CNC ROUTER ACTIVITY

Mr. Volkmar

North Middle School "GRIZZ BIZ"

Purpose and Goals- The purpose of introducing a CNC router into the Grizz Biz is to get kids hands on experience with the X, Y, Z axis. While becoming familiar with the X, Y, Z axis, students produce a customized wood shop project. For many students the X, Y, Z axis is like a foreign language and difficult for them to grasp the theory and even more confusing knowing how the X, Y, Z axis is the coordinates for all CNC machines.

Students in Grizz Biz run a for profit business. Thus there is a Bookkeeping/Marketing department, Production/Development department and Technology/CNC department. The Grizz Biz takes orders from local customers or online orders. The CNC router allows students to design and sell custom signs.

Introduction – Student first run or introduction in to CNC router, student is to brainstorm a design for a personal house or room sign. This design must have included: 2 line text and one image.

Directions- Student will take a piece of pine and first cut 7 ½ inch by 18 ½ inch (use base fractions).This will be the base for the sign. The student

Parameters-This piece of pine must first be surface sanded in panel sander. Then use ¼ inch cove bit to router outside edge.

Procedures/

-Must pick piece Clear Pine minimal knots and relatively straight

-Joint 2 sides of the pine board, Panel sand, use palm sander (150, 220 grit), apply spray on polyurethane

-Set router bench at proper height (¼" cove bit set to remove ¼ inch of material) will end up with a usable surface of 7" by 18"

-Must have two lines text and image on sketch so Idea for sign is mapped out

-Design sign in V-carve (see attached tutorial)

-Send file to CNC/control panel (see tutorial)

-Upon Competition of sign blow off take to finish room (use lacquer) spray paint lettering and image

-upon paint drying run sign though panel sander to remove excess paint

-router key hole in back of sign

-Recoat light couple coats exterior Poly urethane Front and back

* There isn't a set number of letters for this activity or type of logo but more is though the setup of this file students will get actively engaged in the use of the CNC Router and thus learning operation and practicality of the x ,y, z axis.

V-carve Tutorial

-**Open** ***V-carve*** (software program on desktop or in program file)

-either Create new file -OR- open existing

-will go to **"JOB SETUP "**

Set "JOB SIZE" (inches) <must measure X and Y or length and width>

-"MATERIAL THICKNESS"

(Z – set thickness of material being routered in inches)

-X Y DATUM POSTION

(starting position start center, look red dot in center of screen)

-MODELING RESOLUTION (standard)

-APPEARANCE (pick like material "wood type ")

- Click **"OK"**

Will then go to file operation **" DRAWING TAB"**

-File operations / Icons for **SAVE, COPY,PASTE** and others

-2D CONTROL – Changing view and options (not necessary)

-CREATE VECTORS (options and ways to create things for router to cut out

-Circle (diameter or radius, or manual)

-Ellipse (either click drag or X , Y coordinates)

-Square(X & Z distances of square or rectangle)

-Text - (place cursor where location of text to be or after text created move)

(In text option box decide type text and size and position)

-If image is brought in for exterior location use – Import by using copy/ paste or cut/paste

-Once new image is placed in work area (in file or what you are routering) then select image and then click on "TRACE TO BITMAP " (if you don't do this the image isn't able to be routered by CNC)

NEXT STEPS USE ONLY IF NEEDED!

-TRANSFORM OBJECTS -To move, to size, to rotate , to mirror, to distort, to align

-EDIT OBJECTS

-OFFSET OBJECTS

ONCE TO THIS POINT **"Save As"** (make unique name for drawing describing what it is and date saved Like 2015- 10 – 5 Bobcat logo) Save this **in V-carve folder**.

Go to Right hand side of screen find tab called **"TOOL PATH "** (pin it to the side of screen)

(make sure to select all image to be routering)

-MATERIAL SETUP

Material - Z-zero set thickness in inches < .000 of inch> (must be accurate)

-X , Y DATUM

Set using center of material watch red dot in drawing area

- Check SHOW DETAILED SUMMARY OF TOOL PATH
- Check MODEL POSITION IN MATERIAL
- Check GAP ABOVE inches 0.00
- Don't check GAP BELOW
    - In Box Put THICKNESS OF MATERIAL in inches

RAPID Z GAPS ABOVE MATERIAL

- CLEAREANCE (Z1) .1 inches (start above material)
- PLUNGE (Z2) .1 inches (same amount so contacts material)

HOME/ START POSITION

-X=0 Y=0 Z= 0.1 (so starts above material)

TOOL PATH OPERATIONS (where you decide tool path for procedure you preforming)(this will chang based on project

-PROFILE TOOL PATH- cutting tool path like sign or engraving

-POCKET – for cutting out pocket or center of box

-DRILLING – for series holes like cribbage board

-ENGRAVING- for signs with larger images want texture in the routered area

-INLAY- for cutting excess around letters so have effect of raised letters

-V- Carve – multi purpose tool path that can be used for signs with image and flat bottom in cut

-FLUTTING – for cutting flutting or grease catch path in cutting board

-TEXTURING – for cutting with texture in cut

-PRISM ENGRAVING – for a 3-d like cutting with options based on bit used.

The tool path choosen will require you to set a **2D-PROFILE TOOL PATH**

-With start depth inches > **START DEPTH** ____ inches usually 0.1

-then **> CUT DEPTH** ___ inches around ½ of thicknes of wood but also varies with router bit

-TOOL BITS:- Select Tool Bit before selecting tool path (usually V-Bit or End Mill , must know diameter of bit)

Once bit selected form list in program (If bit isn't list notify instructor so can be set up)

-Most tool paths have to be set up Look Below:

-MACHINE VECTORS (could change with materials) check one of 3 circles

-__ OUTSIDE RIGHT

-__ INSIDE LEFT

-__ ON (these 3 indicate where you want bit to cut)

-ALLOWANCE OFFSET

-__USE VECTOR START POINT (leave unchecked, don't optimize)

-LAST PASS

-__ DO SEPARATE LAST PASS (if checked must decide allowance & if cut in reverse direction)

SERIS TABS (these are options to cuts can leave default)

-TABS, LEADS, RAMPS,ORDER,CORNERS

-SAFE Z 0.10

-___**PROJECT PATH ONTO 3D**

-**VECTOR SELECTION (MANUAL)**

-**TOOL PATH NAME** (Create tool path name) (MAKE SURE SPECIFIC)(2015-10-7 Bobcat)

-**CLICK (CALCULATE**) this make tool path

Then **– PREVIEW TOOLPATH**

-Click Preview to see what cut looks like

Then **CLOSE**

BACK TO ORIGINAL" **TOOL PATH OPTIONS"**

- Go to bottom row click on option that look like floppy disk (**SAVE TOOL PATH**)
- New box shows up **(CHECK NO BOXES!!)**
- MAKE SURE CORRECT CUTTER BIT SHOWS UP
- POST PROCESSOR find **CNC SHARK-USB CONOUR (INCH)**
- CLICK **SAVE TOOL PATH** AGAIN (same tool path as earlier)

OPEN from programs or desktop

-***SHARK CONTROL PANEL*** **(new program on desk top)**

-Click on (**LOAD G CODE**)

-Select Tool path name

- then pop up **(SET UP VIRTUAL ZERO (NO))**

FIND CENTER OF PIECE TO BE CUT secure work with carpet tape on bottom work piece and slide clamps

Go to upper right of screen click **"JOG"** tab

-use **X or Y** buttons to move router bit to center of piece

- click **Z** button to move router bit till just touches work

Go to upper LEFT HAND SCREEN click on (**SET**) Button

-Make X=0 Y=0 Z=0

Then Click tab **(XYZ 0)** tab

On lower left hand side of screen click **(RUN FILE)**

POP up Screen (**TOUCH PLATE**) click on **(CANCEL**)

Then file load icon will pop up If ready to run click (**OK**)

IF PROBLEM BE READY TO CLICK **(E STOP)**

If need to move Clamp or work to continue later click **(PAUSE)**

HIGHLIGHTED words are commands on the V-Carve or Shark Control Panel

-