

Preparing a Cut File

[← Back to Onefinity hub](#)

Preparing a Cut File

Design at Home Checklist

1. Confirm project units (**inches vs mm**) - must match your post-processor choice
2. Set up the correct **machine profile** in VCarve (per the [Onefinity + Vectric setup guide](#)):
 1. VCarve → **Machine** → **Machine Configuration Management**
 2. Click **Download from the Cloud** (upper-right corner) → Yes
 3. Manufacturer: **Onefinity**
 4. Series: **Elite Redline** (correct for Gen 2 Redline/Realtime)
 5. Model: **Foreman 48×48**
 6. Configuration: **Defaults** (unless rotary added later)
 7. When saving toolpaths, choose: **Onefinity Redline/Realtime (inch)** or **(mm)** - match your project units
- Backup/manual post-processors: [RealTime CNC](#)
3. Design your part and assign toolpaths - see [Speeds and Feeds](#) for starter values
4. Run **3D preview / simulation** in VCarve - check for unexpected plunges, tabs, and rapid collisions
5. Save your project and bring it to the shop via **USB drive or cloud** (either works)

At the Shop - Export Toolpaths

1. Open your file on a **master VCarve** computer (CNC PC or vinyl cutter PC)
2. Re-check tooling and run 3D preview again
3. **Save toolpaths** → export G-code with the correct post-processor
4. Transfer G-code to the dedicated CNC PC running RTS-X (USB or cloud)

Once G-code is on the shop PC, continue to [Running a Job](#) for simulation, machine startup, and job execution.

[← Speeds and Feeds](#) | Next: [Running a Job](#) →

Revision #4

Created 2026-06-22 21:49:24 UTC by Spencer

Updated 2026-06-22 22:24:56 UTC by Spencer