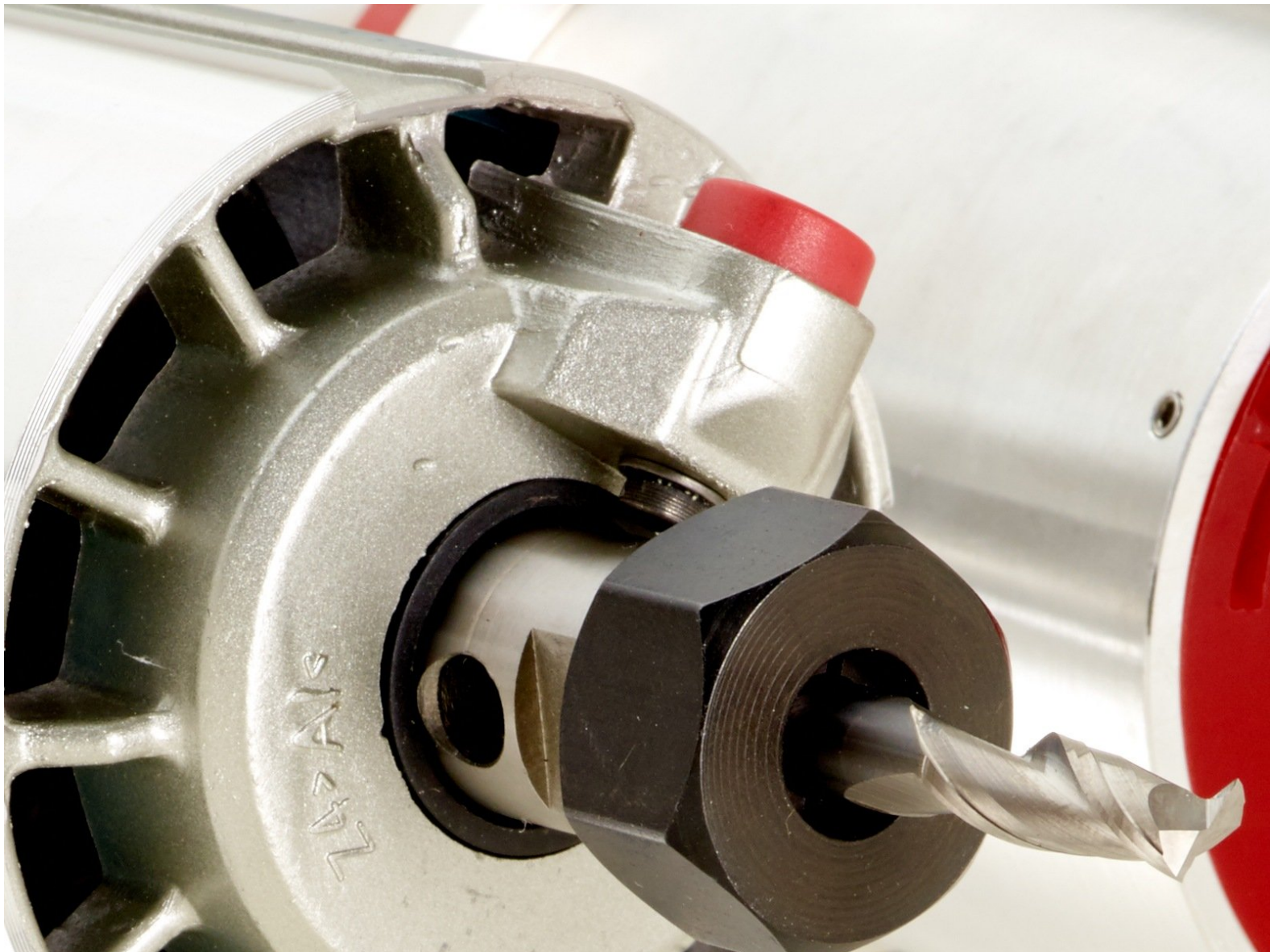




# Creating a Milling File for BoXZY

This is where you'll learn how to prepare and optimize a file for BoXZY to mill.

Written By: Joel Johnson



## INTRODUCTION

This is a mini-manual that expands on topics introduced in [2.3 CNC Milling with BoXZY](#).

**By beginning this manual we assume you have read and understand the following manuals and wiki. If you have not read them, please review them now:**

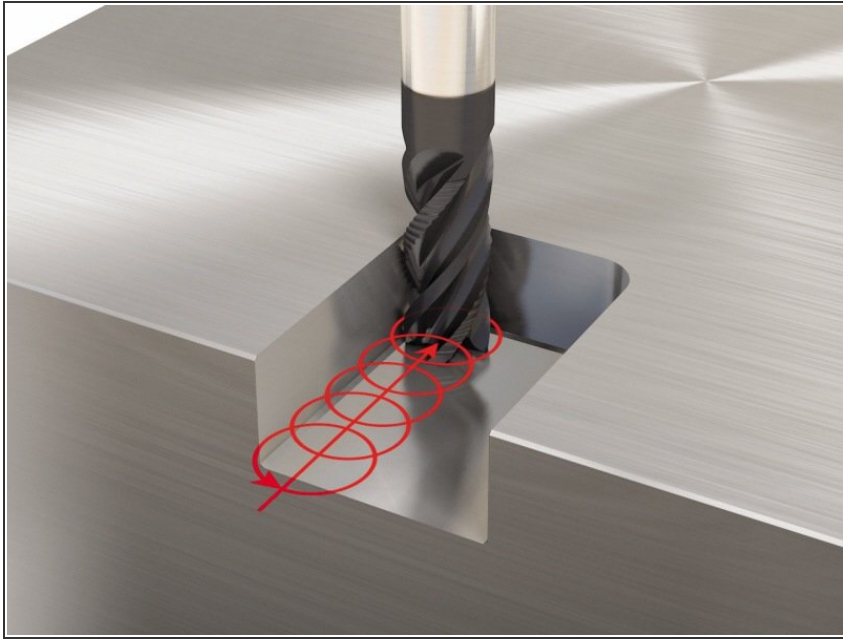
- [0.1 Preparing to Use BoXZY](#)
- [1.0 BoXZY Safe Use](#)
- [1.1 Un-Boxing Your BoXZY](#)
- [1.2 Setting Up Your BoXZY](#)
- [1.3 Using Your BoXZY](#)

## Step 1 — Getting Started with CAM



- In CNC milling you'll use **CAM** Software to create **toolpaths** that are generated into G-code scripts for you to use in BoXZY.
- Why is this different from generating G-code scripts for 3D printing or laser etching?
  - ⓘ Milling requires more user observations and setup than etching or 3D printing. Settings must always be determined individually for each part based on many factors.
- Don't get overwhelmed by the amount of details it seems you need to understand. There are simpler options for the beginner provided below.
- There are countless resources that explain CAM and CNC Milling online, like [CNC Cookbook](#). Feel free to search for further resources. We will also provide some basics in this guide. Don't get lost, you don't have to understand everything.
- ⓘ Milling objects from **foam or machinable wax** will allow you to make mistakes without serious consequences, as long as you work safely and pay attention. Some things will be difficult to understand without experience, so don't be scared to gain some!

## Step 2 — Understanding Toolpaths

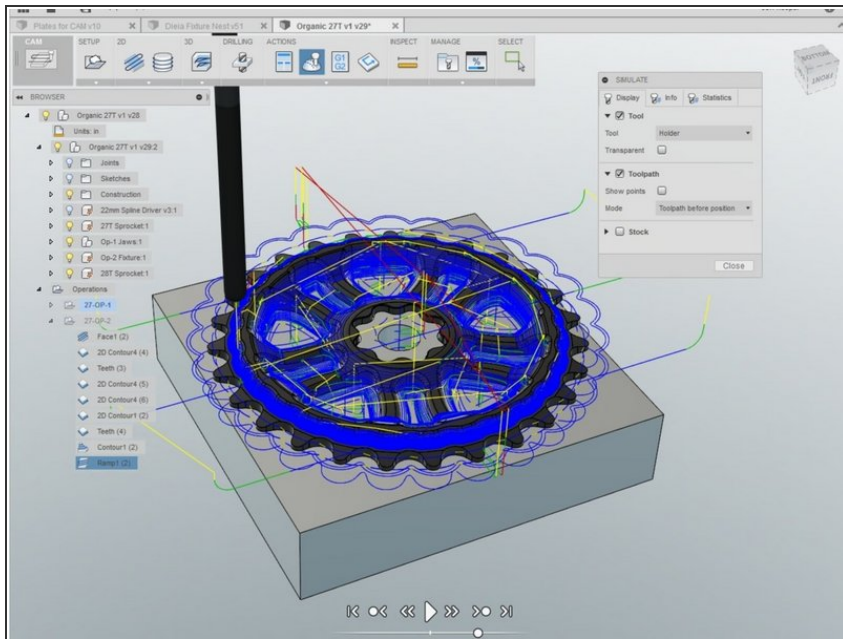


- ❗ A **toolpath** is the path a cutting tool (also referred to as a "milling bit" or "endmill") follows. It allows the cutting tool to cut material at a specified depth and speed, in a specified geometric pattern. There are different types of toolpaths that vary depending on the task. Two examples include "pocketing" and "profile cutting."
- ❗ Before you can CNC mill on BoXZY, you need to understand **Feed Rate** and **Speed**. Speed and feed rate are settings that depend on the material you're working with, the bit you're using, and the amount of material you want to remove per pass.
  - **Feed Rate** is the rate at which the cutting tool travels on the X and Y axes. This is written as distance/time such as mm/second or inches/minute. **Plunge rate** is the feed rate at which the cutting tool moves as it enters the material on the Z axis.
  - **Speed** is the rate at which the cutting tool spins. It's measured in units of RPM, or Revolutions Per Minute.



- A **pass** is a single layer of the milling operation at a given depth. Typically, you use multiple passes to cut out your part, cutting deeper with each pass. As you gain experience you will learn how to optimize your toolpath speeds, feeds, and passes.
- **Stepover** (seen in image) is the distance the endmill moves laterally over uncut material as it mills a shape. Stepover is usually entered as a percentage of bit diameter. Ex: 20% stepover means that 80% of the bit's diameter is over previously cut material and 20% cuts into new material with each path.
- **Depth per pass** is the vertical movement into the material for each pass required to cut the desired shape.

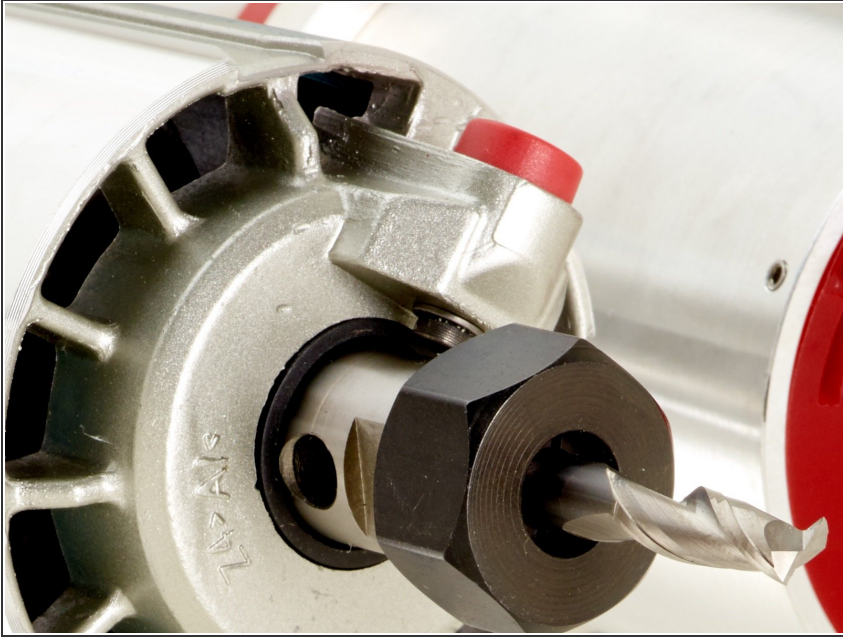
## Step 3 — Resources for Creating Files in CAM



**i** Learning how to create milling files in CAM takes time and practice. Our CNC milling guides mainly focus on what to do *after* the file has been created, but we've included some resources to get you started with designing in CAM.

- **Beginners** can get started with our [Easy CAM Guide](#), which explains how to create milling files using Easel's browser-based CAM.
- **Advanced users** can learn more about CAM with Autodesk Fusion 360 by checking out this [Fusion CAM writeup](#).
  - Be cautious and do not focus on optimizing your CAM creation for ultimate speed until you have mastery of using your BoXZY.
  - Z0 should always be set as the surface of the raw material in your CAM setup to match the real world setup procedure for BoXZY. This way plunges happen only with negative distance/coordinates.
  - To learn more about determining speed, feed rate, stepover, and depth per pass for your milling file, check out this Answer: [Table of CNC feeds and speeds...](#)

## Step 4 — Return to the Milling Manual



- Return to [2.3 CNC Milling with BoXZY](#).