

Also known as a Collet Chuck. Device that locks the tool onto the shaft of the spindle.

Cooling Lubricant

Mixture of a coolant (such as water) and a minimal amount of lubricant for simultaneous cooling and lubrication while milling

Cut Depth Per Pass

Defines the amount of material removed in the Z direction per pass

Down-cut End Mill

See Left-hand Spiral End Mill

Downcut Milling

Milling technique whereby the tool rotates in the same direction as the direction of feed

Dust Removal Classification

Vacuum systems are often classified according to the amount of dust they can remove. Domestic vacuums are not usually classified, even if they are described using terms such as HEPA. The following classes are an indication of the amount of dust a vacuum allows to pass:

Class L < 1 per cent

Class M < 0.1 per cent

Class H < 0.005 per cent

End Switch

When triggered, signals that the end of an axis has been reached

Engraving Bit

Specialized milling tool with an angled tip designed for engraving and milling PCBs

Eurocard

Colloquial term for a printed circuit board manufactured to the European DIN 4149 standard

Excess Material

Predefined amount of material left over for finishing after roughing

Facing

Milling the entire surface of a workpiece to make it smooth

Feed

The movement of the tool along one of the axes

Finishing

Milling technique that removes a minimum of material during a large number of passes. Serves to provide maximum precision and the best possible finish.

Fish-tail End Mill

End mill type that engages easily and produces clean edges. Less rigid than other shapes.

Flat-nose End Mill

End mill with flat cutting edges. Good for face milling and cutting hard materials.

Flute

Slot cut in an end mill to aid chip and waste removal

Fourth Axis

An additional axis that enables rotation of the workpiece. Enables access to otherwise hidden surfaces.

G-code

Programming language used to formulate commands for controlling CNC routers

GFRP

Glass-Fiber Reinforced Plastic

HEPA Filter

High Efficiency Particulate Arrestance Filter. Certified filter type used in many types of vacuum systems.

HF Spindle

High Frequency Spindle. Spindle designed to run at extremely high speeds.

HPL

High Pressure Laminate. Composite material manufactured using pressure of at least 1,000 pounds per square inch.

Left-hand Spiral End Mill

Milling tool with flutes that run counterclockwise. This type of mill exerts downward pressure on the workpiece and prevents unwanted lift in thin materials.

Linear Guide

Mechanical component that aids precise linear movement. High-quality linear guides guarantee minimal deviations of a few thousandths of a millimeter or less, even when subjected to significant force.

Lip Angle

The angle described by the outer sides of a mill's cutting edge

Lost Step

If a stepper motor is subjected to great force in the direction opposite to the direction of rotation, it may not be able to reach its next designated position. If the software cannot compensate, all subsequent movements will be out of step.

Machine Vise

Vise specifically designed for use with machine tools

Machine Zero Point

The point from which all relative positions within the working area of a machine tool are measured. The machine can move the tool precisely to the zero point at any time, from where it can reach any point with consistent repeat accuracy.

MDF

Medium Density Fiberboard

Micro-tooth End Mill

Mill with multiple opposing flutes and diamond-shaped teeth. This design provides multiple, short cutting edges that produce very small chips.

Minimum Quantity Lubrication

Lubrication technique that uses the minimum possible amount of lubricant while ensuring a constant effect

MQL

Minimum Quantity Lubrication

NC

Numerical Control. Early automation process for remote control of machine tools. Predecessor of CNC.

Parallel Port

Legacy computer interface, usually consisting of a 25-pole D-sub socket. Still used as interface between computers and CNC routers, but rapidly being replaced by USB.

PCD

Polycrystalline Diamond. Bonded monocrystalline diamonds used as a super-hard coating for milling tools.

Pinole

Type of machine tool shaft that exerts linear pressure. Enables the generation of feed, even when a machine is being controlled manually.

Pocket

Recess in a workpiece surrounded on all sides by uncut material

Rake Angle

Angle of the cutting face relative to the workpiece. Helps to guide chip flow.

Ramp

Technique that involves penetration of material at an angle. Used when direct plunge not feasible. Requires the tool to be smaller than the area to be milled.

Reference Movement

Predefined tool movement—usually to the machine zero point—used to derive all other tool positions

Repeat Accuracy

Value that denotes the degree of accuracy with which a machine can duplicate movements

Right-hand Spiral End Mill

Milling tool with flutes that run clockwise. Exerts pull on the workpiece and provides efficient chip evacuation.

Rotary Table

Circular machine vise that can be rotated

Roughing

Removal of the greatest possible amount of material within the smallest number of passes. Produces fast results but poor-quality finish. Usually used with a predefined excess of material left over for finishing.

Sealing Air

Air pumped under pressure into sensitive components such as motors. The air escapes through all available openings, preventing dirt and dust from entering.

Service Life

Indicates how long a tool can be used before it needs to be replaced.

Sliding Nut

Nut designed to slide within T-slot Channels. Often used instead of purpose-built T-nuts.

Slot Nut

A nut with a profile designed specifically to fit into the slot on a T-slot machine table

Spindle

A motor with adjustable rotation speed used to drive milling tools

Spindle

General term for the motor used to drive the tool in a CNC router

Spiral-tooth End Mill

End mill with two opposed flutes with differing depth and steepness

Spoilboard

A sheet of cheap, soft material placed between the machine table and the

workpiece. Prevents damage to the machine table during milling.

Stepper Motor

A type of electric motor that can be precisely moved to a preset number of positions within each revolution

T-Slot Machine Table

Machine table with T-shaped slots for clamping workpieces and attaching other accessories

Tabs

Connecting elements that prevent the workpiece from flipping out of the blank when milling a cutout

Tailstock

Holder for large workpieces when milling with a fourth axis

Tool Changer

Device used for automated swapping of tools

Tool Length Sensor

Device for precisely locating the tip of the tool. Also used to derive other locations, such as the workpiece Z zero point.

Tooth

The cutting edge of an end mill or other milling tool

Underwater Milling

Milling technique that submerges the workpiece in water. Dust and waste is bound in the water, which actively cools the tool.

Upcut Milling

Milling technique whereby the tool rotates in the opposite direction to the direction of feed

Water Bath

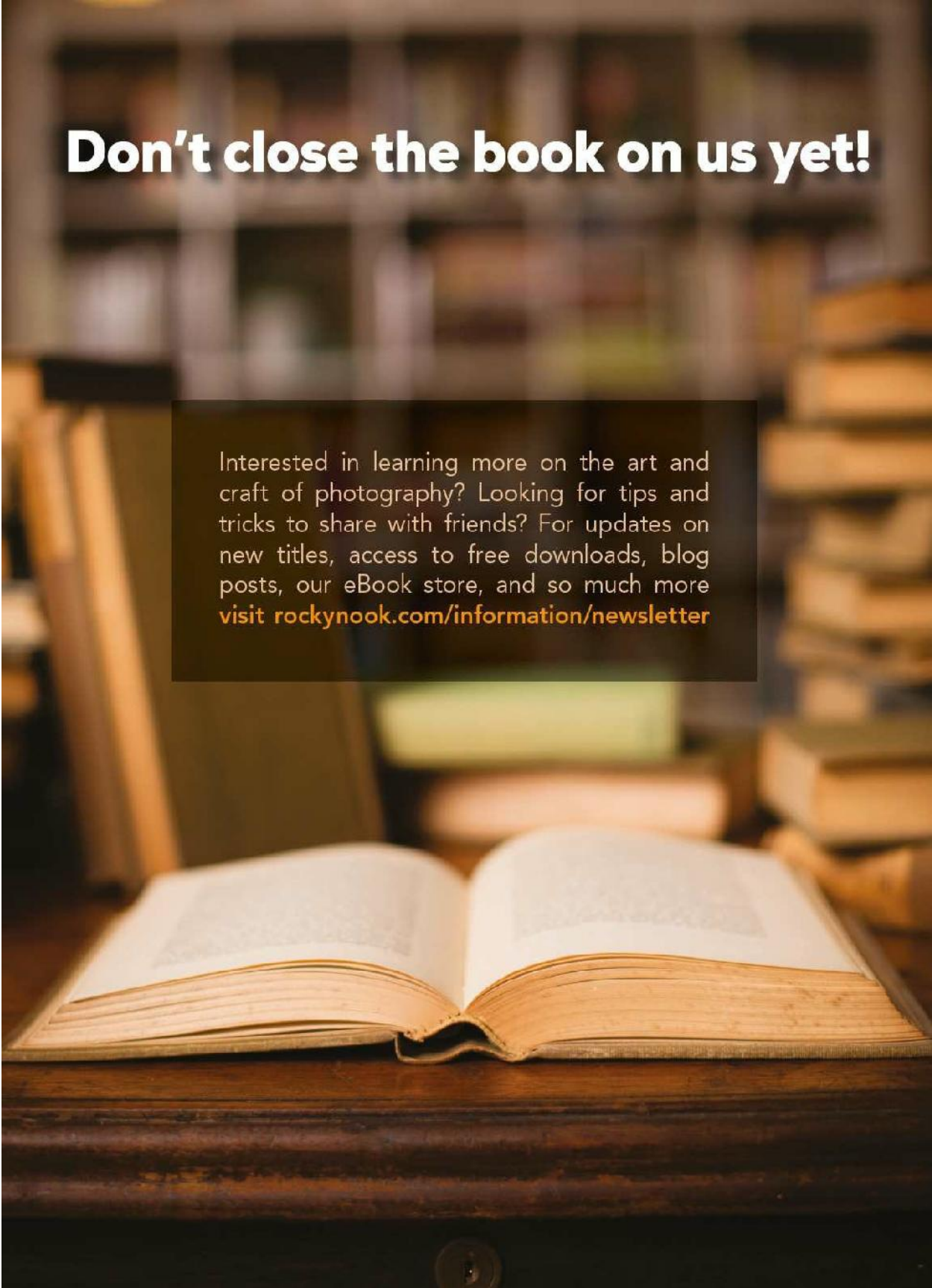
Receptacle mounted on the machine table to hold water for underwater milling

Wiggler

Also known as an Edge Finder. Device mounted in place of a cutter and used to measure the dimensions of a workpiece.

Workpiece Zero Point

A location established relative to the workpiece that serves as a reference for subsequent movements. When milling using CAD/CAM applications, the workpiece zero point has to match the zero point of the model.



Don't close the book on us yet!

Interested in learning more on the art and craft of photography? Looking for tips and tricks to share with friends? For updates on new titles, access to free downloads, blog posts, our eBook store, and so much more visit rockynook.com/information/newsletter



Join us on social!@rocky_nook

rockynook