One Page CNC Quick Fact Sheet

Tool manufactures, such as Onsrud and Vortex, classify our material as 'soft plastic' in their catalogs and information. This is important because the geometry of the bit changes given the material it is intended to cut.

For most applications you want to cut our plastic with an up-cut O-flute bit. Example: Onsrud 63-781 3/8" up-cut o-flute bit.

Down-cut and compression bits should be avoided, although, down-cut bits can be used if there is sufficient room for chip removal.

The most common bit we use is a 3/8" up-cut, single O-flute bit with an overall cutting edge length (CEL) of 3/4" (Onsrud bit 63-781). This bit works well on any material 3/4" thick and under. You want to keep the CEL as short as possible (determined by the gauge of the material) to keep chatter to a minimum. Chucking your tool in the tool holder as far as possible is also important in reducing chatter, but do not go past the shank; no cutting edge should be in the tool holder.

For bits 1/4" – 1/2" diameter we keep the RPMs at 18,000-20,000 and the feed rate at 150-250 IPM (inches per minute). You will find that the RPMs and IPMs necessary to achieve a quality cut vary from machine to machine. Your IPMs are determined by the rigidity of the CNC machine you are using and depth of each cut. (Note: if your RPMs is significantly different then 18,000-20,000, your IPMs will be different also). As a good rule of thumb, you should only cut as deep as the diameter of the tool to achieve maximum quality. For 3/4" thick King Starboard, make two cuts with a 3/8" bit to go all the way through. If quality is a non-issue you can make deeper cuts.

Depending on the quality of the first pass (this varies with the speeds and bits you decide to use) you may or may not deem a finishing pass necessary. If you find that a finishing pass helps, we recommend leaving .015" on the roughing pass.

When profiling, it is best to use a bit with a rounded bottom. This helps reduce swirl marks in the material. Chip relief is also important. Wide profile bits may need to run at RPMs as low as 8000 and IPMs of 100 or less.

A combination of sloping and arcing lead-ins is the best way to remove entry tool marks. We found down cut speeds of 50 IPMs with lead-ins increasing in length relative to depth of cut works great. This reduces pressure on the spindle, bit and material itself. Usually no lead-out is the best option because it minimizes pull on the cut pieces as the bit exits.

When drilling with a CNC machine you may have to experiment to come up with the right speeds for your machine. Using our software's 'pecking' function, we drill at 4000 RPMs, 75 IPMs down feed and .125" down each peck. This method works for both drill and end mill bits. The pecking motion and slow speeds allow for bigger chips to be removed each peck and not wind up on the bit or get jammed into the hole.

If you are surfacing a sheet or machining large pockets there are several precautions you should take to reduce the chance of relieving too much stress in one area and bowing your sheet. If surfacing, always start at the inside of the sheet, this helps reduce the chance that the ends will bow up and cause you to lose vacuum suction. Ideally you would like to surface the sheet on both sides to reduce the risk of the sheet bowing. Always surface and pocket before you cut pieces out so that maximum vacuum is available. Another trick that helps with bowing is to tape the edge of a sheet all the way around to the table. This will stop more suction from escaping at the edges. Jigs and clamps may be used in extreme applications where bowing may result.