



CNC Cutting Reference Guide for Foam Materials.

"My experience is you can push EPE to a higher cutting rate over most foam materials with less tool marks"

Starting guidance setup for EPE.

1. RPM Speed: 9000 to 12,000 rpm (I have cut EPE at 15,000 rpm and 150 ipm but that is long straight cuts)
2. Starting Feed rate: 50 to 75 ipm (Once your program is verified you can optimize)
3. Cutting Bit: 4 flute, up-cut bit ½" dia (Inside corners will have ½" radii, can be cleaned up with smaller bit)

Example: If you have a 3" cavity on #1.5 EPE

- Step down .75 to 1.0" at pass
- Based on cutting bit size ½" (Note that inside corners will have ¼" radii and can be clean up. Cutting Bit ¼" Dia. gives 1/8" inside corner radii. Stepdown is determined by bits cutting depth)
- Rough cut and leave small material for full depth clean up bottom and side walls.
- On the finish cut you will find that a small increase in rpm and slow down ipm will give a nice clean finish.

Tips:

- If you notice rips: Speed up rpm and slow down feed rate
- If your melting material slow down rpm and increase feed rate
- Smaller cutting bits like 1/8" and ¼" need to run much slower the longer the bit. (Prevent chatter of bit)
- Be sure your material is secure to cutting table.

*****Everyone's machine capabilities are different so some trial-and-error setups will be required to achieve desired quality and efficiency.**