scmgroup
north america
Applied Technology and Software

Tool Definition: Defining a Flat End Mill

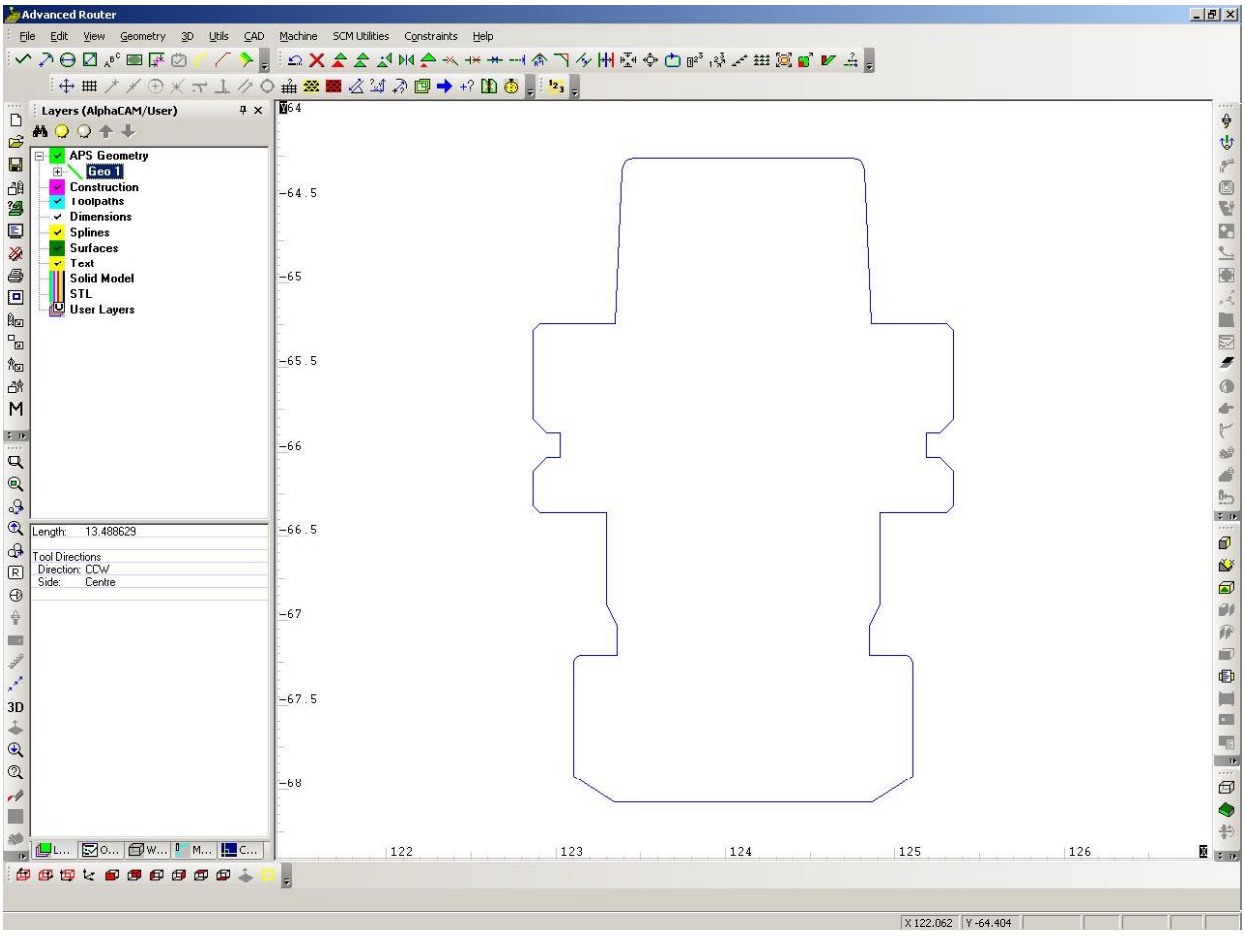
AlphaCAM provides the user with the capability of defining tooling AND the geometry of the tool holder. While defining the tool holder is optional, I highly recommend that the user does so. Having the holder just provides better information to the user (plus it just looks cool).

Having said this, our first step will be to summon the geometry of the holder and INSERT the holder into the main drawing:

STEP 1:

1. File → Insert
2. Find the file named "HSK.ard" located under C:\
3. Drop it anywhere in the drawing. Zoom All.

Your screen looks like this:

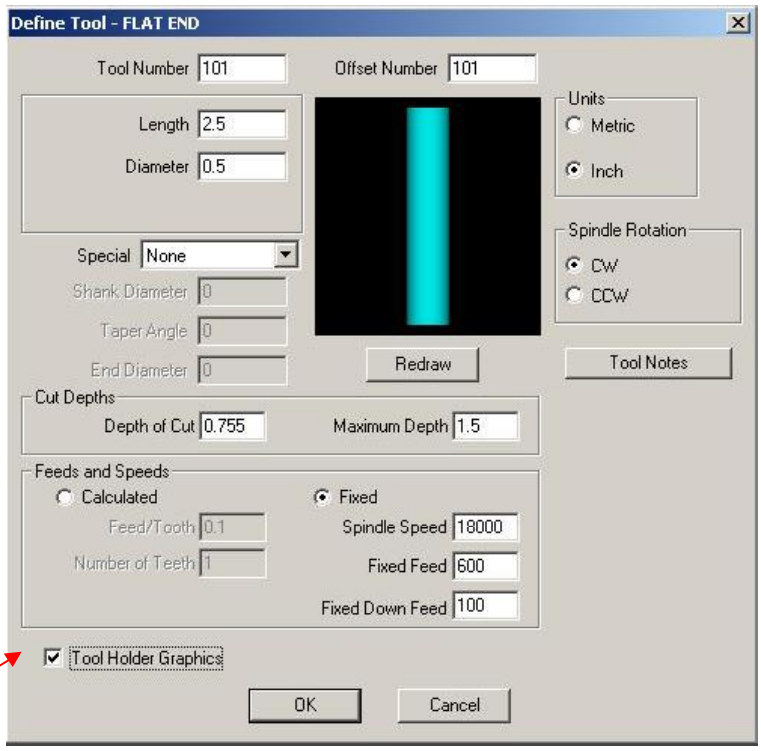


STEP 2:

Next, we will define a Flat End Mill by opening the Define Tool function in AlphaCAM:

1. Go to Machine → Define Tool
2. This will open the following window:

3. Click on "Flat End" and proceed to the next window:



Don't forget to check the option "Tool Holder Graphics"!!!

Full Description of Parameters

Note: The parameters that are shown for this tool apply to a Half Inch cutter, considering that the cutter will work on ¾" Plywood or Particle Board. Different Materials require different feed rates, rpms and depths of cut. You should consult with the tooling manufacturer if you do not have these parameters for your tool.

Tool Number: 101
This is the actual number that will show up in your G-code. This number should reflect the actual position in the Toolchanger where the tool has been placed.

Length: 2.5 (inches)
To AlphaCAM, the length of the cutter is the portion of the tool that protrudes from the COLLET, not the actual usable or maximum depth.

Depth of Cut: 0.755 (inches)
This is the standard depth that we will use with this cutter.

Maximum Depth
This is the Maximum Depth that the user will allow the cutter to work to.

Fixed (Feeds and Speeds)
AlphaCAM allows the user to use "calculated" feed rates. These feed rates depend directly on the type of MATERIAL that the user has selected. In other words, the tool does not have a fixed speed, but a dynamic speed that will change every time that the user selects a different material. Historically the wood and plastics industry has always used FIXED feed rates as opposed to calculated because they are easier to use and understand. A fixed rate is the given speed at which the cutter will attack the piece.

Spindle Speed: 18,000 (rpm)
The spindle speed is defined in Revolutions Per Minute. It is simply how fast the tool spins. Most Router spindles are *optimized* to produce their highest torque at 18,000 rpm, which is why most tooling manufacturers design their tools to work at this rate.

Fixed Feed: 600 (in/min)
This is the Rate at which the cutter will cut the material.

Fixed Down Feed: 300 (in/min)
This is the Rate at which the cutter will *penetrate* the material. In other words, this is the *slower* rate at which the cutter will approach the cut. Some people regard this as the lead-in / lead-out rate. A good rule of thumb is to use ½ of the Fixed Feed.

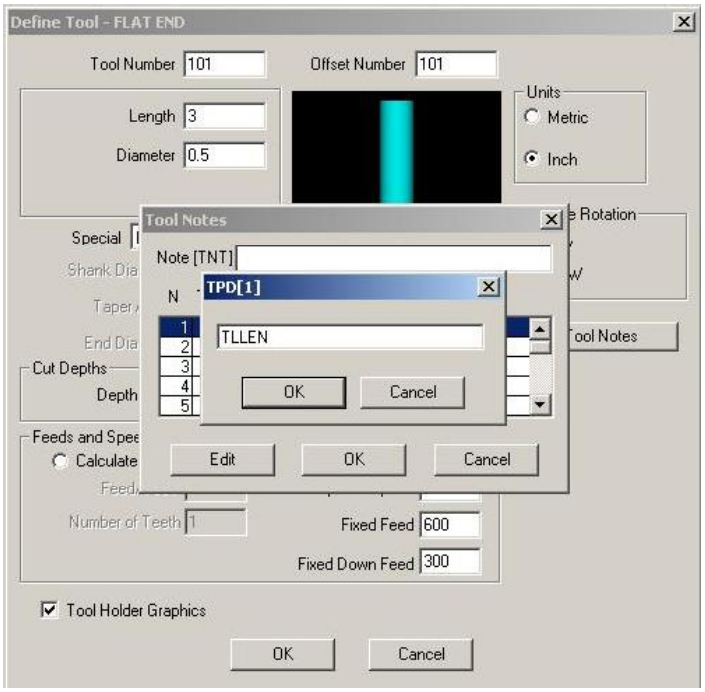
Units: IN
These are the Units to be considered for the tool. Make sure they match your application (inches or millimeters).

Spindle Rotation: CW (right-hand cutter)
Select CW for right-hand cutters or CCW for left-hand cutters.

Tool Holder Graphics: Checked
Optional. Check it to allow AlphaCAM to prompt you for the geometry of the holder.

STEP 3:

You will need to put the Dust Collection Hood Position in Tool Note # 1. To do so, click on the "Tool Notes" button and double-click on the First Tool Note. Write there the special value "TLLEN". The Screen should look like this:



After that, click OK twice to go back to the tool definition screen.

STEP 4:

Once you have filled out all the information, click "OK".

Since you checked the option for tool holder graphics, AlphaCAM will now prompt you for the geometry of the holder. Click anywhere on the holder that is on the screen.

STEP 5:

Save your Tool. You may choose any name for the tool; however I recommend the following notation:

[Type] [Number] [Description] [Units]

Example:
[R] [101] [Half Inch Compression] [IN]

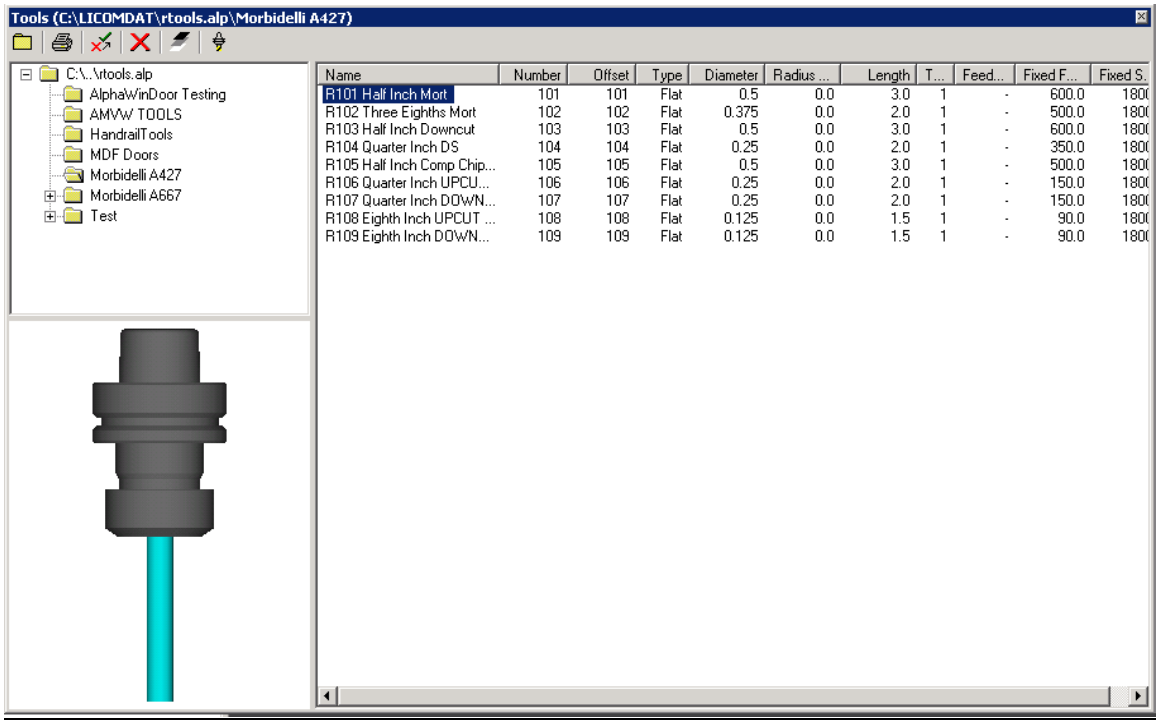
So, the name of the cutter will be:
R101 Half Inch Compression – IN

This way, you will get all the information that you need just by looking at the tool name.

ALPHACAM Tutorial: Tool Selection

TOOL SELECTION

The first step of all machining processes is selecting the tool to be used. This tool will remain active until a new tool is selected. To select the active tool click on the SELECT TOOL icon. The following screen appears:

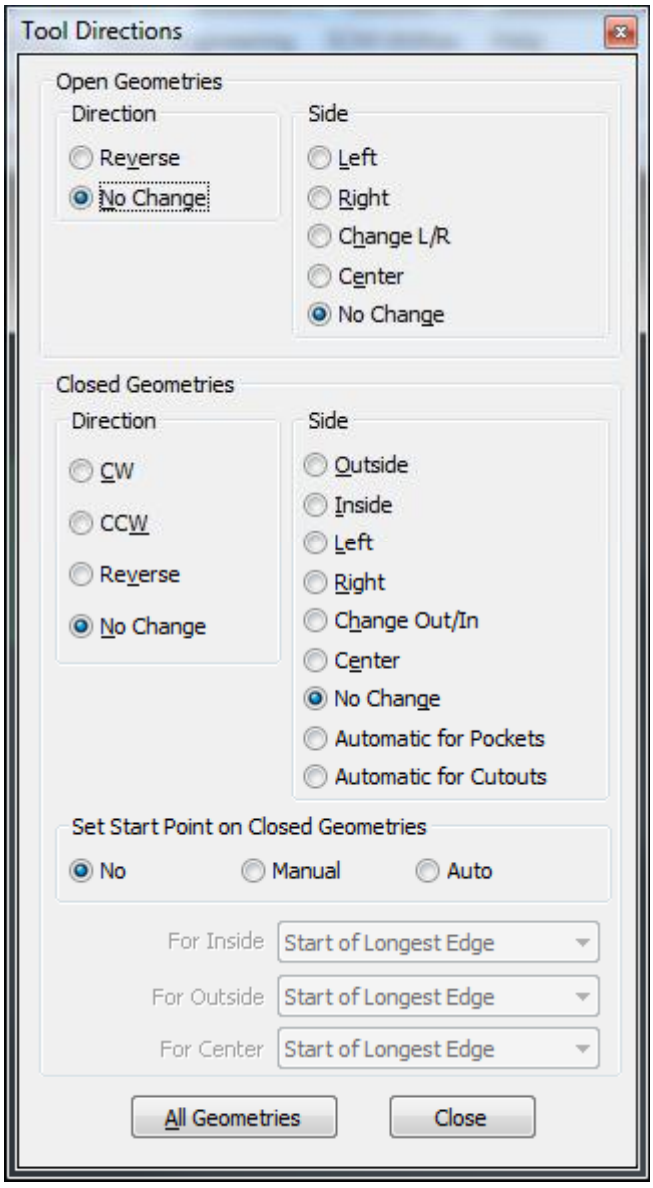


Double click on your tool to select it and proceed to click on Tool directions

ALPHACAM Tutorial: Tool Direction

TOOL DIRECTION

"Tool direction" allows the user to specify which SIDE of the material he wants to cut (inside, outside, right, left, etc…). Make your choices and click on "Selected"



TIP!

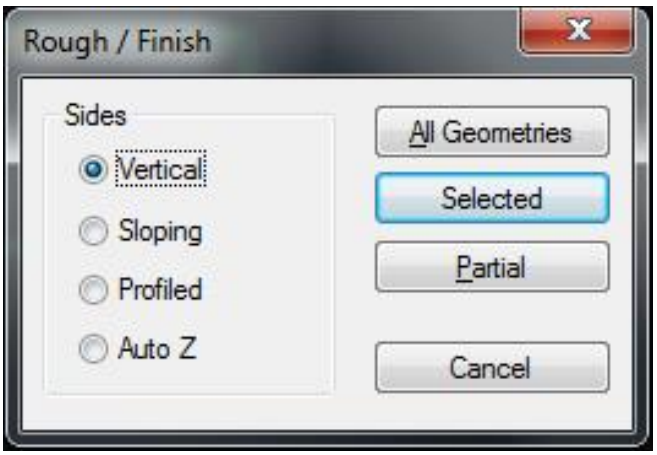
Notice how you can set the START POINT on any CLOSED geometry at the same time that you set you tool directions.

ALPHACAM Tutorial I: Rough or Finish

ROUGH OR FINISH SCREEN No 1.

The first window inside of Rough / Finish allows the user to select which entity or entities will get machined:

These options will allow you to define the TYPE of SIDE of your finished piece. We will discuss only VERTICAL sides in this tutorial.



These buttons allow you to select for machining:

a) All Geometries – Will machine EVERYTHING on the screen.

b) Selected – Will machine those geometries that you will click on later.

c) Partial – Will allow you to machine a specific PORTION of a CLOSED geometry.

Click on "Selected" and move on to the Second Screen:

ROUGH OR FINISH SCREEN No 2.

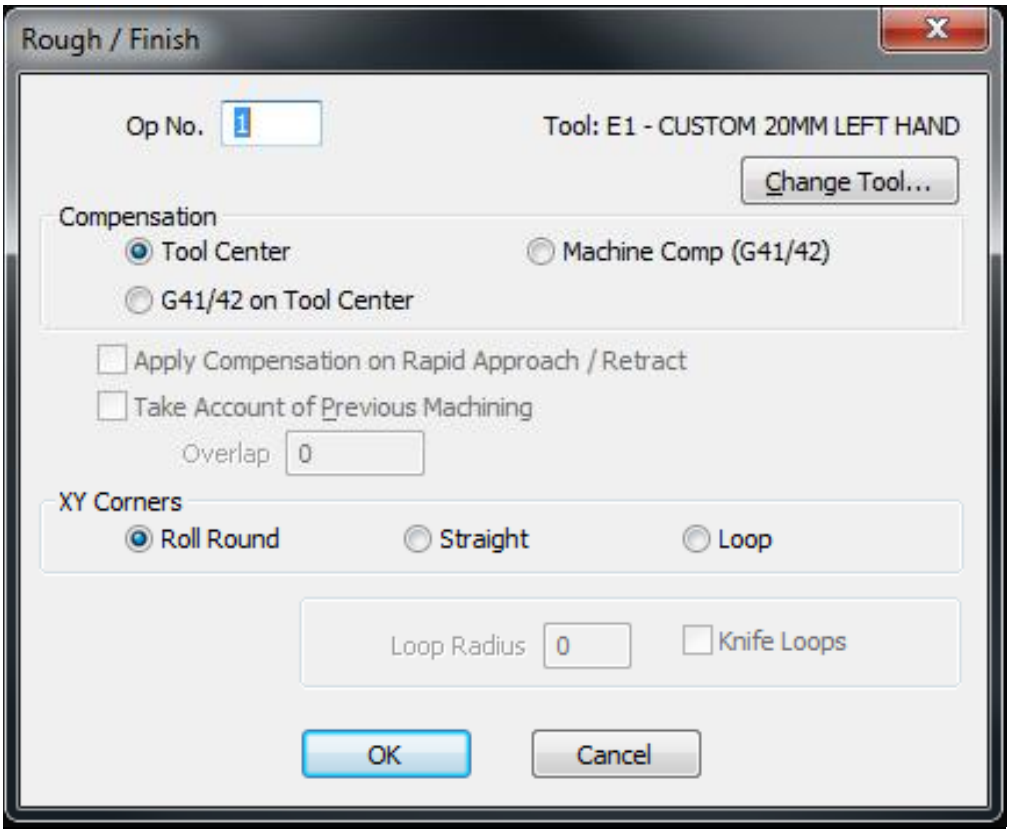
Op No.: This is the sequential number of all total operations that occur on the panel. Usually requires no modification.

Compensation: There three types of compensation available on CNC centers:

1. Machine Comp (G41/G42). This compensation allows the user to output G-Code with the use of an internal compensation command of the CNC controller (i.e. C1 or C2). This option will be used most of the time and it is especially useful if tools are sharpened.
2. APS Tool on Center. This powerful AlphaCAM function will generate the code via **offsetting** the geometry by the radius of the cutter. Works great whenever machining gets complicated (i.e. 3D) but lacks real-time compensation at the machine.
3. G41/G42 Tool on center. Not used on SCM equipment.

NOTE: Make sure to ALWAYS select the box labeled: **"Apply compensation on Rapid Approach / Retract "**

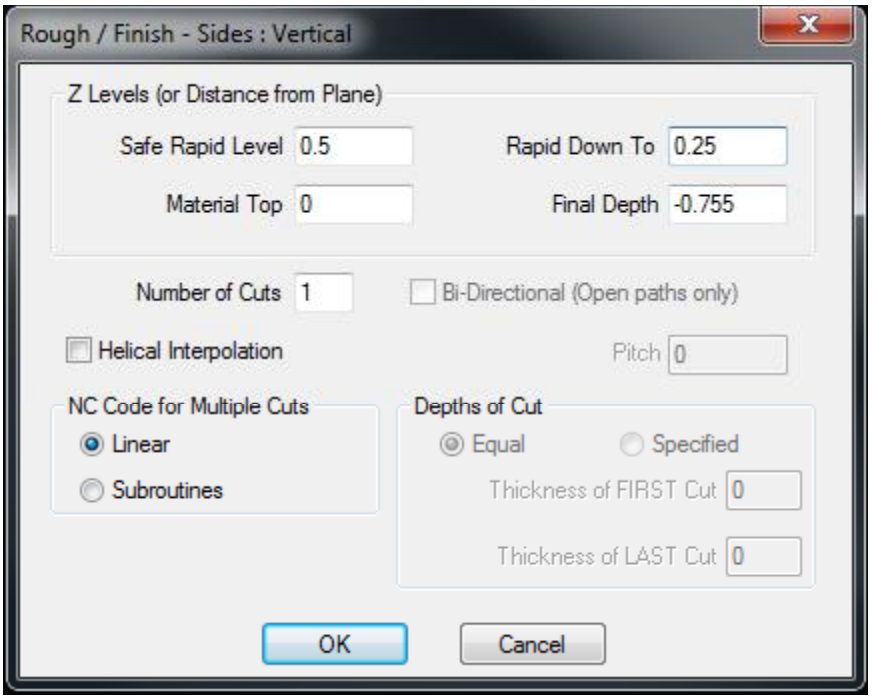
XY Corners: Defines the type of cornering of the panel. Select Straight



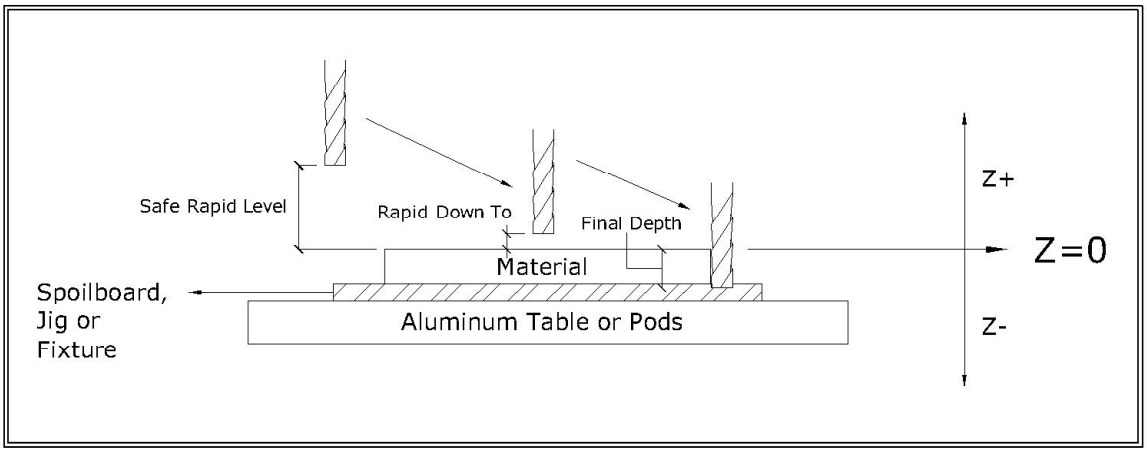
TIP: Notice how you may CHANGE the Tool here!

ROUGH OR FINISH SCREEN No 3.

The third screen pertains to the Z LEVELS that the machine will use. We use these parameters to define things like depth of cut and Safe travel values. The following screen appears:



These values physically represent the following:

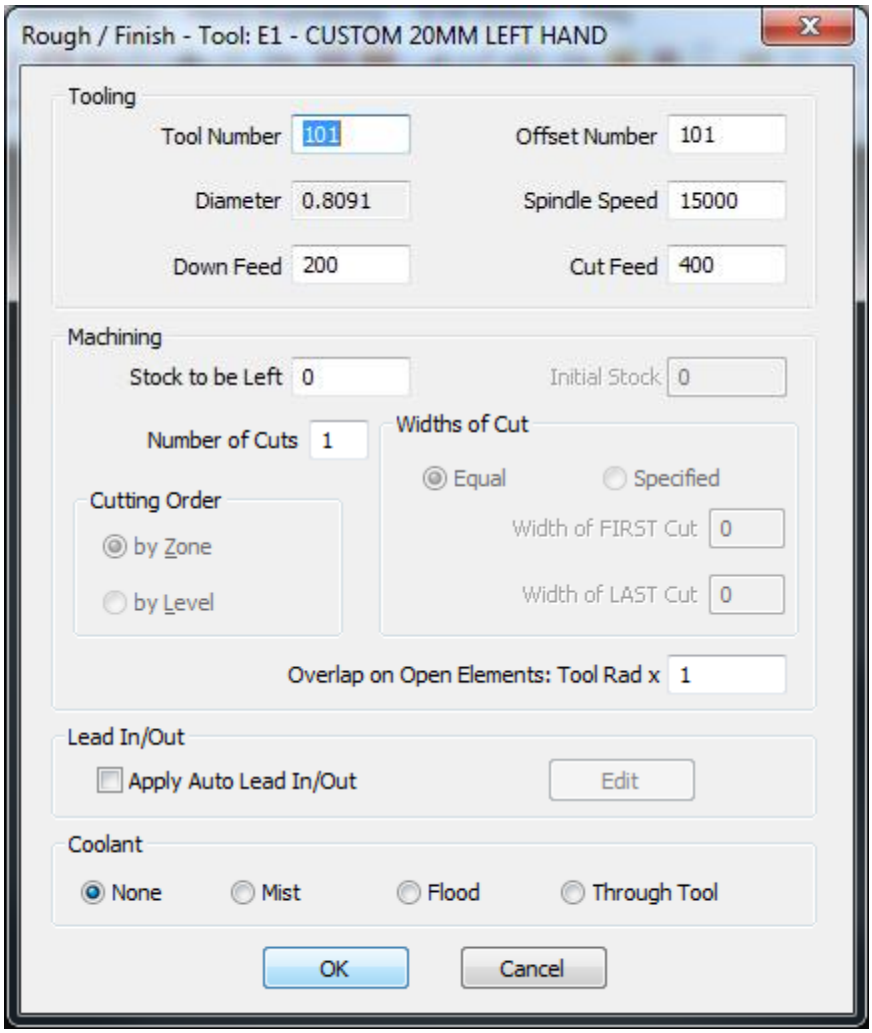


Where:

Safe Rapid Level:	SAFE level at which the tool can travel at FULL speed, clearing Any potential hurdles in its path (i.e. clamps or fixtures).
Rapid Down to:	Distance to begin the approach to the material.
Material Top:	Always Zero.
Final Depth:	Desired depth of cut.

ROUGH OR FINISH SCREEN No 4.

The fourth and last screen of the Rough and Finish screen pertains to the tool itself. If the tooling information was initialized properly, there's usually nothing to change here. The screen looks as follows:



TIP: Notice how, starting in 2011, you can ADD Lead-In and Lead-Out data DIRECTLY to the Rough/Finish Pass!

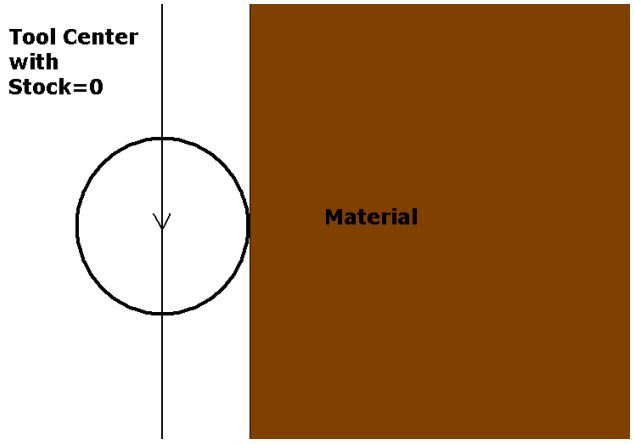
Tool Number:	The number that will be assigned to the T (tool) parameter of the G0 / xG0.
Offset Number:	Represents the tooling offset. Not needed on SCM machines equipped with a Xilog™ controller interface.
Diameter:	Diameter of the tool (not modifiable).
Spindle Speed:	Value in Revolutions Per Minute at which the Spindle will spin.
Down Feed:	Attack speed for the Lead – In.
Cut Feed:	Desired cutting speed.

STOCK TO BE LEFT

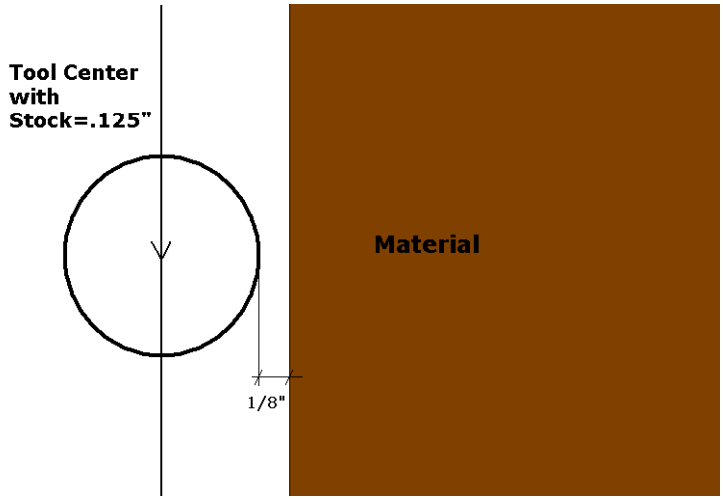
If the desired cut is to be a "finished" cut (i.e. we want the edge of the cutter to follow EXACTLY the geometry that we drew) then stock to be left will be ZERO. On the other hand, if the cut is to be a "roughing" cut (i.e. we are just hogging the material out), then the stock is the value of the OVERSIZE that we want to leave behind.

Effectively, the center of the cutter will move "away" from the material to leave additional material around the edge. It is necessary to use a secondary operation to "finish" the piece.

This parameter is responsible for naming the operation either Rough OR Finish, depending of whether or not stock exists. See example below
Example of cut with Stock=0 (FINISHED cut)



Example of cut with Stock=.125" (ROUGHING cut)

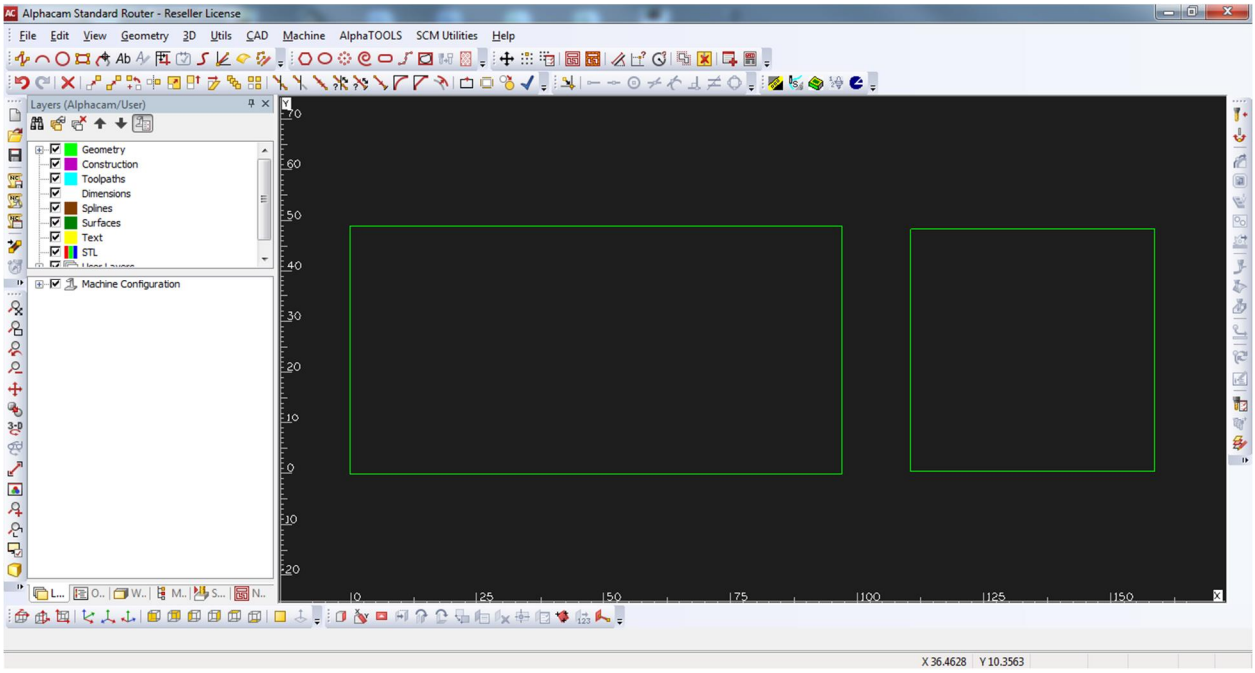


AlphaCAM Tutorial: Basic Nesting

Nesting is an automatic function available in all AlphaCAM modules Standard with Advanced Nesting and above. In simplest terms, nesting is a method for arranging parts you want to cut on the material you have. Nesting is not specific to panel processing; it's just as capable of laying out solid wood pieces on rough or prepared stock. The following is the process in AlphaCAM for basic nesting including outputting NC code.

Material

Open a new file in AlphaCAM and draw a rectangle with the dimensions of your material. For example, if you have a 6" x 86" piece of solid stock or a 97" x 49" sheet of plywood; draw a rectangle of the specific size of material you have. You'll notice in the image below there are two rectangles. One is a 4' x 8' sheet; the other represents a cutoff (leftover from another panel processing job). Regardless of how many sizes of material you have you only need to draw one rectangle for each size of material regardless of how many sheets it will take to nest the quantities of parts you have.

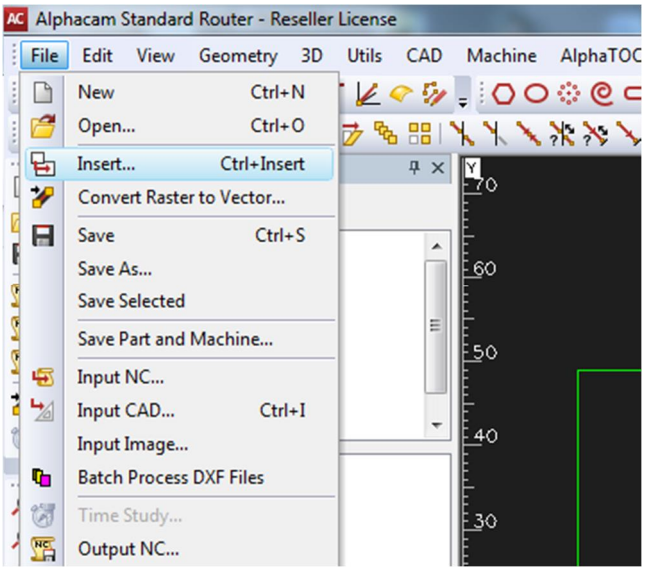


scmgroup
north america
Applied Technology and Software

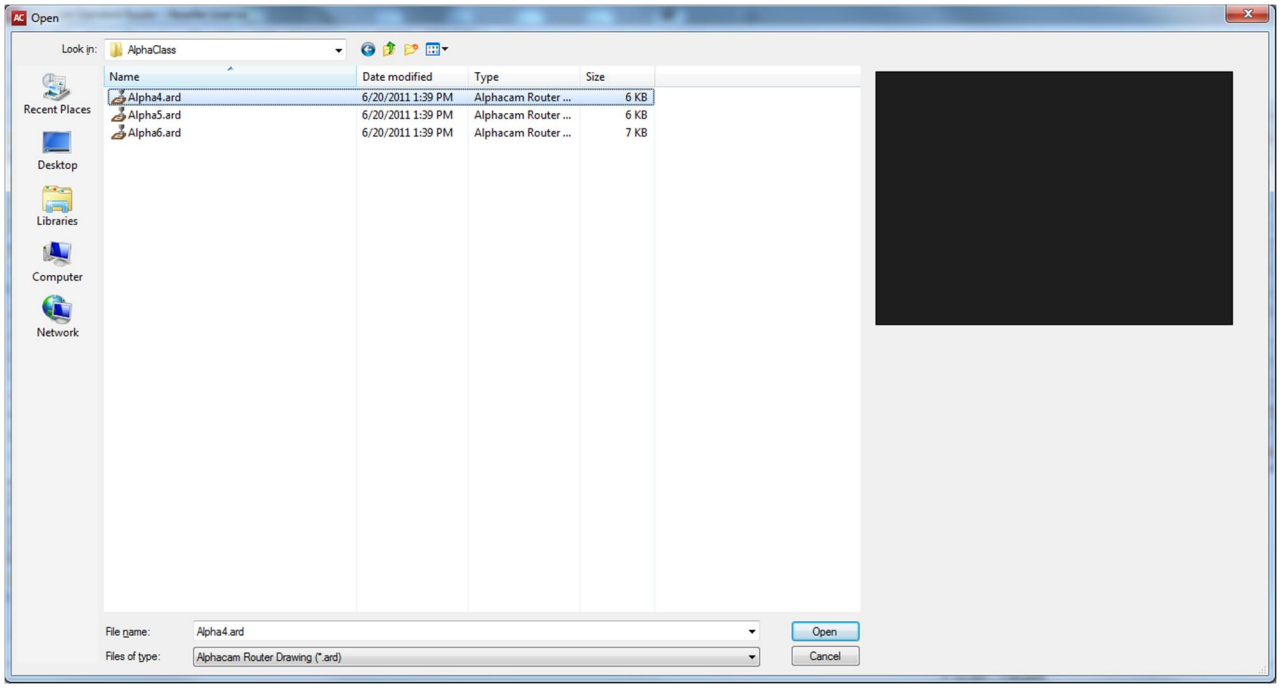
Insert Files

The next Step is to insert the files or the parts you want to nest. The following images describe what you should see as you go through the process of inserting your files.

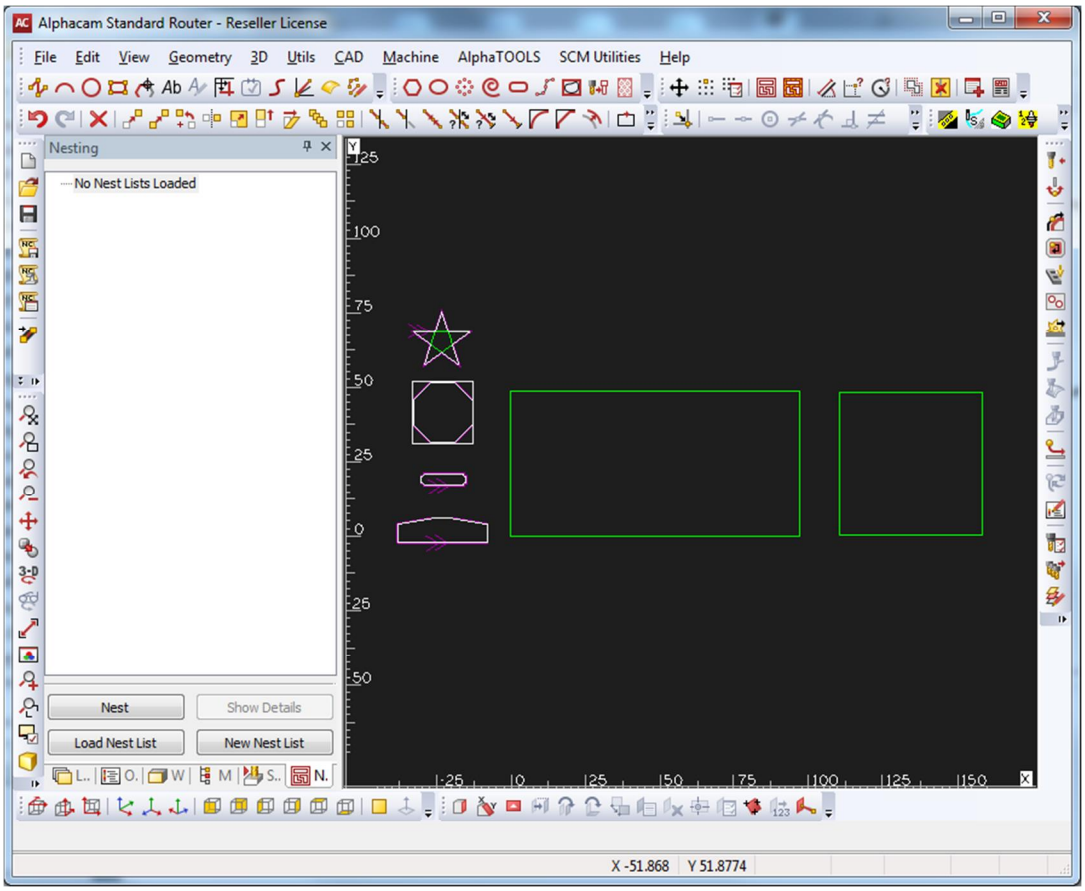
- File → Insert



- Select the files you want to insert, ONE AT A TIME, and click Open.



- Drop each part on the screen, near the rectangle(s) you've draw to represent the material.



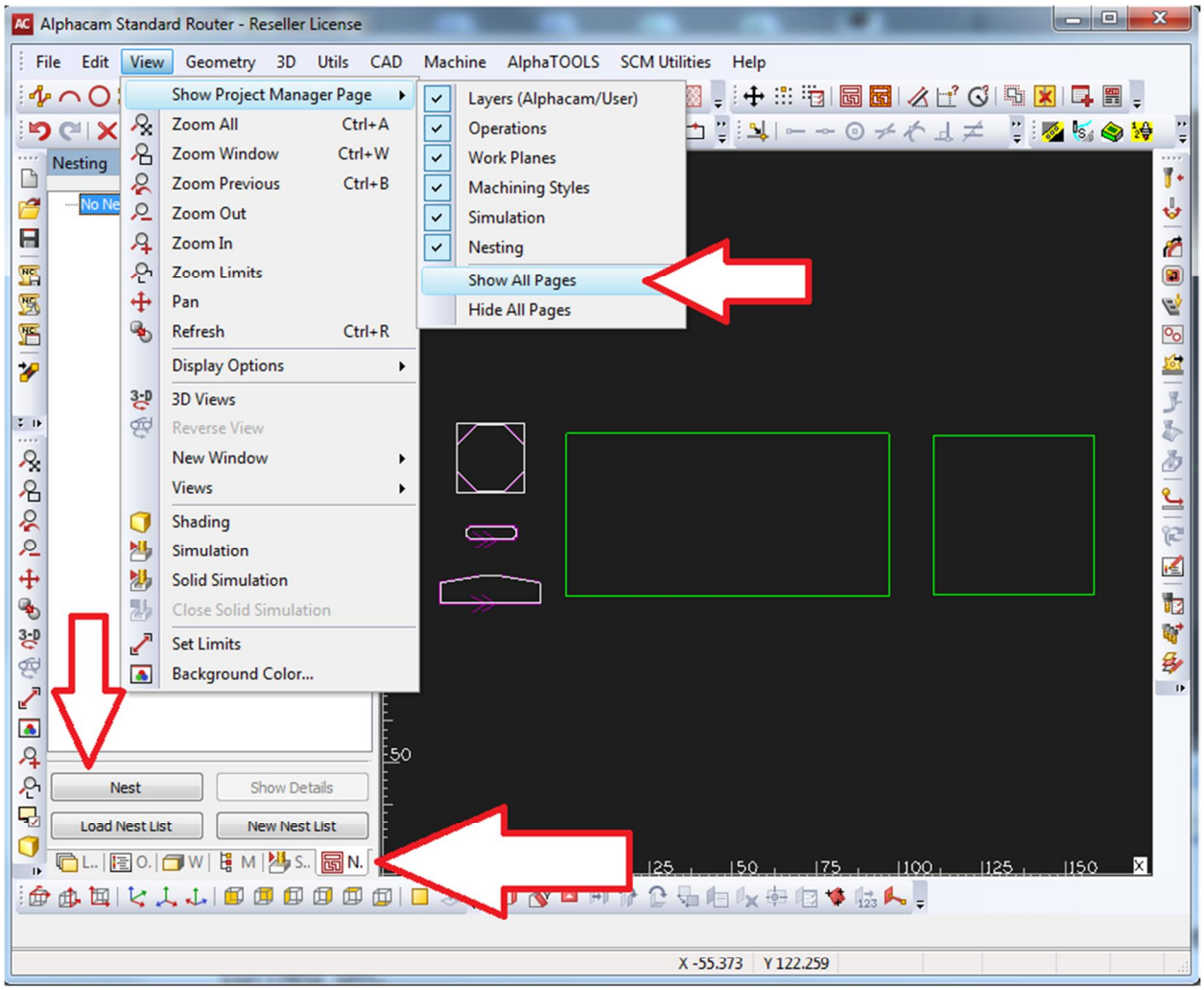
Notice in the image above that the enclosing material is around the stop sign, but not on the other shapes. The other shapes were prepared without enclosing material or the enclosing material was deleted form the files before being inserted into the nesting file we've prepared.

Delete the enclosing material on all parts inserted into the drawing. We do this because later in the process our rectangles we drew to represent our material will be define as the enclosing material for nesting and subsequent NC output.

Nest

To perform the Actual Nest, go to:

View → Show Project Manager Page → Show all Pages.



This will ensure you have your project manager page with the nesting tab turned on. The tab represents a very useful set of tools when nesting.

Click on the Nest Tab on the project manager page, then click the Nest button (see red arrows on above image).

When you click on the Nest button in the nesting tab on your project manager page, a window will appear (see below) with 3 choices for Nesting: True Shape Nesting, Rectangular Nesting, and Manual Nest. This window will also allow you to specify what parts you want to nest and

what features of the parts you want to nest: geometries, tool paths, or geometries and tool paths.

True shape nesting

TSN will draw a virtual boundary around your part by means of an offset that mimics the shape of your part. If you had squares or parts with a rectangular outer shape, like cabinet sides, then true shape nesting wouldn't be necessary, although it would work fine. True shape nesting is best used for irregular shaped parts, such as the star in this example. Since I'm nesting the star along with the other parts this will be the option I choose.

Rectangular Nesting

Rectangular Nesting, like the name implies, is best used for rectangular shapes. In the example in this tutorial, if I did not have the star, I might have considered this for the first two shapes because they are mostly rectangular.

Manual Nesting

Manual Nesting allows you to place parts on the material given some basic guidelines such as spacing between the parts and spaces of the parts from the material edge. This will be demonstrated later.

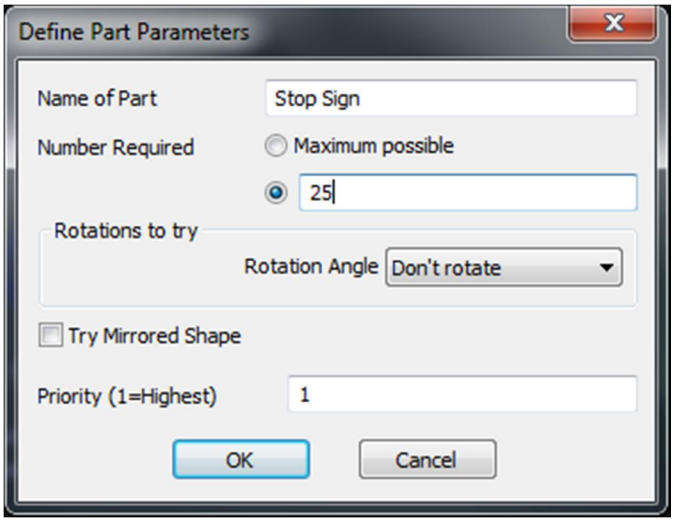
Select Parts From Screen: This option will require you to select the parts from the screen and specify the properties for nesting each shape.
Use nest list: can only be chosen if you've created a nest list prior to this. The nest list we create here cannot be saved as a nest list. This is a separate procedure not outlined in this tutorial.
Tool paths: choose this option only if you want to nest just the tool paths.
Geometries: choose this option if you only want to nest the geometries. This option will require you to set up tool paths after nest if you want to output NC code.
Toolpaths and Enclosed Geometries: this is the option we will choose and will nest the parts along with their individual tool paths.

If you're following along in AlphaCAM, make your selections on the nesting window match the previous image and click ok.

You will see that your prompt line is asking you to select the parts you nest. **IMPORTANT –** draw a bounding box around our parts, do not select the part, this may lead to one or more geometries or toolpaths being left **unselected**.

To begin Nesting:

1. Select the Part to Nest (Stop Sign). You will see the following window after you select the shape:



- Name the shape.

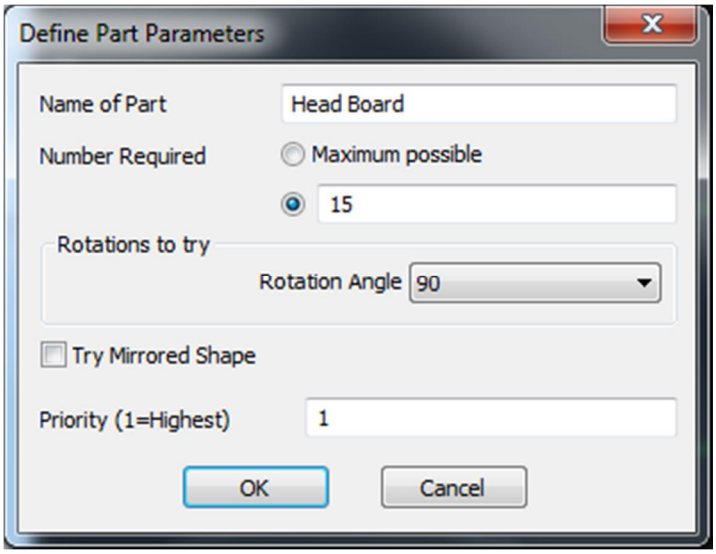
- Select the button next to the part quantity window; fill in a quantity.

- Rotations to try: in this case I did not rotate because my part is basically a square, and rotations may not provide any different results in the nest than not rotating at all. Of course, you would choose don't rotate as well if you are working with a certain grain direction and the way you drew it is the way you want it to be nested.

- Try Mirrored Shape: Only used when you have grain-less material and a part which is symmetrical on both sides. This will allow AlphaCAM to try to mirror the part to try to get a better overall yield. This option is not an automatic "nest a mirrored version of my part" button. If you need a mirrored part (as in cabinet sides), draw a mirrored version and nest it along with all the other components. Priority: I'm setting this at 1 for this shape. All 1's are done first, 2's second, 3's third … you get the picture.

- Limit Number on each sheet: this can be used to help maintain your vacuum when there are many, many small parts in the mix. Every time you cut a part, we open up a small channel (the tool diameter) which causes the vacuum to leak. By controlling the number of small parts with the Limiting Number, we can have some influence on the leaking vacuum.

- Click OK and move on to the Next Part.



The settings for the new part are going to be a little different; we'll change our quantities, rotations and priorities.

- Name your part and select a quantity.

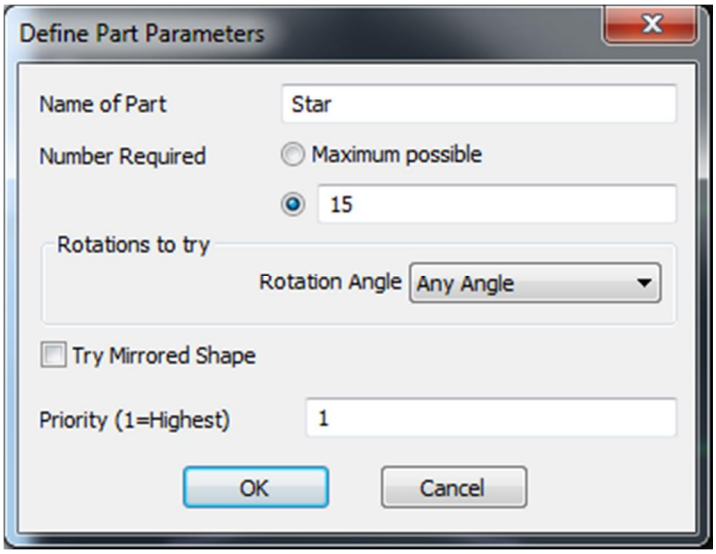
- Select "rotations to try" and select 90. As AlphaCAM goes through the nesting process, it will rotate the part by intervals of 90 degrees to try and fit the part into available openings.

- Try rotated part first: this button will do just as it says. If you selected 90 degrees as your rotation angle, AlphaCAM will immediately rotate it 90 degrees at the start of the process. I may use this if I don't care about grain orientation and as I was inserting files in the earlier steps of this tutorial I see that the rotate part will fit better in the layout.

- Priority: Same as before.

- Click OK.

- Select your next part.

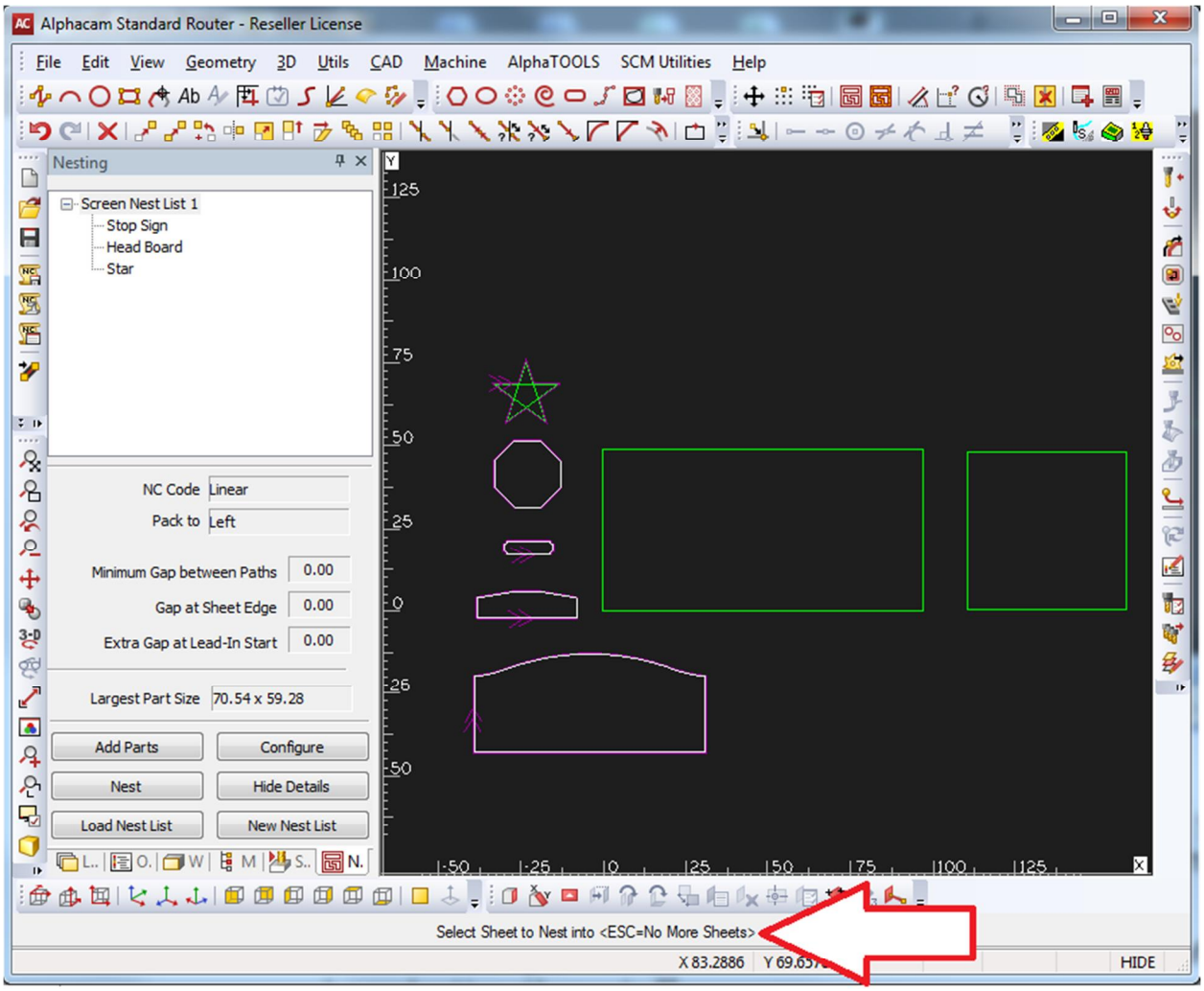


- In this case I changed the rotations to “Any Angle”. Please note that when you set it to this option, AlphaCAM will try its hardest to come up with a solution for the nesting. This might take a while on a slow computer. Try to use this only when you know you will benefit from the extra effort on the Nesting side.

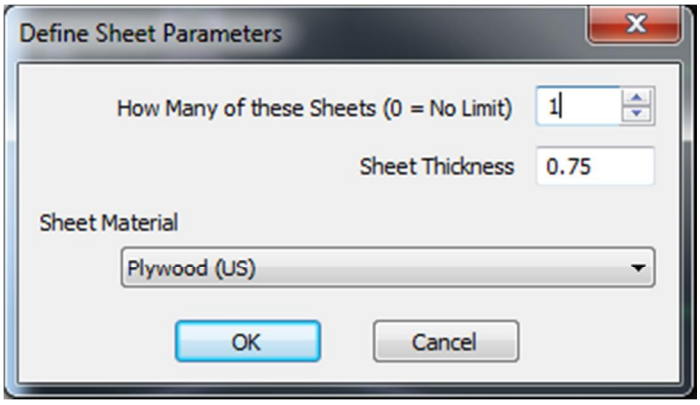
- When you have selected all your parts and completed the part parameters screen for each one of them, your screen should look like the image below. Read your prompt line.

- Click ESC on your keyboard.

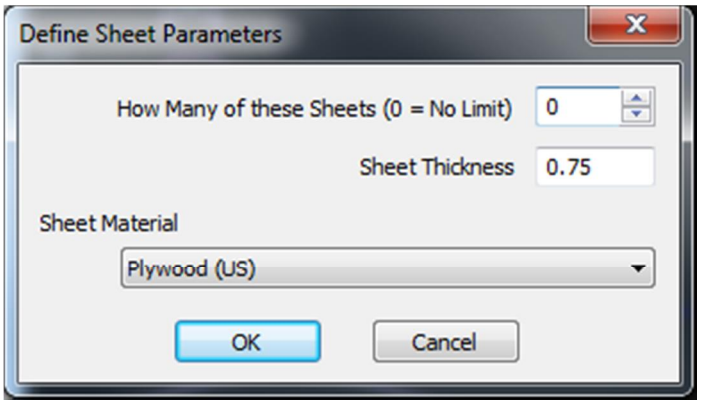
- After you click ESC on your keyboard (or right click on your mouse), your shapes will reappear on your screen and the prompt line will ask you to select material.
- When nesting panel products with full sheets and cut offs, always start by selecting the cutoffs.



- When you select the cutoff, the Define Sheet Parameters window will appear. This same window will appear whether you are nesting with flat panel sheets or dimensional lumber.



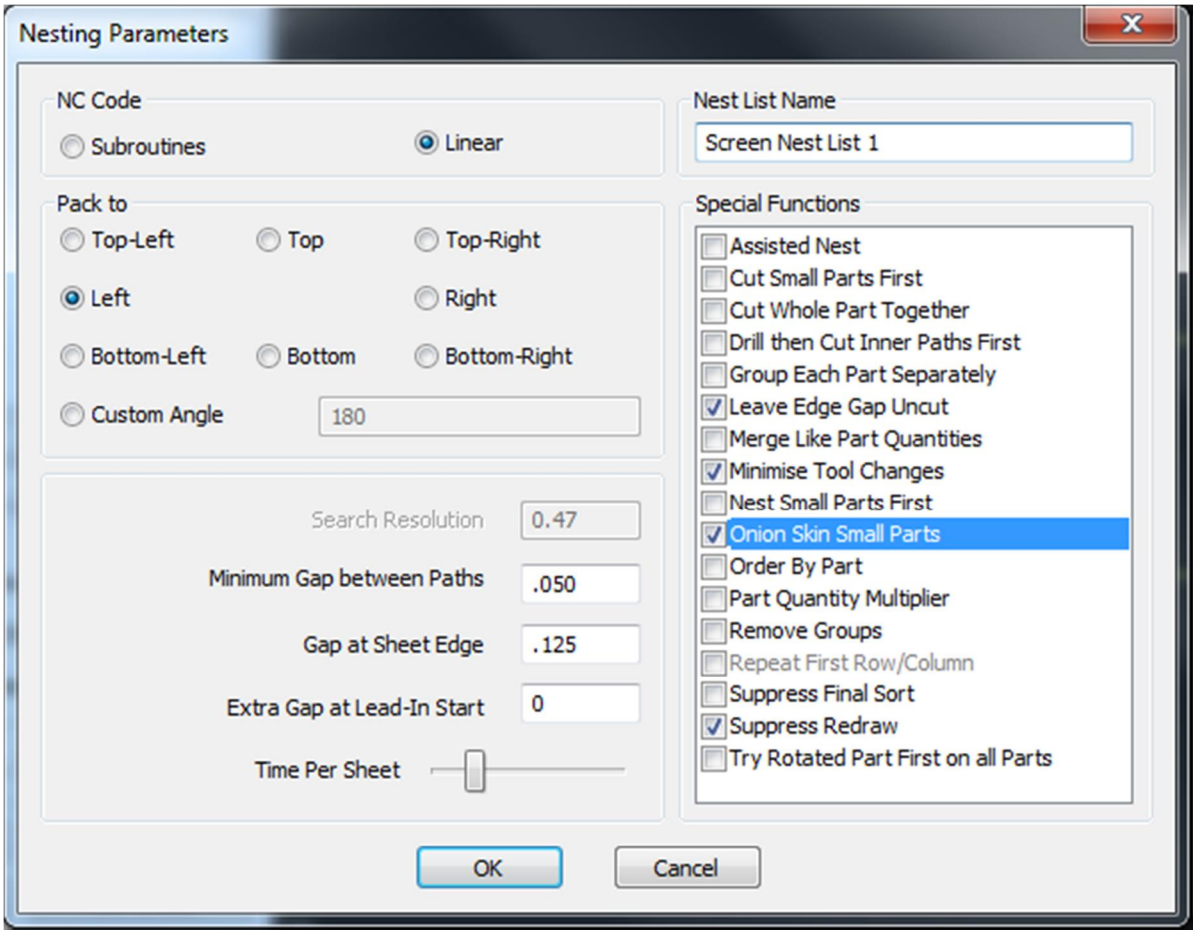
- When selecting the cutoffs, fill in the quantity with exactly the number you have – usually this is 1, unless you had a previous job that created all the same size cutoffs. However, this number, for cutoffs, is never 0 (which if you'll notice "0= no limit". 0 is only used if you want AlphaCAM to tell you how many sheets you need to nest the quantities you've required in your part parameters. We'll use 0 for our full sheets.
- Sheet thickness is a positive number of the thickness of your material. Much like setting up enclosing material, it's a good idea to take a sampling or measure exactly the thickness of your material.
- Sheet material is used only for reporting purposes. This will be the Material Name reports in the Nesting Reports and Part Labels, if you choose to output them. Also, choosing the "wrong" material will not affect your overall nesting.
- Click Ok and select the full sheet.



The only difference with the full sheet will be the "How many sheets" section. Set this at 0, which will prompt AlphaCAM to use as many of this shape/sheet it takes to give you the quantities your require.

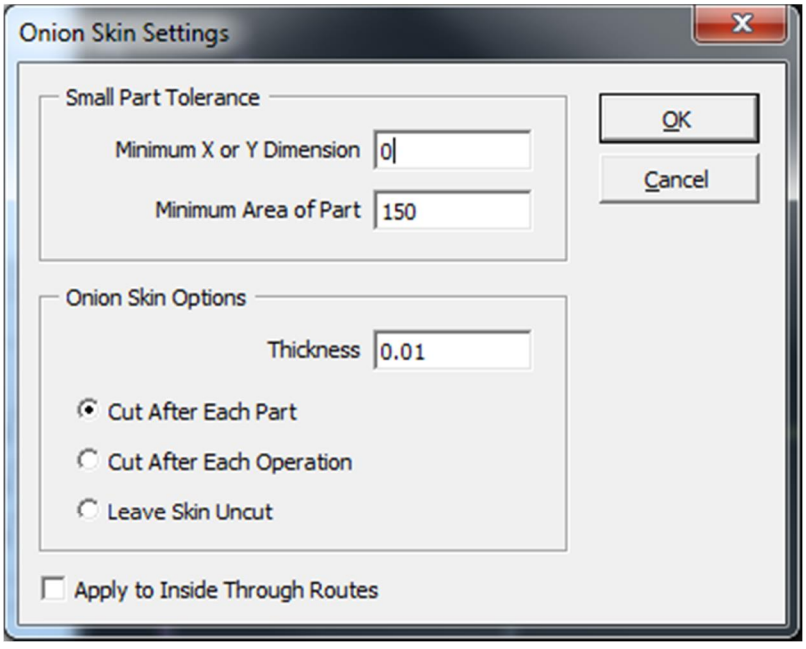
- Click OK. You'll notice the prompt line is still looking for another sheet, or for you to say no more sheets. We have no more sheets so we'll click ESC key on our key board or right click on the mouse.

In doing so, the following window appears, Nesting Parameters:



- Pack To: this section has more to do with your shop layout than anything else (for table machines). Ask yourself where am I going to unload the machine, then select the corresponding location. Another consideration is if you're using a rake. Rakes don't like to push against uneven surfaces. Since rakes move from left to right when cleaning the table, it's wise to select Pack to Left when using a rake.
- Search Resolution: This is a default based on your files you've set up to nest. If you don't like the outcome of the default, reduce this number. Be careful how much you reduce it. The smaller you make it, the harder AlphaCAM will work and the longer it will take. Adjusting parameters like your priorities may yield better results than forcing this number to .01.

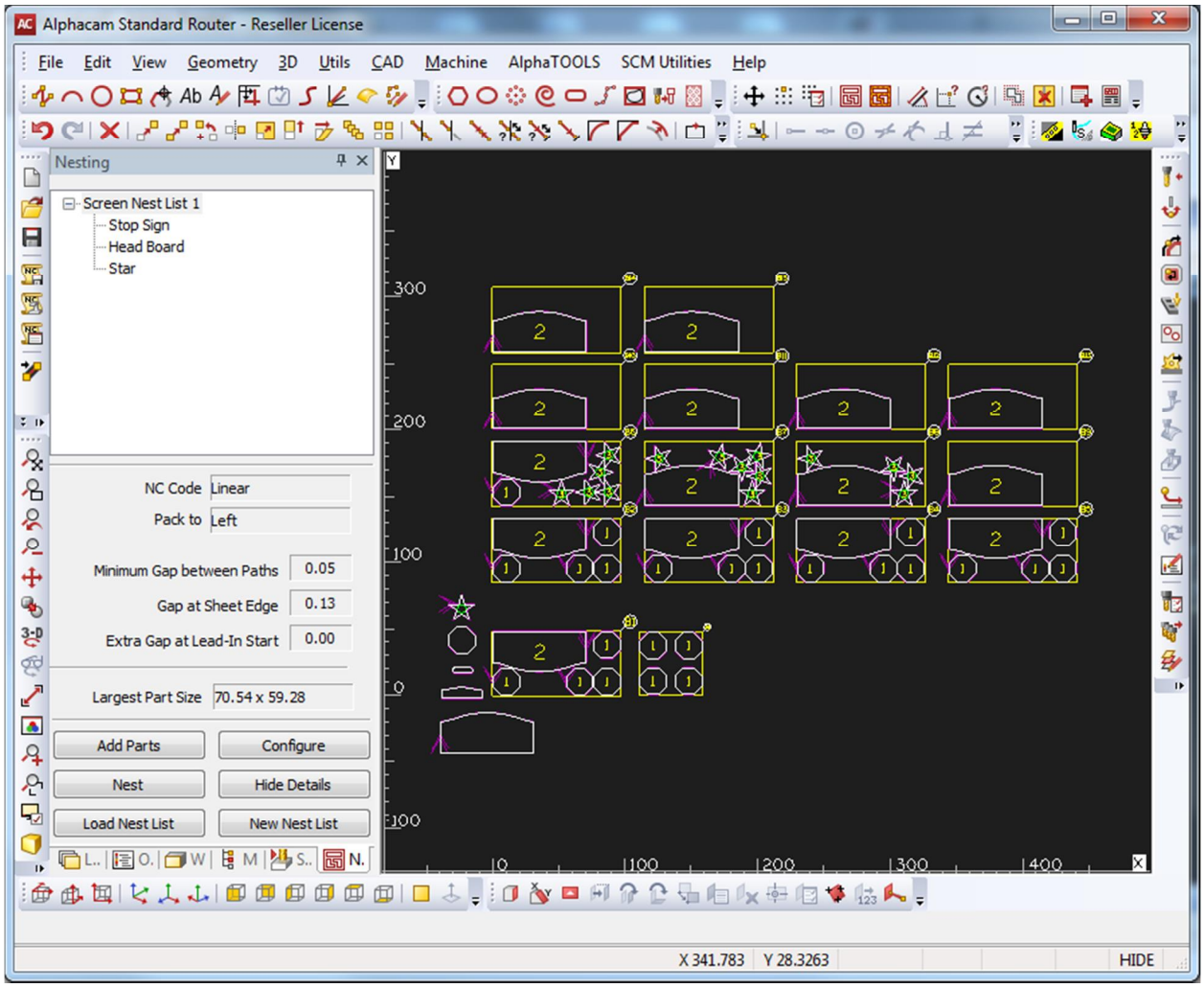
- Minimum Gap between PATHS: theoretically the minimum gap between your PARTS is the width of the tool. But then you're cutting a conventional cut of one part and a climb cut on the one next to it. Having said that, this is asking you how far apart you want to place your tool PATHS since they are taken into account when nesting. A small amount like 0.050" or 1.5 MM is plenty to assure your cut directions are maintained.
- Gap at Sheet Edge: How rough is the edge of your material? Usually, plywood may have dings and dents about 1/8" to ¼" around the edge of the sheet. For prepared stock, this would be zero. For rough stock, this number should be how far you have to offset your parts in from the edge of the material to overcome the rough edge.
- Extra Gap at Lead-In Start: I always make sure that my Lead-In is only a couple of degrees, thus minimizing the impact of it during nesting. Because of that, I usually have a value of 0 here. If you require more distance in your Lead-In put it here.
- Leave Edge Gap Uncut: AlphaCAM will always try to nest your shapes INSIDE the sheet. If you do not UNCHECK, you will find that your distance from the edge of your raw material to the first finished edge is the tool diameter + Gap at Sheet Edge. Unchecking this option makes the Gap at Sheet Edge Absolute.
- For Info on the rest of the special functions, there could be a tutorial on just that, but it's best to read up on the help section about what each one does. Most are pretty self-explanatory. I do what to highlight the Nest Small Parts and Onion Skin options as these can be common.
- Check the boxes shown above and click ok. The following window will appear.



- Since you selected nest small parts and onion skin, AlphaCAM needs for you to tell it what you consider a small part.

- Small Part Tolerance: this requires that you either give a value for the smallest x or y dimensions to define a small part or specify a minimum area for the part (length x width).
- Onion Skin Options: requires a thickness for the amount of material you want left (the onion skin) which will hold all the parts together with the scrap. You can then choose to set when you want the skin cut, or if you want it cut.
- Click ok.

Your nest will be calculated and shown. This is the result of the nest list we created here.



You have a few options here:

1. Undo and remove head boards from your nest list (you can accomplish this by right clicking on the head board in the project manager page and clicking configure; the nest list part parameters will show allowing you to change the quantity).

2. Add a line and tool path to cutoff the excess.

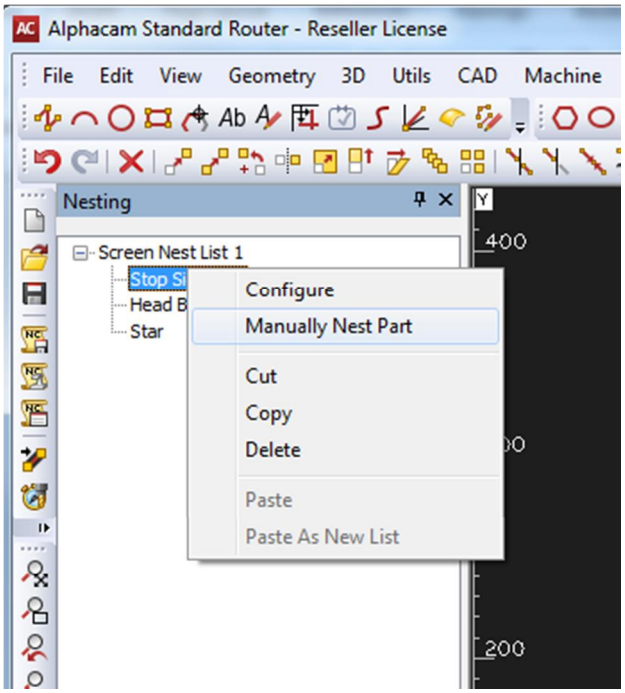
3 Add additional parts using manual nest.

Manual Nest

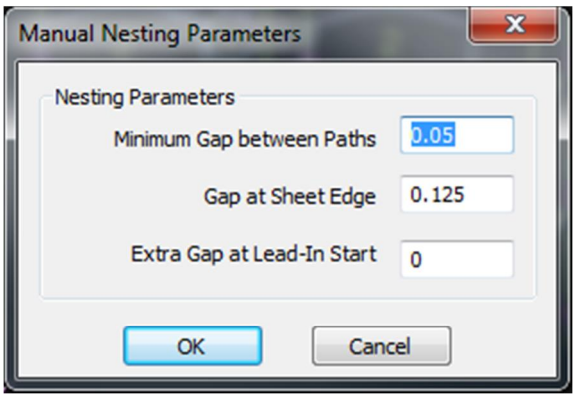
Manual Nesting allows the user to create a Nest by placing the parts on the screen by hand. This is very useful when trying to maintain grain continuity, for example, in a set of drawer fronts that must be grain matched.

To add parts to the manual nest:

- Go back to the Nesting Tab and r-ght click on any of the parts you would like to nest manually. Like this:



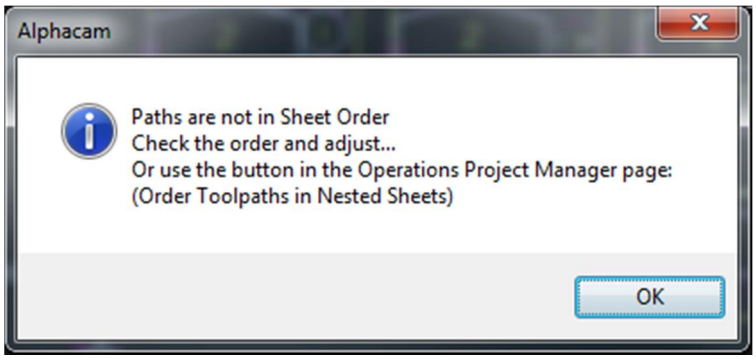
- Select “Manually Nest the Part”. The following screen will appear:



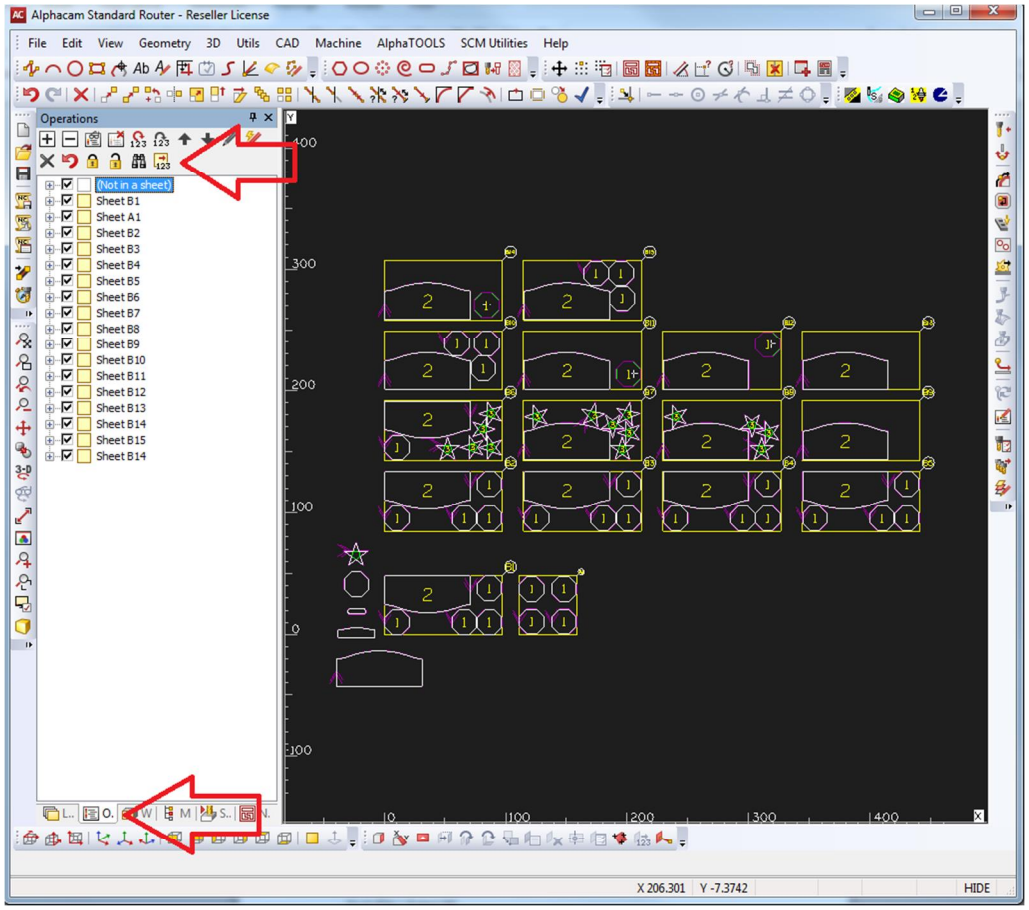
Notice how these parameters have the same names as the ones in the TSN Window.

When you click OK, you will be able to "drop" that part on any sheet. If you added ONION skin to the Nesting, then you will be prompted again for the Onion Skin Parameters. Following the instructions on your prompt line, place the part in any sheet! Press F2 to rotate the part.

TIP: If you ever get the following message:

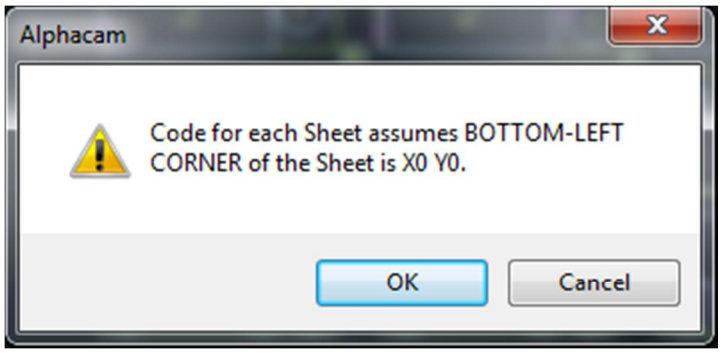


Go to the OPERATIONS Tab and hit the button that re-orders the toolpaths:



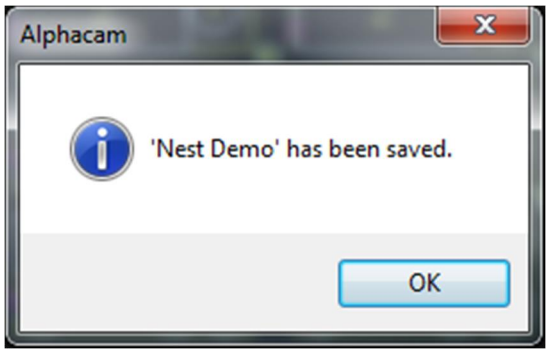
- Press ESC on your keyboard or the right mouse key.
- Go to File → Output NC.

The Following window appears:

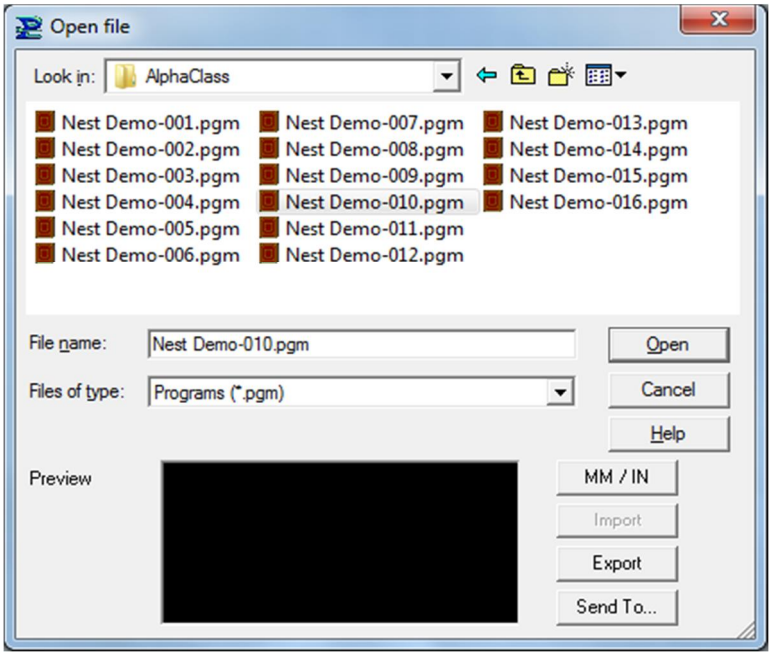


This is telling you that the bottom left corner for each of the individual sheets will be set at x0, y0 … just as if you were outputting NC for each individual sheet in separate files.

If Output NC was successful, you'll see this window:



Open Xilog Plus→ File →Open



Notice the individual files Created form the "Nest Demo program". One program for every sheet.

Congrats! You have successfully Nested your parts!