VCarve Check Before Cutting

Intro

If you are using VCarve to complete your CAM for either Big Red or the EBF, here are some of the many things you should check prior to running your project. Using a shared PC/VCarve can introduce changes to your file that you didn't intend which can do everything from make your project take a lot longer than expected to ruin your workpiece and endmills.

VCarve Job Setup Screen

You see the Job Setup screen when you create a new file, but not when you open an existing file you created on a different machine then open on the shared computers with VCarve in Leesburg and Purcellville. To open the Job Setup screen after you open your project, click the icon below:



It's important to check that the "Z Zero Position" is set to "Material Surface" and that the "Use Offset" box in the "XY Datum Position" section IS NOT ticked, unless you really want to use offsets. For some reason, an offset added by a prior user can flow through to your project, even after the program is restarted.

🌾 Job Setup
Job Type © Single Sided 1 Double Sided O Rotary
Job Size Width (X): 48.0 inches Y Height (Y): 24.0 inches Thickness (Z): 0.5 inches Units (inches) mm
Z Zero Position Material Surface Machine Bed
XY Datum Position
XY Datum Position Use Offset X: 0.0 Y: 0.0 Design Scaling Scale design with job size
XY Datum Position Use Offset X: 0.0 Y: 0.0 Design Scaling Cale design with job size Modeling Resolution Standard (fastest) 1 million points
XY Datum Position Use Offset X: 0.0 Y: 0.0 Design Scaling Scale design with job size Modeling Resolution Standard (fastest) 1 million points Material Settings

Tool Settings

In theory, the feeds and speeds that you setup on your own computer should flow through to VCarve when you open your file on one of the shared computers. In practice, many of us do a little more work on our files when we get to the shared computer which can cause issues from settings others have put in. It's worth a quick check of all of your tool settings before you start. On each toolpath, click the "Edit..." button and check your settings.

Toolpaths	💽 ? 🛱
Pocket Toolpath	
Cutting Depths Start Depth (D) 0.0 Cut Depth (C) 0.56	inches inches
Tools:	fi U
End Mill (1/8") Select Remove	Edit
Passes: 8 Edit Passe	·s

Feeds and speeds are an art and science to themselves so Members are encouraged to do their research based on their tool, the material they are cutting, what kind of cut they are doing etc. We have recommended baseline settings that will work for most users who are cutting wood or MDF, however it is your responsibility to research and test settings so that you don't break your endmills or damage your workpiece.

These settings are for baselines for wood and mdf.

Pass Depth = the radius (also 1/2 the diameter) of your endmill. So a 1/8 endmill should cut 1/16th per pass.

Spindle Speed = 12,000 for 1/4 and larger diameter endmills, 14,000 for 1/8 and smaller endmills

Feed Rate = 100 IPM for 1/4 and larger diameter endmills, 80 for 1/8 and smaller endmills

Plunge Rate = 30 for 1/4 and larger diameter endmills, 20 for 1/8 and smaller endmills.

Tool Number = Applicable to the EBF only. It is up to you to choose the tool numbers that correspond with which tools you plan to put in each tool holder slot. VCarve will usually show an error if you try to use two tools with different profiles with the same tool number.

Example Settings for 1/8 End Mill

Edit Tool

Name	End Mill (1/8")
Notes	
Tool Type	End Mill \checkmark
Geometry	
Units	inches V
Diameter (D)	0.125 inches
	UI III
No. Flutes	2
Cutting Para	meters -
Pass Depth	0.06 inches
Stepover	0.025 inches 20 🚔 %
Feeds and Sp	eeds
Spindle Spee	d 14000 r.p.m
Feed Units	inches/min v Chip Load 0.0029 inches
Feed Rate	80 inches/min
Plunge Rate	20 inches/min
Tool Number	
	OK Cancel

Example Settings for 1/4 End Mill

	End Mill (1/4")	
Notes		Variables
Tool Type	End Mill ~	
Geometry		
Units	inches •	11
Vaneter (D)	inches	
No. Flutes	2	22
Cutting Paran	meters	┝╾╹╼┥
Pass Depth	0.125 inches	
Stepover	0.1 inches 40 2 %	
Feeds and Spe	peeds	
Spindle Speed	12000 r.p.m	
Feed Units	inches/min Chip Load 0.0042 inches	
Feed Rate	100 inches/min	
Plunge Rate	30 inches/min	
Tool Number	4 *	
	Remove	Apply

Examples:

- 1. Members have added lots of tools to the Tool Database and those tools may not have the same settings applied that you want to use. If you're editing your file on the shared machine, pick a tool you want to use, make sure to check all settings and don't assume that the tools has been setup the way you want to use it.
- 2. For example, on the Purcellville computer's copy of VCarve, one of the "End Mill (1/8")" tools is setup to do a Pass Depth of 1" per pass, pretty much guaranteed to break the endmill the first time it tries to cut 1" deep through material like wood or plastic in one pass. However this tool may have been setup to cut foam where cutting 1" deep per pass is exactly what you want it to do.
- have been setup to cut form where cutting 1" deep per pass is exactly what you want it to do.
 Check the feeds, speeds, plunge rate and tool number are correct. Again, on a shared machine, someone else may have set these attributes to cut a very different material than you are cutting.

Tool Database			
Material Softwood 🗸 🗹	Online 💄	Machin	e Big Blue 🗸 🗹
Imperial Tools A Ball Nose (1/16") Ball Nose (1/4") Ball Nose (1/2") Ball Nose (1/2") Ball Nose (1/2") Ball Nose (1/8") End Mill (1/4") End Mill (1/4") End Mill (1/4") End Mill TEST Imperial Fond Mills End Mill (1/2")Up/Down Sp End Mill (1/2")Up/Down Sp End Mill (1/2")Up/Down Sp End Mill (1/2")Up/Down Sp End Mill (1/4")Up/Down Sp Compression End Mill (1/4") Compression End Mill (1/2") End Mill (1/4") End Mill (1/4") End Mill (1/4") End Mill (1/2") End Mill (1/4") End Mill (1/4") End Mill (1/4") End Mill (1/4") End Mill (1/4") End Mill (1/4") End Mill (1/6") End Mill (1/6") End Mill (1/8") End Mill (1/8") End	Notes	nd Mill (1/8") inches v 0.125 inches eters 1 inches 2 0.025 inches 20 v % eds 6000 r.p.m inches/min v Chip Load 0.0008 inches 10 inches/min 3 1 v	Remove Apply
			Select Close

Additional Job Settings

Before you cut your project, press the "Set..." button on the Toolpath menu.

Toolpaths	💽 ? Ф
Material Setup Set Z 0.0	XY Datum
Toolpath Operations	
	2
	🥼 🦱
J (
III 🚿 🖏 🔀 🔚	
🍟 😩 😩 🎡 🎗	
🌢 🕑 📲 🖥 🛱	
✓ Toolpaths Sheet	1 ~ 兌 🕹
 ✓ Ø Holes Pocket ✓ Ø Cutout ✓ Ø Rounding Edges 	

The following menu looks a lot like the first Job Setup menu, but has some additional settings that should be reviewed:

Toolpaths	[]? "	Ą
🍖 Material Setup		
Active sheet: Sheet 1		
Thickness 0.5	inches	
XY Datum Use Offset Image: Constraint of the second		
Show detailed summary on toolpath tab		
Z-Zero Material Surface Machine Bed		
Model Position in Material		
Gap Above Mo 0.0 inc Gap Below Mo 0.5 inc Model Thickness 0.0 inches Se Modeling Plane Z 0.0 inches	odel ches del ches et	
Panid 7 Gans above Material	4	1
Image: Transmission of the second	1 inches inches	
Home / Start Position X: 0.0 Y: 0.0 Z Gap above Material 0.2	2	
OK Cancel		4

- The "Clearance (Z1)" setting in the "Rapid Z Gaps above Material" setting determines how high the endmill moves above your workpiece between cuts. This needs to be high enough to clear any clamps, screws, or fixturing materials that stick up above the top of your workpiece. If you are using Big Red and you have clamps that stick up above your workpiece by 2", you want to make the "Clearance (Z1)" value over 2".
- 2. The settings in the "Home / Start Position" section govern where the machine will move to before it starts cutting your project. In the example above, the machine will move to its X0 Y0 position first, then move to the first cut. Just like the "Clearance (Z1)" field above, the "Z Gap above Material" settings should be high enough to clear any clamps, screws, or fixturing materials that stick up above the top of your workpiece.