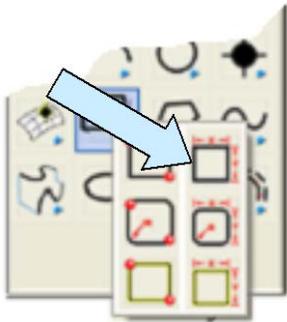
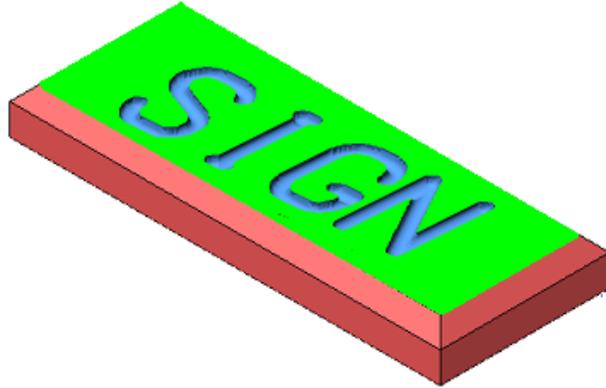


# KEYCREATOR 3D Direct Modeling Software

## KeyCreator Lesson KC8191

### Machining Text on a Plaque

In this exercise we'll review how to quickly change text letters into a tool path that can be used to drive a ball mill.



Start with a new file in View 1. (The Top View.)  
Click on the CREATE RECTANGLE BY WIDTH HEIGHT Icon.

Use the MidCtr Anchor Option. Type 7 for the Width and 2.75 for the Height. Using the KeyIn Option, hit the ENTER Key three times to place the rectangle centered on the origin.

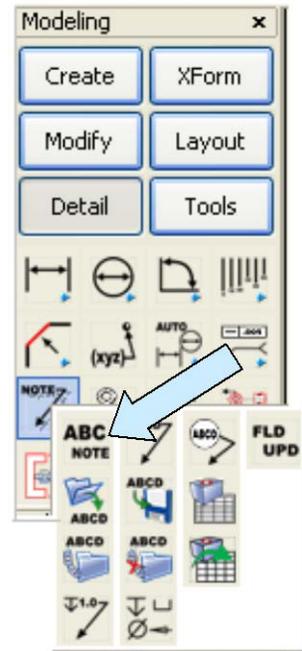
Click on the CREATE NOTE BY TYPING Icon.

A Dialog Box appears.

Type the word "SIGN" using all capital letters.  
Click on the Formatting Option on the left side of the Dialog Box. Type 1.5 for the Text Height.

Use the Center Anchor Option and the Center Justification Option and hit the ENTER Key.

Using the KeyIn Option, hit the ENTER Key three times.



Your screen should look like this:

# KEYCREATOR 3D Direct Modeling Software

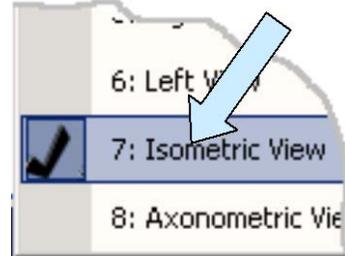


To use the letters as a path for a cutter, we must first change them into regular geometry. To do this, click on the GENERIC BURST Icon.

Then drag a selection window completely around the text and hit the ENTER Key. This converts some of the text to polylines and some to lines or splines.

Use the GENERIC BURST Function one more time and drag another selection window completely around the text.

Now, switch to the Isometric View. (View 7.)

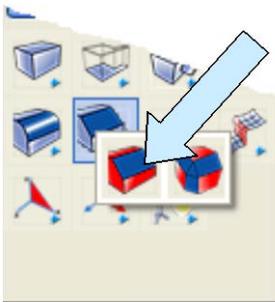


Click on the EXTRUDE Icon. A Dialog Box appears. Type 0.75 for the Length and hit the ENTER Key.

Select the rectangle and hit the ENTER Key. Then, click on the downward-facing vector.



Your screen should look like this:



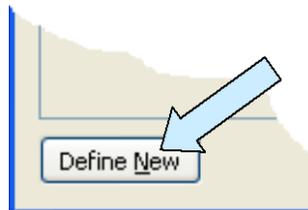
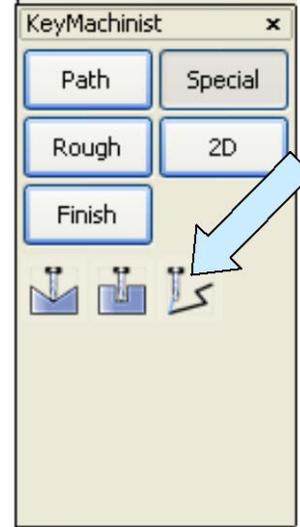
If you want a fancier plaque, you can use the SOLID CHAMFER Tool to add 0.25 chamfers on the four, top edges.

# KEYCREATOR 3D Direct Modeling Software

## Creating the Tool Path

Creating the tool path for the ball mill is very simple.

Click on the GEOMETRY TO TOOLPATH Icon.



A Dialog Box appears, requesting that you define a tool. Click on the Define New Button at the bottom, left corner.

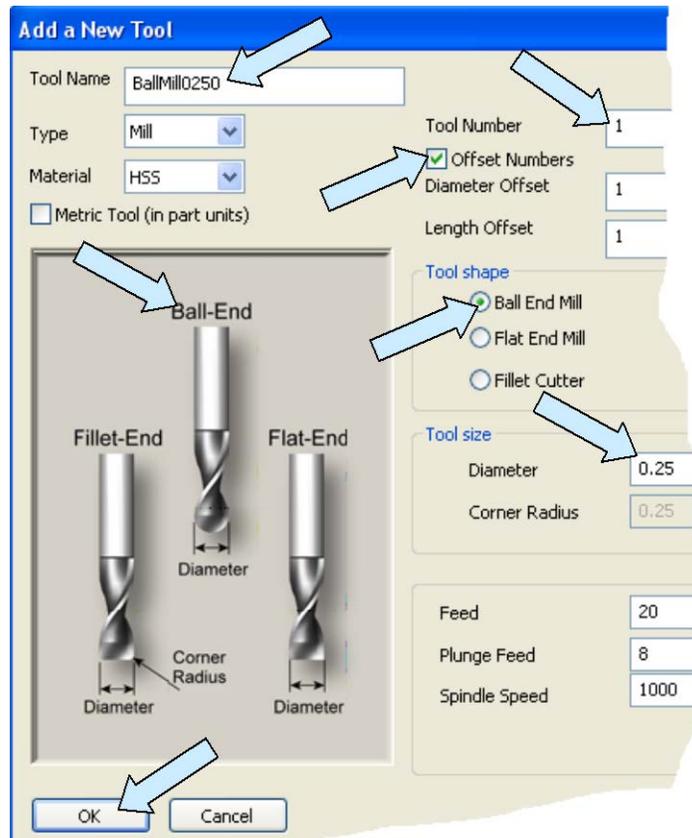
A large Dialog Box appears. Type a name for the Tool that is descriptive.

Enter "1" for the Tool Number.

Click to make the offset Numbers Option Active. This places "1"s in both fields. These numbers appear in the post and can be modified to account for cutter wear or size variation.

Click on the Ball End Mill Option. Type 0.25 for the Diameter. The Feeds and Speed can be changed here or later in the post.

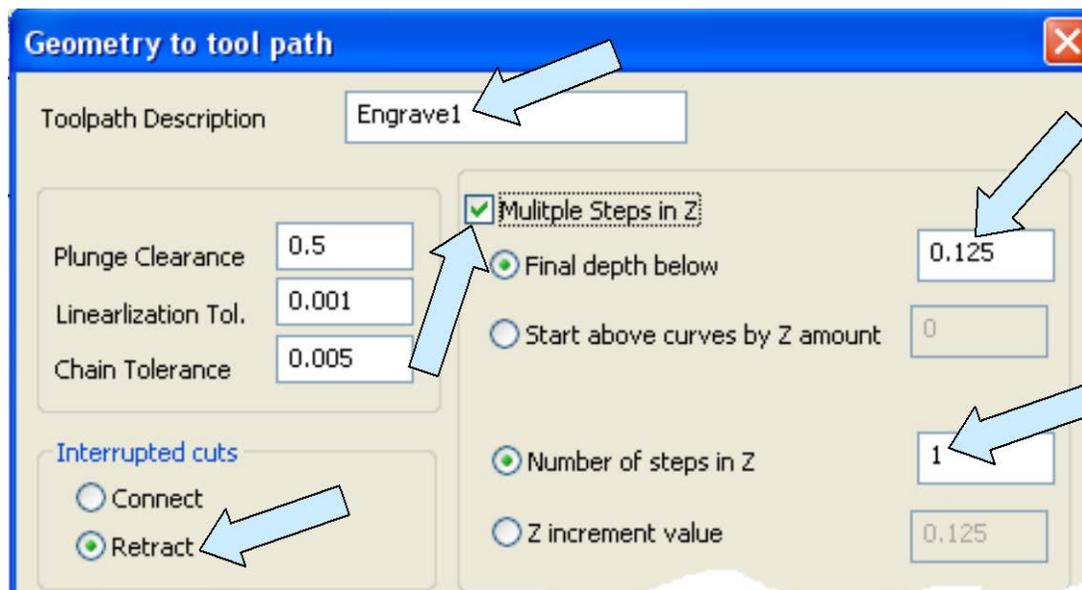
Click on the OK Button.



## KEYCREATOR 3D Direct Modeling Software

You are returned to the main Dialog Box.  
Click on the DONE Button in the bottom, right corner.

The Geometry to Tool Path Dialog Box appears.

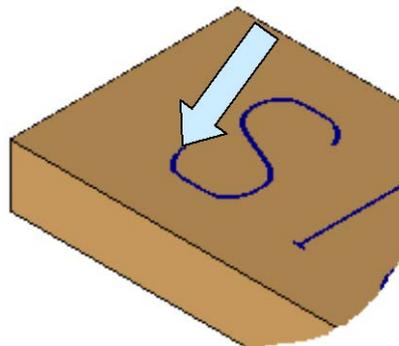


Type “Engrave1” for the tool path Description. Click to place a check on Multiple Steps in Z and select the Final Depth Below Option. Type 0.125 for the Depth. (Since the cutter has a 0.250 diameter, this will create an engraving groove with a depth equal to the radius of the cutter.)

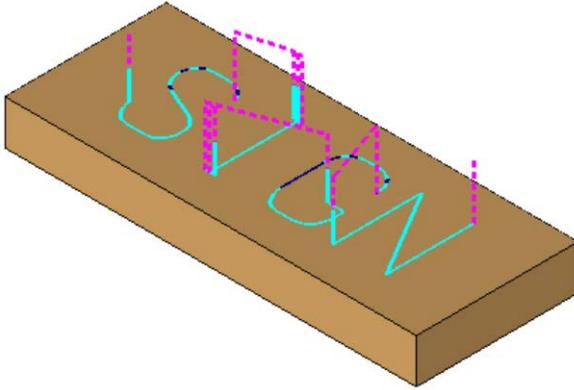
Type 1 for the Number of steps in Z and make sure that the Retract Option is selected for Interrupted Cuts. Click on the OK Button.

You are prompted to select the curves for the tool path. Drag a selection window around the letter geometry only.

Then, using the EndEnt Option, click on the bottom end of the “S.”



# KEYCREATOR 3D Direct Modeling Software



Tool path geometry appears on the screen. The color of the tool path will be the current active color. The lead-ins and lead-outs will be the next color in your color palette after the current active color.

## Verifying Your Tool Path

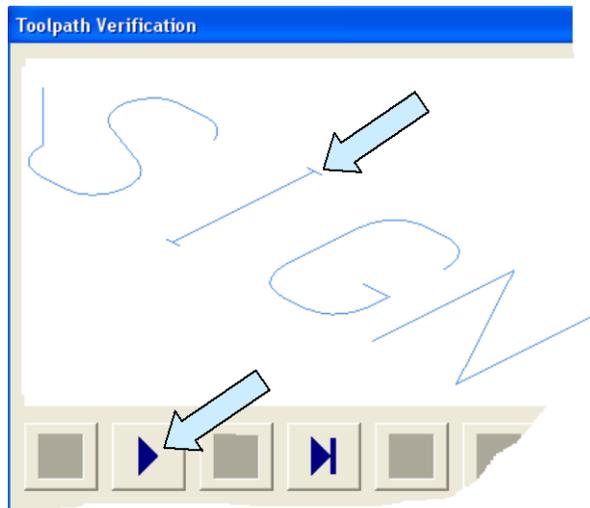
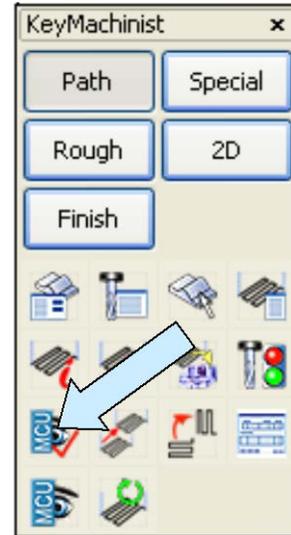
To verify the tool path, click on the VERIFY TOOLPATH Icon.

You are prompted to select the tool path. Move the cursor over the tool path and click on it when it highlights. Hit the ENTER Key.

You are now asked to define the stock.

Click on the Corners Option on the Conversation Bar.

Using the EndEnt Option, click on the top, left, front corner of the block and then on the bottom, right, rear corner of the block. (You are giving the program a bounding volume for the machining operation.)

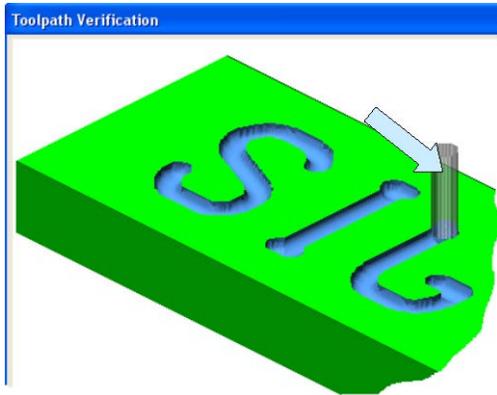


You are now prompted to provide a Part Zero Reference. (Since we located a corner of the original rectangle at the origin (0,0,0), you can just click on the SKIP Button or hit the ENTER Key to accept the default.

You get a message that the MetaCut Utility is starting. Then, a tool path verification window appears with the tool path displayed.

Click on the PLAY Button.

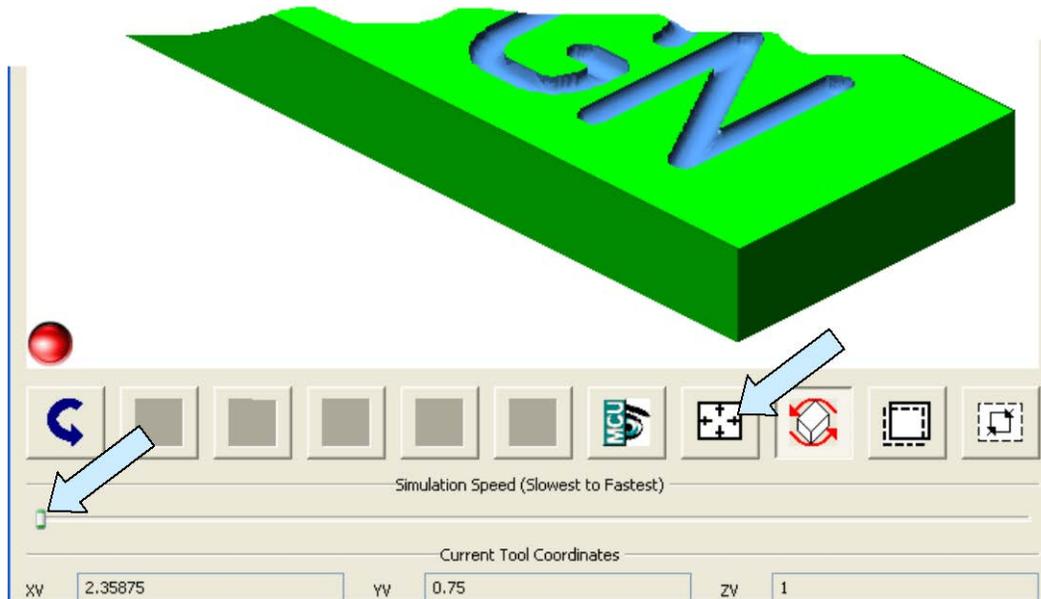
# KEYCREATOR 3D Direct Modeling Software



You will see an animated cutter machining the letters out of a simulated block.

You can replay the simulation if you desire.

If the simulation is not completely displayed in the window, you can click on the AutoScale Button to resize it. You'll also notice a slider at the bottom of the screen. Moving the slider to the right speeds up the simulation.



When you are happy with the tool path that you generated, you can then proceed to create a post and send it to your NC machine.