scmgroup
north america

Applied Technology and Software

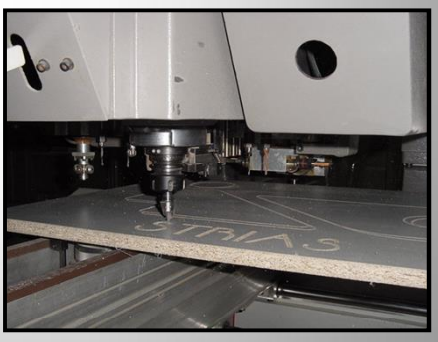
Basic Machining Center Training Course

CAD CAM

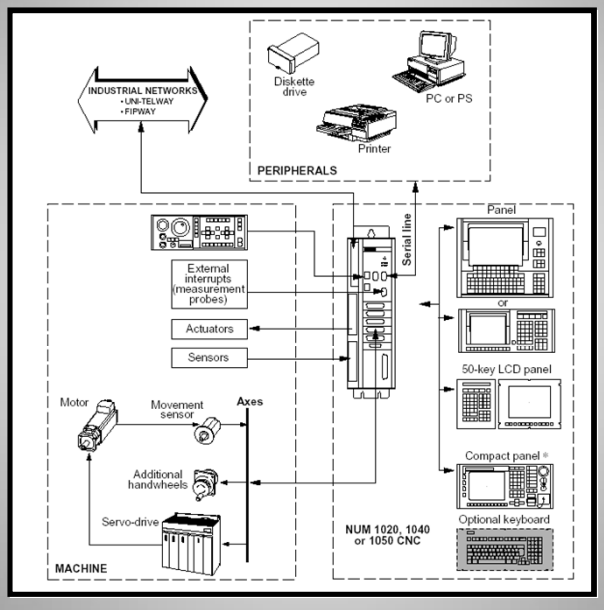
TABLE OF CONTENTS

What is CNC?

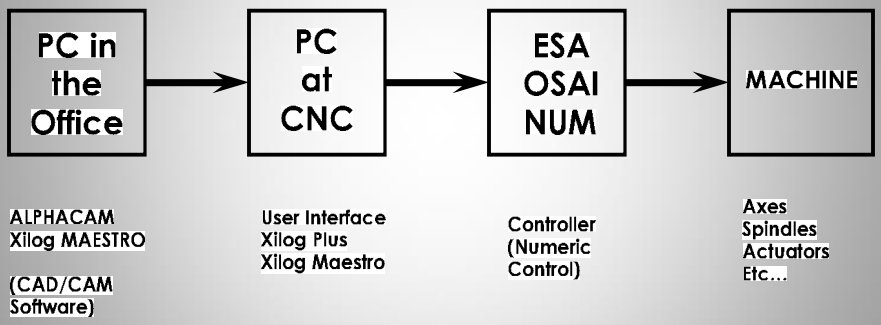
CNC stands for ¡ Computer Numeric Control¢. CNC is a process that allows us to control machines that execute a variety of operations, in response to a program that the user inputs. Operations such as Routing, Drilling and Profiling are of special interest to this course.



How does a CNC Router work?



Process of a Making a Program



ALPHACAM
Xilog MAESTRO

(CAD/CAM Software)

User Interface
Xilog Plus
Xilog Maestro

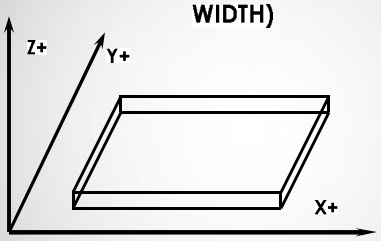
Controller (Numeric Control)

Axes
Spindles
Actuators
Etc...

Main Programmed Axes

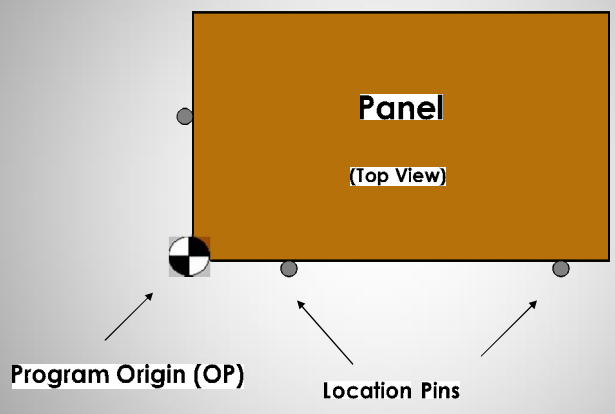
Y+ axis (Along the WIDTH)

Z+ axis (Along the THICKNESS)

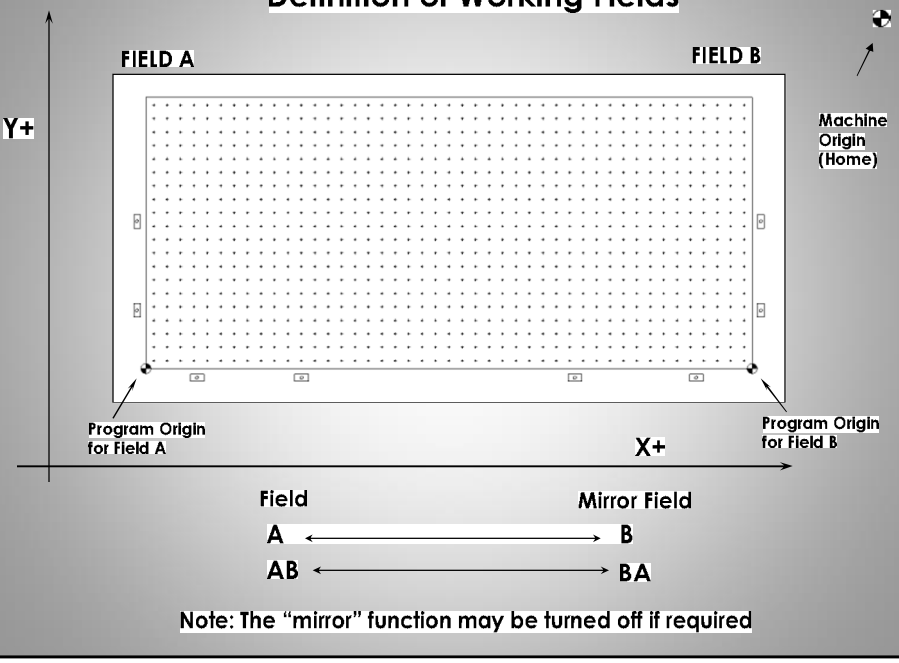


X+ axis (Along the LENGTH)

Reference Points

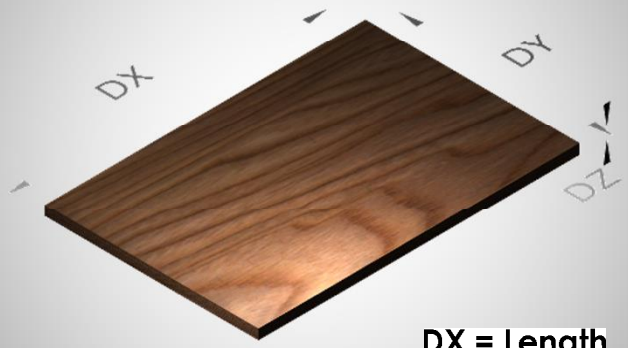


Definition of Working Fields



Note: The "mirror" function may be turned off if required

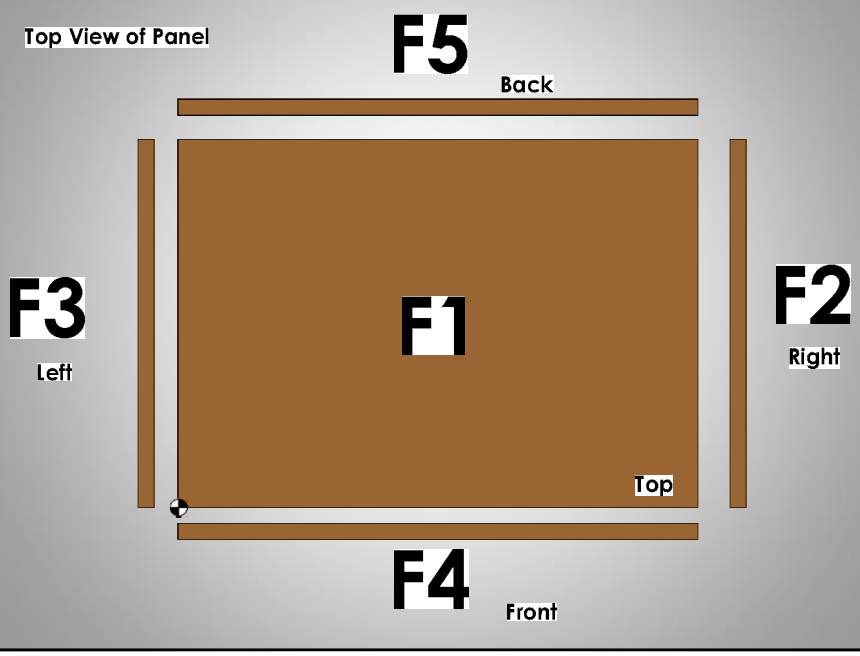
Definition of Panel Dimensions

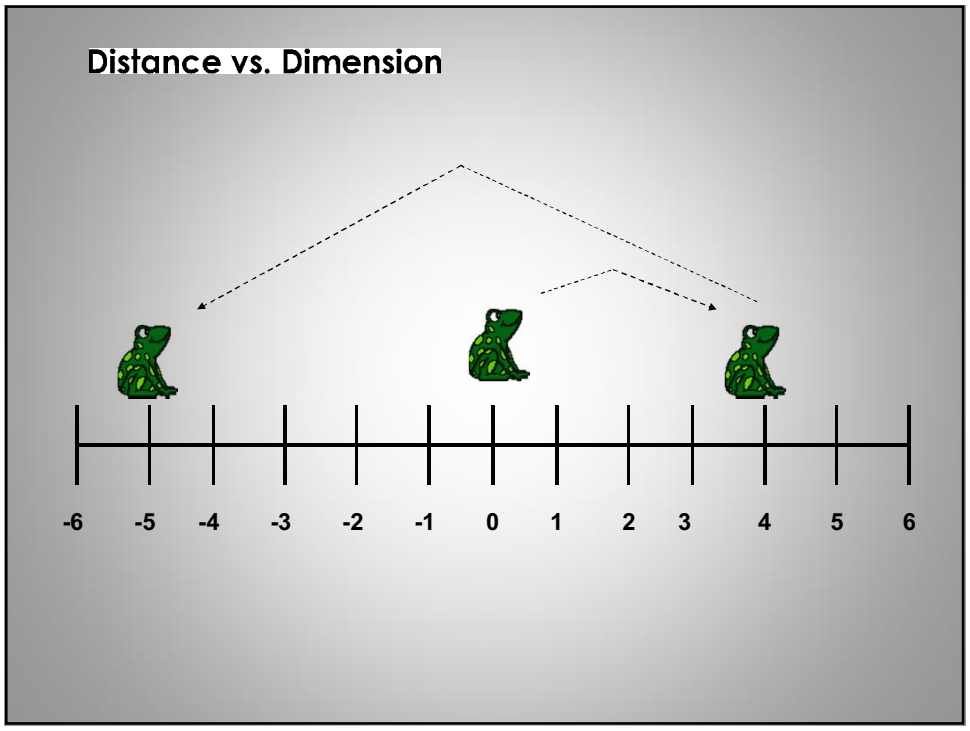


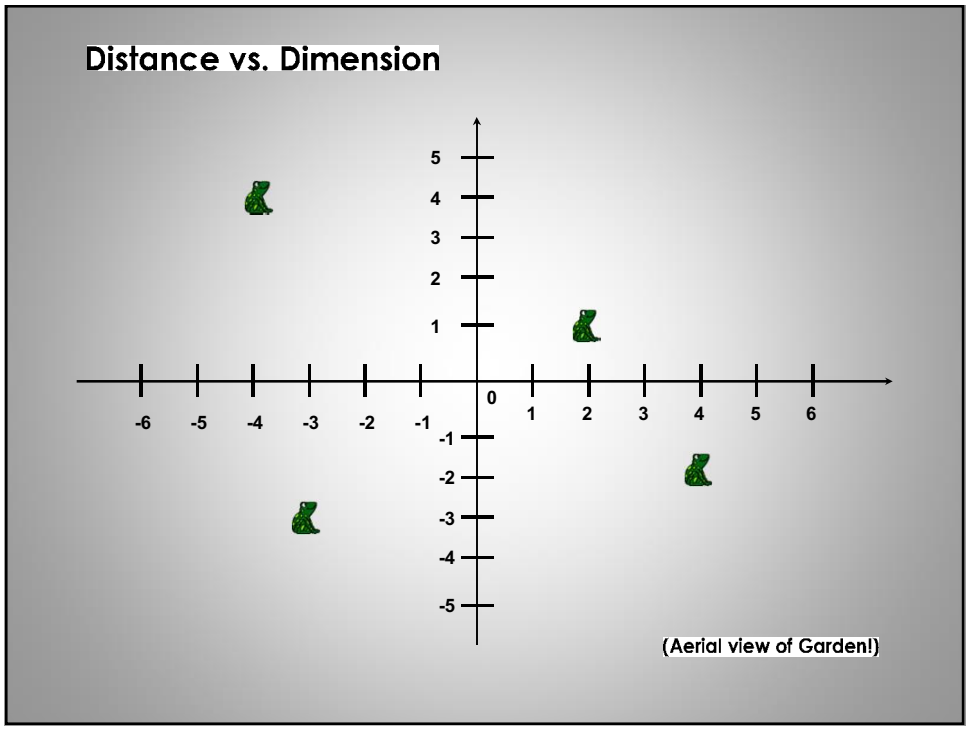
DX = Length
DY = Width
DZ = Thickness

Definition of Panel Faces

Top View of Panel




Distance vs. Dimension
-6 -5 -4 -3 -2 -1 0 1 2 3 4 5 6


Distance vs. Dimension
5 4 3 2 1
-6 -5 -4 -3 -2 -1 0 1 2 3 4 5 6
-1 -2 -3 -4 -5
(Aerial view of Garden!)

Cartesian Coordinates

Cartesian Coordinates are a pair of numbers arranged on a plane that provide the precise location of points on that 2-dimensional space. They where invented by the French mathematician Descartes in the 17th century.

They are an irreplaceable tool that we still use today to help us program and draw parts.

Example:

The following is a square panel that is 11x6 in size; we can use its 4 corner to specify the "coordinates" that make up such geometry.

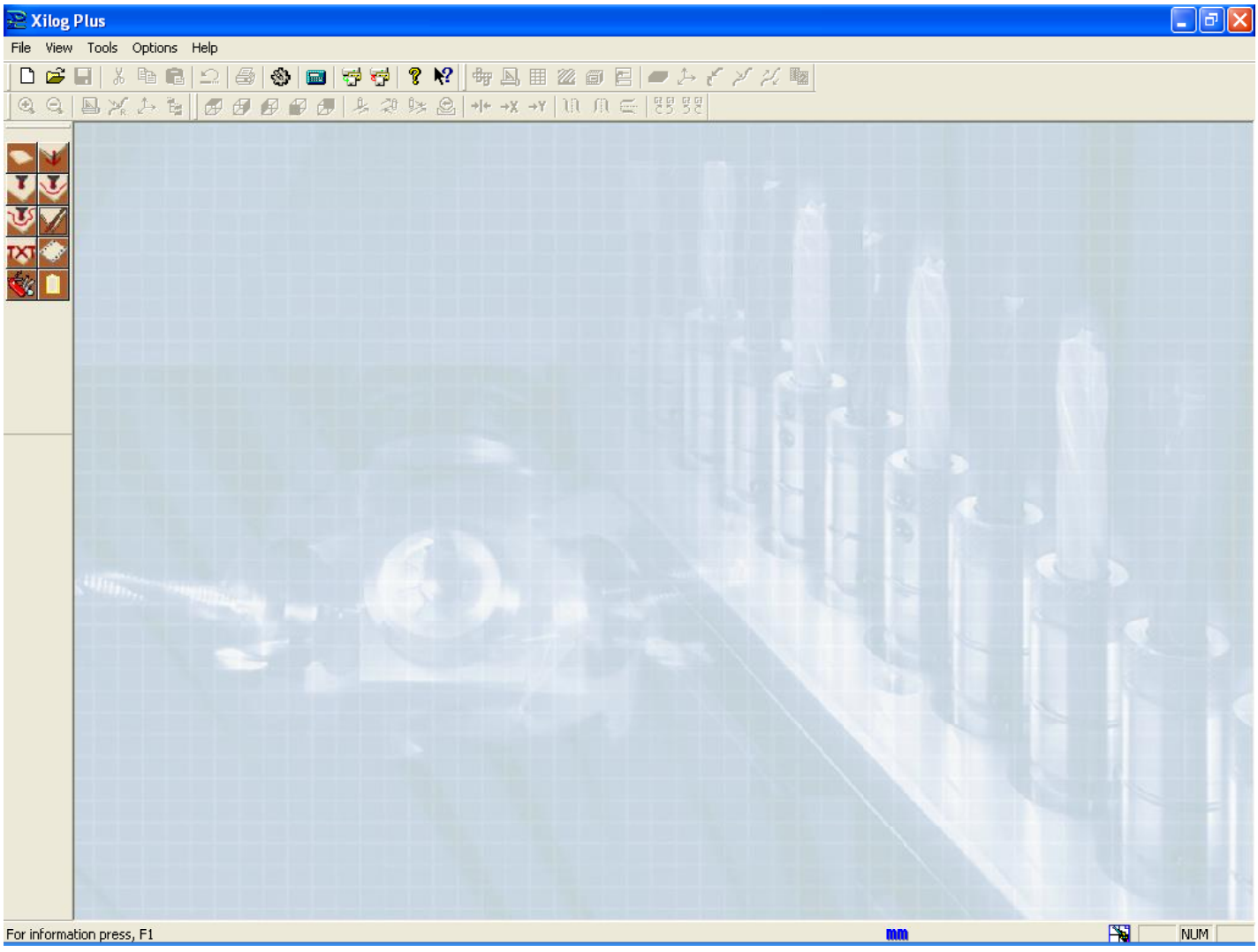
Exercise: Draw the parts that the next sets of coordinates represent:

Example A	Example B
(0,0)	(0,0)
(0,10)	(15,0)
(1,11)	(15,2)
(9,11)	(17,1.5)
(10,10)	(17,0)
(10,0)	(20,0)
(0,0)	(20,10)
	(19,12)
	(17,14)
	(2,14)
	(0,10)
	(0,0)

Xilog Plus User Interface

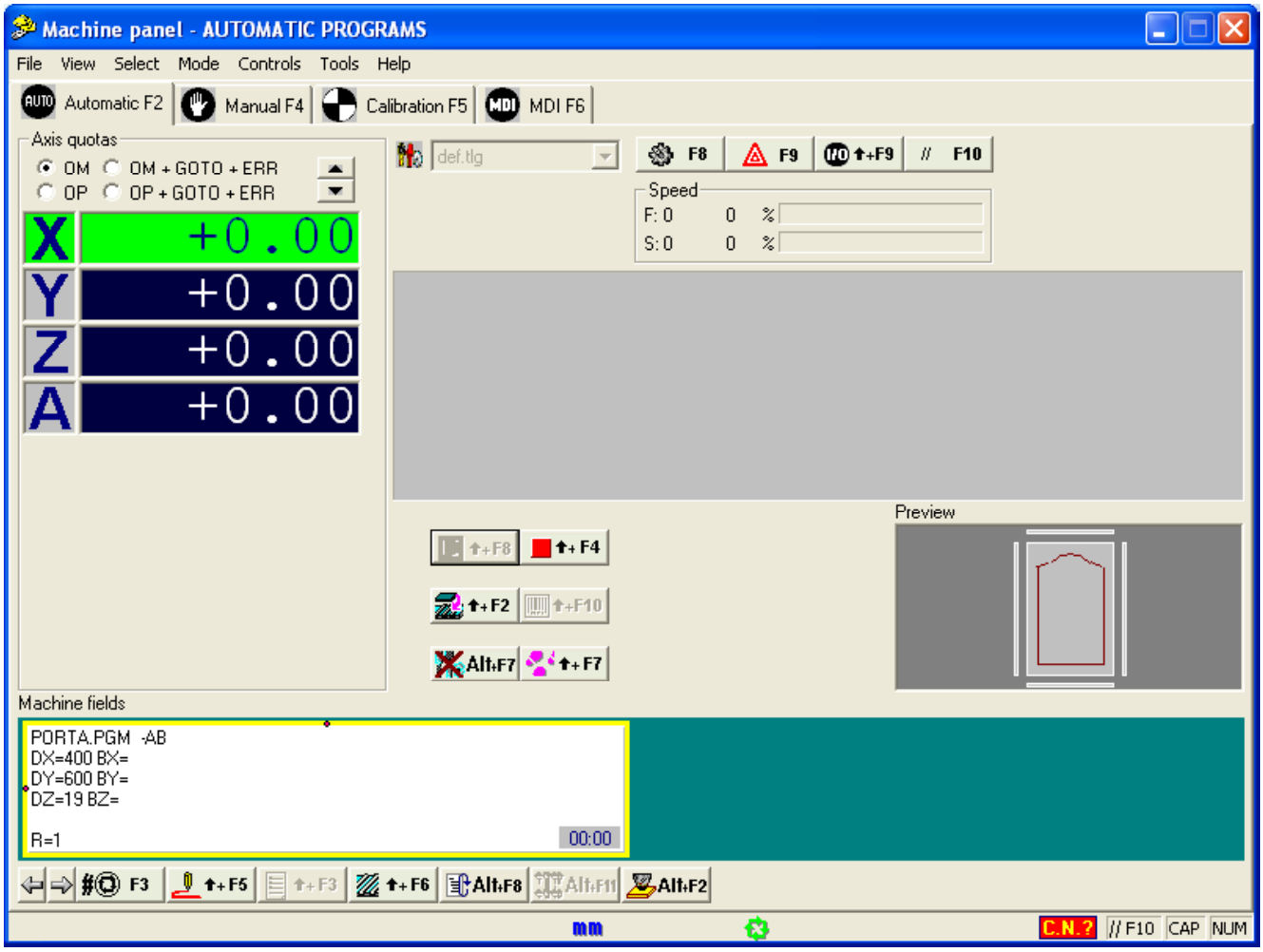
The Xilog Plus User Interface is the software that allows the user to edit programs and macros for the equipment. Xilog then interacts with the controller and delivers the compiled code to the system. Additionally, the interface provides a user-friendly environment where the user can manage the files and parameters relative to the machine.

The Main screen of Xilog looks the following way:



The software is designed for the user to be able to browse through the menus by EITHER clicking on the pull-down menus OR pressing the corresponding shortcut. It is designed entirely for the Windows Operating system.

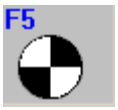
The Panel Mac is an independent application of Xilog that operates the equipment independently. The main screen looks like this:



The following is a brief explanation of the command that each button triggers:

Automatic Execution of a Program. This button allows the user to run a program that has already been written.

Manual movements. Allows the user to move any/all of the machine's axes. Requires remote control or jog knob.



Calibration. Allows the user to "home" ("calibrate", "size") the machine.

MDI. Manual Data Input Menu, this menu allows the user to manipulate individual peripherals such as the Tool Changer or the Electro Spindle. Useful for troubleshooting and testing of the equipment.

Alarms. Creates a list containing the description of all/any emergency condition(s) present on the machine at any moment. If the SHIFT key is held down while pressing F7, we may access 

which enables the user to verify the internal Input and Output signals of the machine.

Basic Programming Instructions

H: (Header Instruction)

The Header Instruction is the most important part of a program, it contains information that is essential for machining and must always be carefully set up. Following is an explanation of each of it's parameters.

DX = Dimension of the Panel X (usually Length). **Required**
DY = Dimension of the Panel Y (usually Width). **Required**
DZ = Dimension of the Panel Z (Thickness). **Required**
- = Working Field. **Required**
/ = Tooling File. Denotes the name of the File that contains all information about tools, by default the name is DEF. **Required.**
R = Number of times that the program will repeat before quitting the program. **Required.**
***** = Units of measurement. Either MM or IN (caps). **Required.**
C = Continuous execution. **DO NOT MODIFY**
T = Raise Bars Enable variable. This is NOT the tool number. **DO NOT MODIFY.**
= name of environmental variables file. **DO NOT MODIFY.**
V = Intelligent clamping of part variable. This is NOT the feed rate. **DO NOT MODIFY.**
BX = Offset on the X axis. **Optional.**
BY = Offset on the Y axis. **Optional.**
BZ = Offset on the Z axis. **Optional.**

B: (Boring Command)

This command executes one or a series of holes, depending on the tools selected (list of tools).

X = X coordinate of the hole. **Required**
Y = Y coordinate of the hole. **Required**
Z = DEPTH desired for the hole. **Required.**
T = Tool list for the operation. The First tool is the REFERENCE tool. **Required.**
E = Position of the hood. **Optional.**
V = Boring Feed Rate. **Optional.**
S = RPM's of the tool. **Optional.**
G = Number of steps for the boring operation (Pecking). **Optional.**
D = Out-of-work dimension. Allows clearance for longer tools or hurdles in the tool path. **Optional.**

BO: (Optimized Boring Command)

This command executes one or a series of holes, depending on the DIAMETER and TYPE of tools.

X = X coordinate of the hole. **Required**
Y = Y coordinate of the hole. **Required**
Z = DEPTH desired for the hole. **Required.**
D = Diameter of the hole. The First hole will be the REFERENCE hole for successive holes. **Required.**
N = TYPE of tool that will perform the operation. (Brad=P, Lance=L, Countersink=S). **Required.**
L = Taper height for the countersink. **Required** only if N=S.
R = Number of holes desired. **Required.**
x = Distance from one hole to another (displacement, step) on the X axis, if more than one repetition is selected. If no step is desired, it is recommend to type 0. **Required.**
y = Distance from one hole to another (displacement, step) on the Y axis, if more than one repetition is selected. If no step is desired, we recommend to type 0. **Required.**
V = Boring Feed Rate. **Optional.**
G = Number of steps for the boring operation (Pecking). **Optional.**

G0: (Profile START Point)

This command POSITIONS the machine at the desired coordinates and ENABLES the selected tool for ROUTING.

X = X Coordinate where the routing will START. **Required.**
Y = Y Coordinate where the routing will START. **Required.**
Z = Depth at which the routing will START. **Required.**
T = Tool that will execute the machining. **Required.**
E = Position of the dust extraction hood. (0=highest, 4=Lowest). **Optional.**
V = Routing Feed Rate. **Optional.**
S = RPM's of the tool. **Optional.**
D = Out-of-work dimension. Allows clearance for longer tools or hurdles in the tool path. **Optional.**
N = Profile name. **Optional.** For use only with the GREP command.

G1: (Linear cut or Interpolation)

This command cuts in a STRAIGHT LINE, used when ROUTING.

X = X Coordinate where the routing will END. **Required.**
Y = Y Coordinate where the routing will END. **Required.**
Z = Depth at which the routing will END. **Required.**
V = Routing Feed Rate. **Optional.**

G2: (Circular cut, Clockwise Direction)

This command cuts in a CIRCULAR motion, clockwise fashion. May cut a full circle or portion of it (an arc is a portion of a circle), used when ROUTING.

X = X Coordinate where the routing will END. **Required.**
Y = Y Coordinate where the routing will END. **Required.**
Z = Depth at which the routing will END. **Required.**
I = X Coordinate of the CENTER of the circle. **Optional.**
J = Y Coordinate of the CENTER of the circle. **Optional.**
r = Radius of the circle. If we use the **r** parameter, we do not require the (I, J) coordinates, but if left blank (I, J) must be inputted. **Optional.**
V = Routing Feed Rate. **Optional.**

G3: (Circular cut, Counter-Clockwise Direction)

This command cuts in a CIRCULAR motion, counter-clockwise fashion. May cut a full circle or portion of it (an arc is a portion of a circle), used when ROUTING.

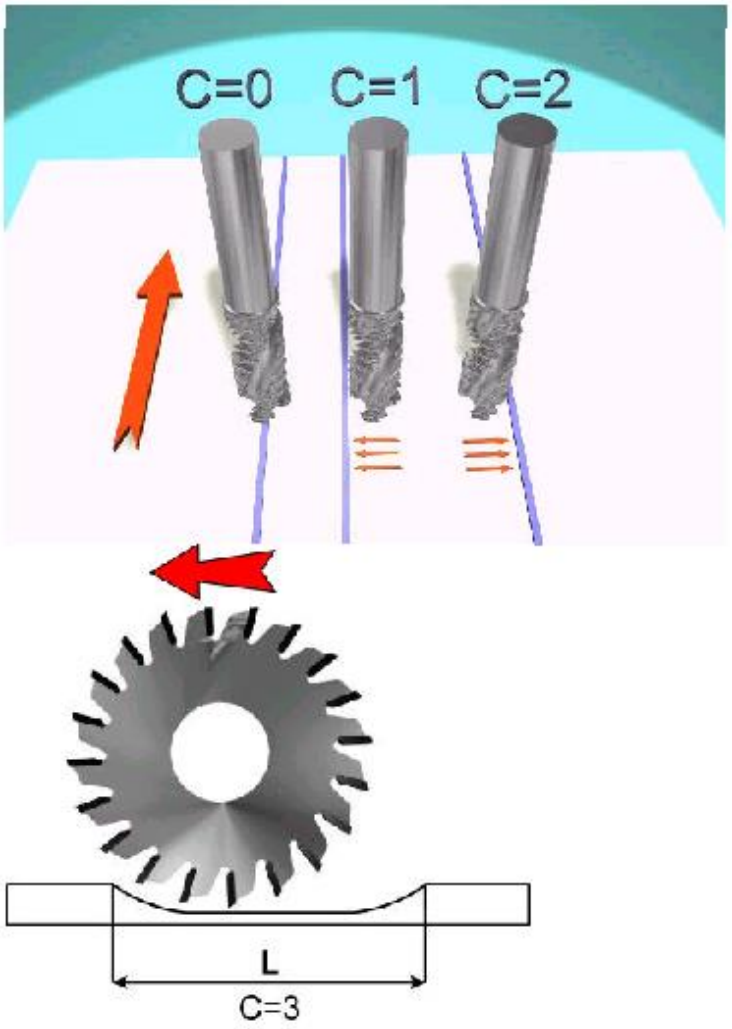
X = X Coordinate where the routing will END. **Required.**
Y = Y Coordinate where the routing will END. **Required.**
Z = Depth at which the routing will END. **Required.**
I = X Coordinate of the CENTER of the circle. **Optional.**
J = Y Coordinate of the CENTER of the circle. **Optional.**
r = Radius of the circle. If we use the **r** parameter, we do not require the (I, J) coordinates, but if left blank (I, J) must be inputted. **Optional.**
V = Routing Feed Rate. **Optional.**

C: (Tool Compensation)

This command is used when we wish to apply compensation to a routing. See Fig 1.

C = 0 No compensation selected.
C = 1 Compensation selected to the RIGHT HAND side of the path.
C = 2 Compensation selected to the LEFT HAND side of the path.
C = 3 Length-of-Radius Compensation. Used only when routing with a saw.
S = Machining allowance. This is a tolerance that enables us to fine-tune adjustments such as those needed because of tool sharpening, incorrect sizing/squareness of the part, etc.

FIG 1. Examples of compensation:



N: (Null Command)

This command allows us to position the machine at a desired coordinate AT MAXIMUM SPEED. It is very useful when we need to move the machine out of the way for any reason. Traveling will occur with the Electro Spindle OUT of work position.

X = Desired X coordinate where we want the machine to move. **Required.**
Y = Desired Y coordinate where we want the machine to move. **Required.**
V = Speed of displacement. If left blank, machine will move at MAXIMUM speed for the given axis. **Optional.**
S = RPM's of the tool. As an option, a tool may be selected and the RPM's defined. **Optional.**
T = Tool that will execute the NEXT machining. **Optional.**

SX / SY: (Mirror commands)

These two commands will automatically MIRROR any coordinate inputted AFTER the command. They are "flags" that need to be turned either on (1) or off (0) hence, it is important to remember to turn the OFF after we used them.

The following program reflects the part:

H DX = 600 DY = 400 DZ = 20 -A R99 /DEF
1. BO X = 9.525 Y = 75 Z = 11 D = 8 N = P R = 4 x = 0 y = 150
2. SX = 1
3. BO X = 9.525 Y = 75 Z = 11 D = 8 N = P R = 4 x = 0 y = 150
4. SX = 0

Note how lines number 2 and 4 are IDENTICAL, nonetheless, the Software is producing the MIRROR image of such holes.

IX / IY: (Incremental machining)

These parameters toggle ON and OFF the INCREMENTAL machining mode. This means that all actual coordinate is referenced FORM THE LAST coordinate that was written. As with the mirror command, it is advisable to always return it to it's OFF position after use.

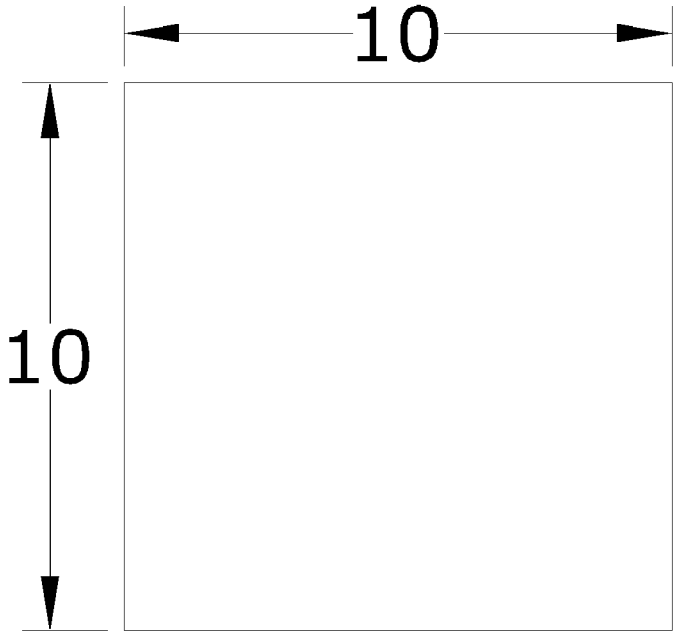
Considering the panel above:

H DX = 600 DY = 400 DZ = 20 -A R99 /DEF

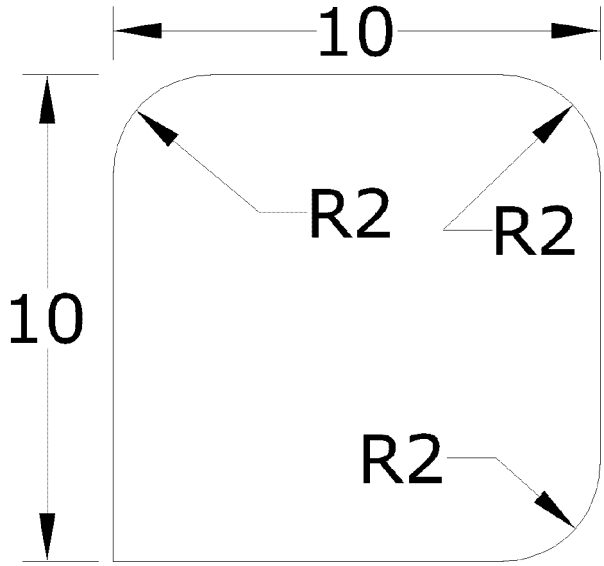
1.	IY = 1	
2.	B X = 9.525	Y = 75
3.	B	Y = 150
4.	B	Y = 150
5.	B	Y = 150
6.	B X = 590.475	Y = 0
7.	B	Y = -150
8.	B	Y = -150
9.	B	Y = -150
10.	IY=0	

Notice how the Y coordinate depends on the accumulated coordinate before it, so that each instruction is relative to the preceding one. Furthermore, since the IX = 1 IS NOT selected, the coordinate on the X is still ABSOLUTE and referenced from the LEFT EDGE of the Panel.

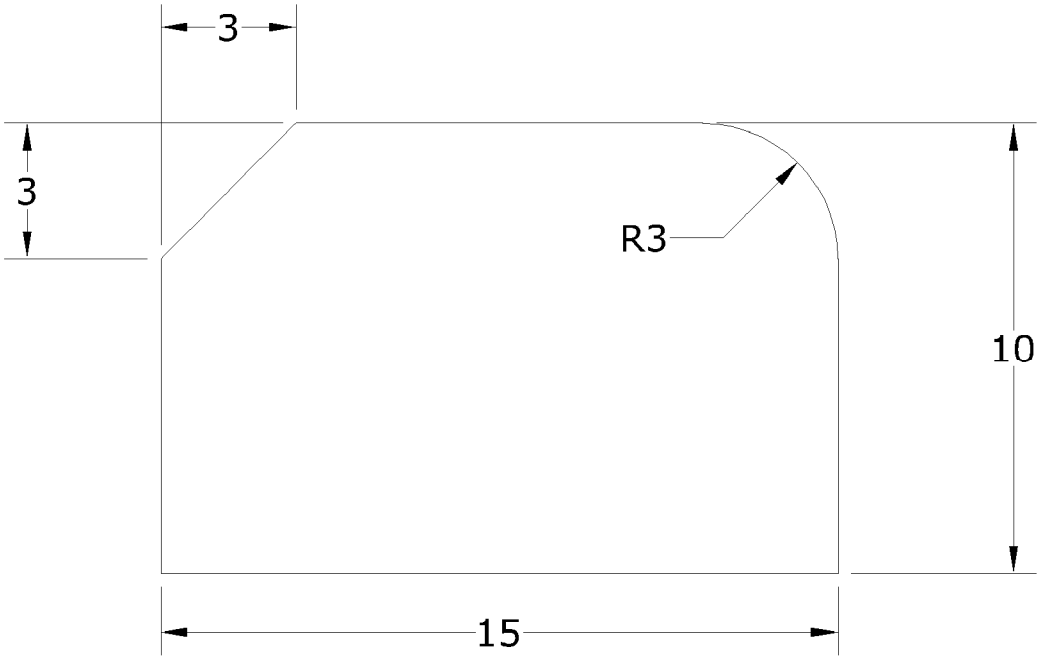
G-Code Exercises



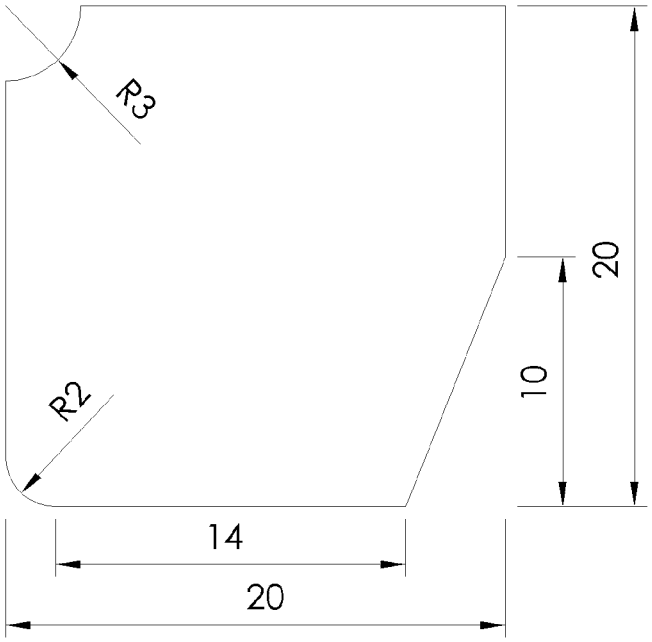
Start CW at (0,0)



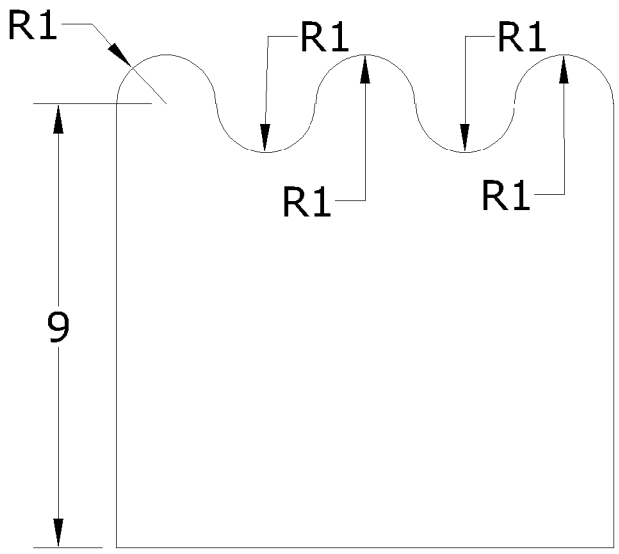
Start CW at (0,0)



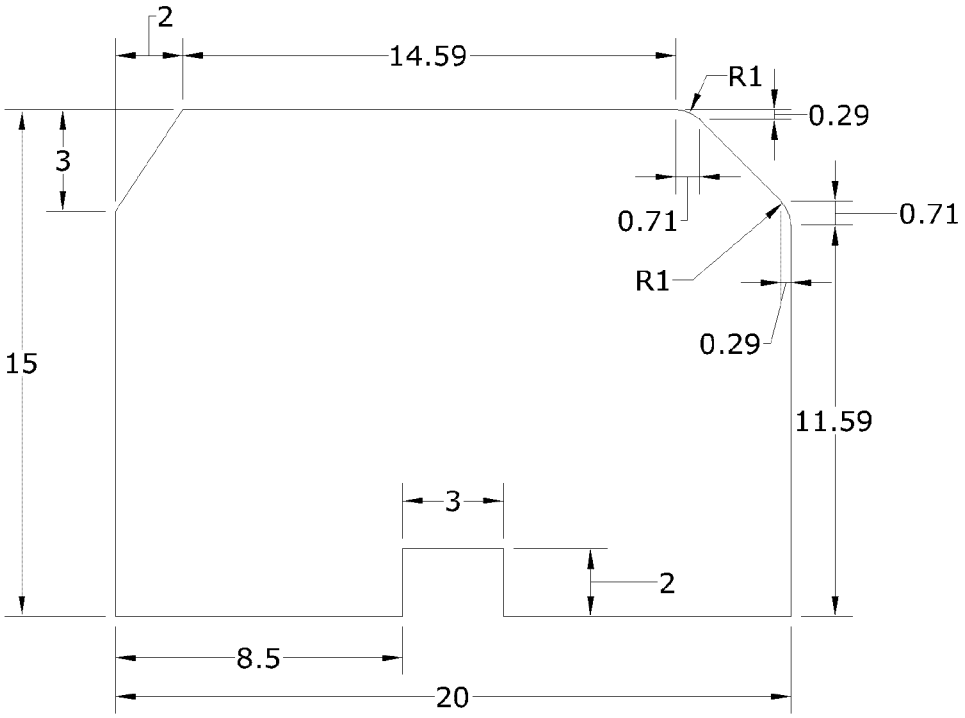
Start CCW at (0,0)



Start CW at (0,0)



Start CW at (0,0)



Start CW at (0,0)

Definition of CAD/CAM

CAD stands for Computer Aided Design. It is a very helpful tool that we utilize to accurately expedite the drafting process. Additionally, it provides a unique platform for the development of new designs and ideas.

CAM stands for Computer Aided *Manufacturing,* which is a *secondary* process that happens only after the design has been created, therefore there can be no CAM unless there is CAD. We use the tools inside our CAM software to expedite the manufacturing process.

Basic Geometric Concepts

The following concepts will be very helpful during the drafting process, since drawing are created based on *geometry*, it is important to define the basic concepts behind the entities that create geometries: lines and arcs.

Point: A specific coordinate precisely defined. A point has no width or length, and it only specifies a location.

Line: A region defined by at least two points.

Intersection: Point where two lines co-exist in a plane (point where they meet).

Parallel Lines: Two lines that have the same *slope*. Hence two parallel lines will never intersect each other.

Perpendicular lines: Two lines that form a right angle (90°)

Tangent Lines: A tangent line is a line that makes contact at a single point, but does not intersect. (Used when generating smooth curves i.e. "French" curves).

FILE Menu

The **FILE** Menu contains the following relevant functions:

 **New** Enables the user to create a NEW drawing.

 **Open** Allows the user to OPEN or EDIT a drawing that he previously saved.

 **Insert** Allows for the INSERTION of another AlphaCAM™ drawing inside this one, therefore the name of the previous drawing is maintained.

 **Save/Save As** This feature allows the user to save his work. If "save as" is selected, a different name other than the first one must be used to prevent a file share violation.

 **Input CAD** This tool allows the user to import other CAD files (DXF, DWG, etc…)

 **Output NC** This tool runs the Post-processor and generates Code.

 **Select Post** Allows for the selection of the adequate Post-processor. The default Post must be selected only once when installing the software, afterwards AlphaCAM will use it as the default selection.

 **List NC Code** This option allows the user to check the Code that the Post is generating at ANY GIVEN TIME during the process.

 **Printer/Plotter** Prints the contents of the file. See also "Print Preview"

 **Configure** This feature is only available via pulling down the FILE Menu. It contains important customization information such as the Button Bars and General menus.

 **Parametric Macro** This function is for use only by those people who already have parametric Macros and know how to use them (legacy). AlphaCAM now uses VBA as the macro-developing engine.

EDIT Menu

The EDIT Menu contains the following relevant functions:


Undo This is the preferred method for correcting mistakes, use it whenever possible.


Delete Deletes from the screen the selected entities.


Move Will move the selected entities from point A to point B.


Copy Creates a copy of the selected entities. User must input a reference point.


Repeat Generates multiple copies of the selected geometries based on a reference point and a second point that provides both direction and displacement.


Rotate This command may rotate a single entity or may rotate AND generate ROTATED COPIES of the selected items, similar to POLAR ARRAY in AutoCAD™.


Array Generates "rectangular" copies of the selected items. Requires number of "rows", number of "columns" and their respective displacements.


Mirror Generates the mirror image of the selected items. Needs a "mirror line" which is the reference point for mirroring.


Scale Enlarges or reduces the selected items, based on a scale factor.


Stretch Produces the irregular but proportional enlarging/reducing of an entity. Useful whenever we need to enlarge/reduce entities and the scale factor is not known.


Break Produces a discontinuity at the selected point on the entity.


Trim Will shorten the selected entity to the reference entity/geometry.


Extend Will extend the selected entity to the reference entity/geometry.


Join Blends two (or more) geometries into one. The END POINTS of such geometries MUST touch each other!


Explode Opposite of JOIN, it dissociates a geometry into its constituting entities.


Fillet Blends two lines with the designated radius.

 **Chamfer** Blends two lines together with the designated chamfering distances.

 **Offset** Generates a copy of the selected geometry/entity at a specific distance called the "offset distance".

 **Change** Allows the user to change the type of the entity (Construction, Geometry, Toolpath, Material, etc…)

VIEW Menu

The VIEW Menu contains the following relevant functions:

 **Zoom All** Displays THE ENTIRE contents of the drawing.

 **Zoom Window** Used for zooming in into details. Prompts the user for a WINDOW (two points) that defines the ZOOM AREA.

 **3D Views** Displays all the views of the geometry (TOP, FRONT, LEFT and ISOMETRIC).

 **3D Simulation** Provides a solid-model rendering of the machined part.

 **Redraw** "Refreshes" the screen.

GEOMETRY Menu

 **Line** Creates a line based on two points.

 **Arc** Creates an arc (portion of a circle).

 **Rectangle** Creates a rectangle based on two of its corners.

 **Circle** Creates a circle.

 **Text** Generates text

 **Special Geometries** Generates useful Geometries, such as the "enclosing rectangle".

SNAP Functions

The SNAPS are a set of functions that are extremely helpful when drawing. They use properties of the geometries as reference points hence, making the selection perfect, quick and easy.

GENERAL: Function Keys
Most of the UTILS menu buttons have Function Key equivalents and buttons.

F3 = ORTHO 
F5 = SCREEN GRID SNAP 
F6 = END of 
F7 = MIDDLE of 
F8 = CENTER of 
F9 = INTERSECTION of 
F10 = TANGENT to 
F11 = PERPENDICULAR to 
F12 = PARALLEL to 

If the <Ctrl> key is held down when F3, or F5 - F10 are pressed, the Snap is modal, and will remain active until the mouse Right Button (or <Esc> key) is pressed.

Also, **F1** = UNKNOWN
F4 = CLOSE and FINISH geometry 

All of these functions may also be summoned by their FUNCTION KEY to expedite the process.

MACHINE Menu

The following are the relevant MACHINING commands that we must use. They are being displayed IN THE CORRECT SEQUENCE that the user must follow:


Select Tool

The first, and most important, step is the selection of the adequate tool, which is crucial for successful machining.


Tool Directions

Once the tool has been assigned, the appropriate tool direction must be set.

After these two parameters have been set, we may machine with ANY of the following methods:


Rough or Finish

Used when dimensioning or profiling parts, generally.


Pocketing

Used when pocketing material generally.


3D Engraving

Used when engraving letters or machining doors generally.


Drill/Tap Holes

Used when drilling a part.

If the machining has been generated successfully, the following icons will appear:


Tool Lead In/Out

Applies proper Lead-In/Lead-Out to the tool (use when applicable).


Edit Operations

AlphaCAM generates a LIST of OPERATIONS describing the machining being done on the part. This button allows for operation MANAGEMENT and EDITING.


Update Toolpaths

If any changes have been made to the geometry after the machining has been applied, clicking on this button will update those changes and the toolpaths change accordingly.

Tool Definition

Accurate tool definition is crucial for proper machining of the parts, since AlphaCAM™ uses the information provided by the tool to machine, therefore the better we set up our tools, the more accurate the description of the software will be.

You may access the Tool Definition command by clicking on MACHIINE→DEFINE TOOL. The following screen will appear:

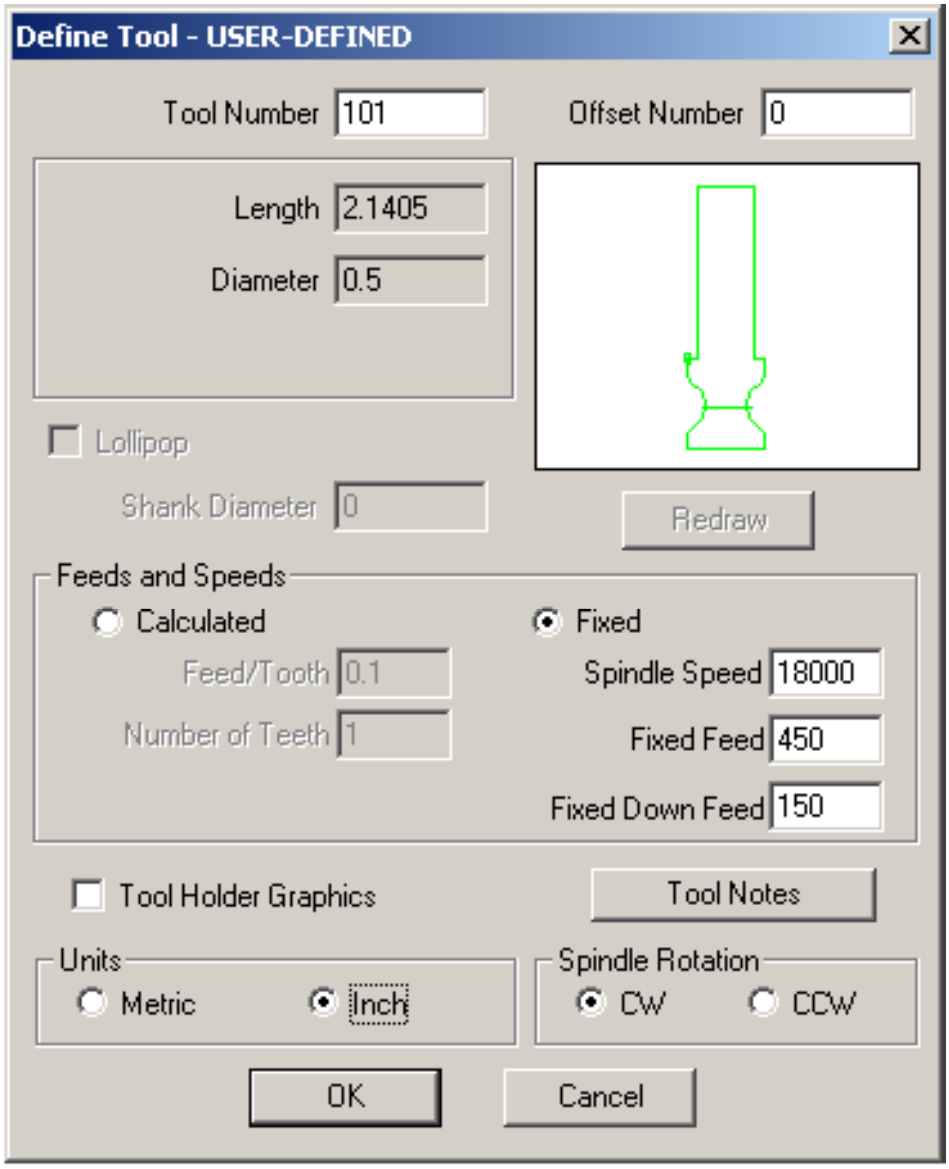
The specific process of generating a tool depends on the TYPE of tool that we are trying to define. Overall, the following considerations should be kept in mind whenever defining a tool:

a) Know before hand your tool numbers and offsets.
b) NEVER mix units (Inch or Metric).
c) Always check rotation direction.
d) Save the tool in a dedicated folder for Metric or Inch and use a name that is meaningful.

Tips for User defined tools:

a) Draw symmetrically (try to use the MIRROR function)
b) Always remember to set your CIRCLE and LINE for reference (see Help File for details).

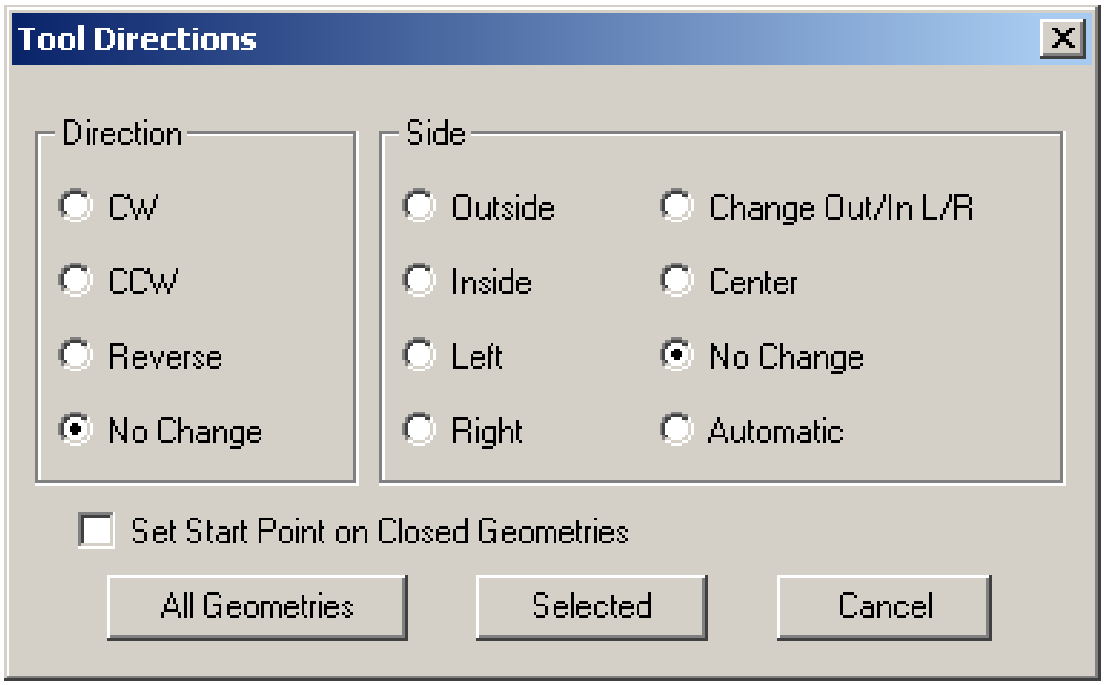
Example of Definition on a bull nose cutter:



Note that the software, thanks to the provided CIRCLE and LINE that define length and diameter respectively, has automatically assigned values to those boxes (this is the reason why the "diameter" and "length' boxes are gray).

Tool Direction

After the proper tool has been selected, this is the next step. Probably, this is the most sensitive of parameters; based on what we select here, several assumptions will be made by the Software. It is VERY important to define the right Tool Direction before attempting any cut.



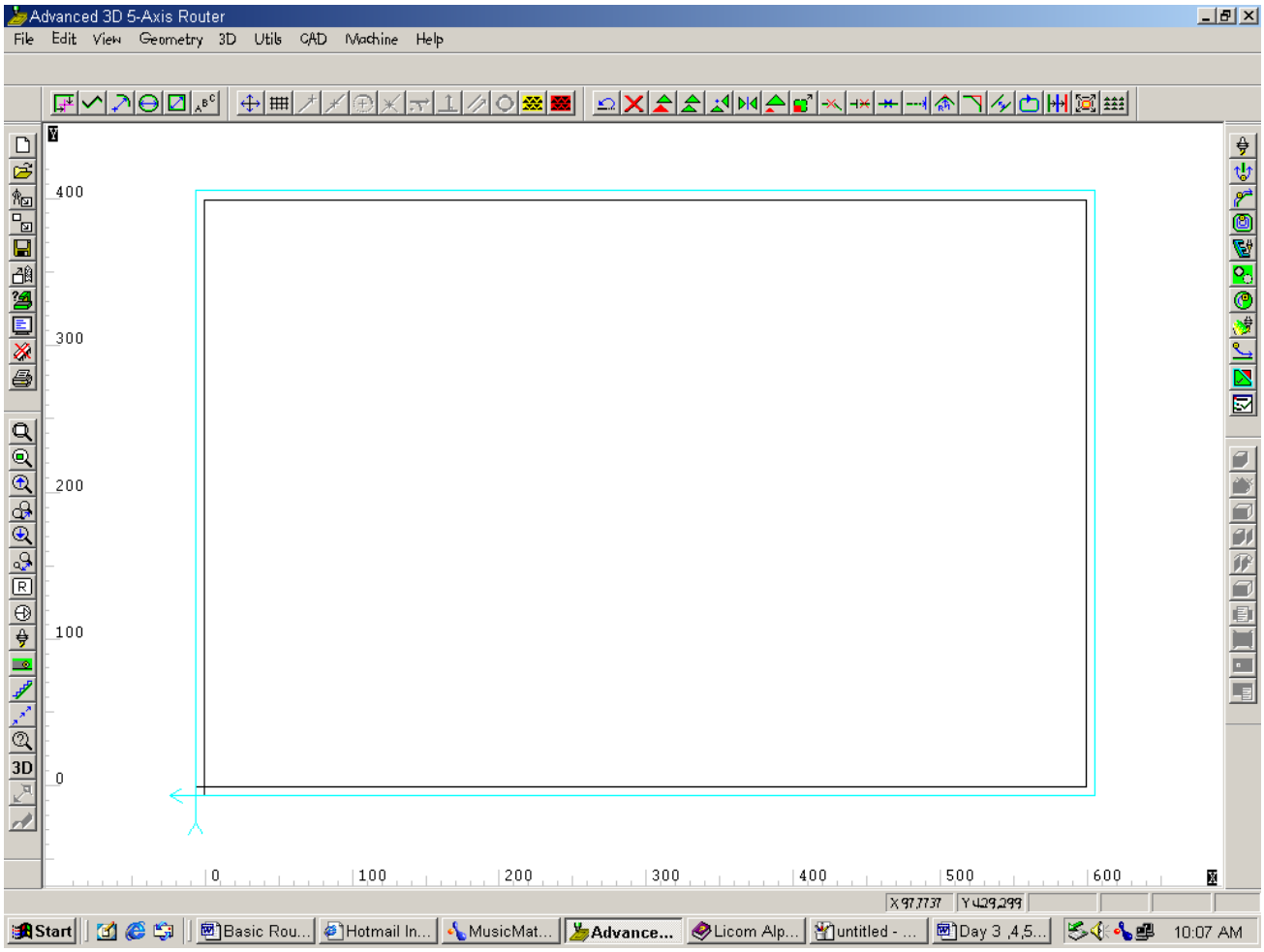
The above selection will climb cut a CLOSED geometry in a clockwise fashion, compensating to the OUTSIDE of the geometry. Notice how, if we wanted so, we could also change the start point on a closed geometry.

Rough or Finish

Rough or Finish is the first machining METHOD that we will discuss. It represents the majority of the work performed on a router (usually every parts need to be squared, shaped, profiled or modified).

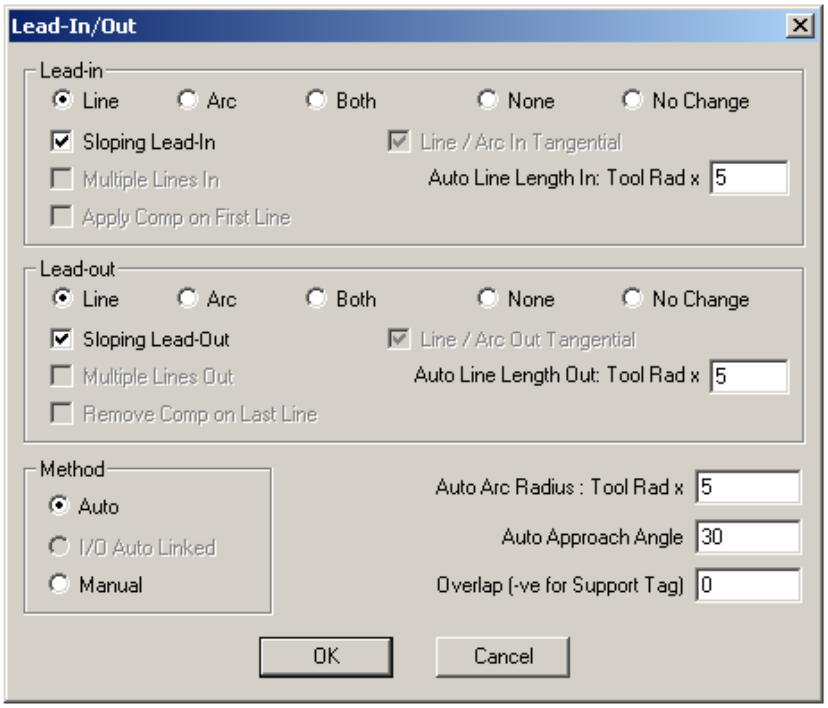
Please draw the next simple part and apply machining to it:

Tool: Half inch cutter
Tool Direction: CW Tool Side: Outside

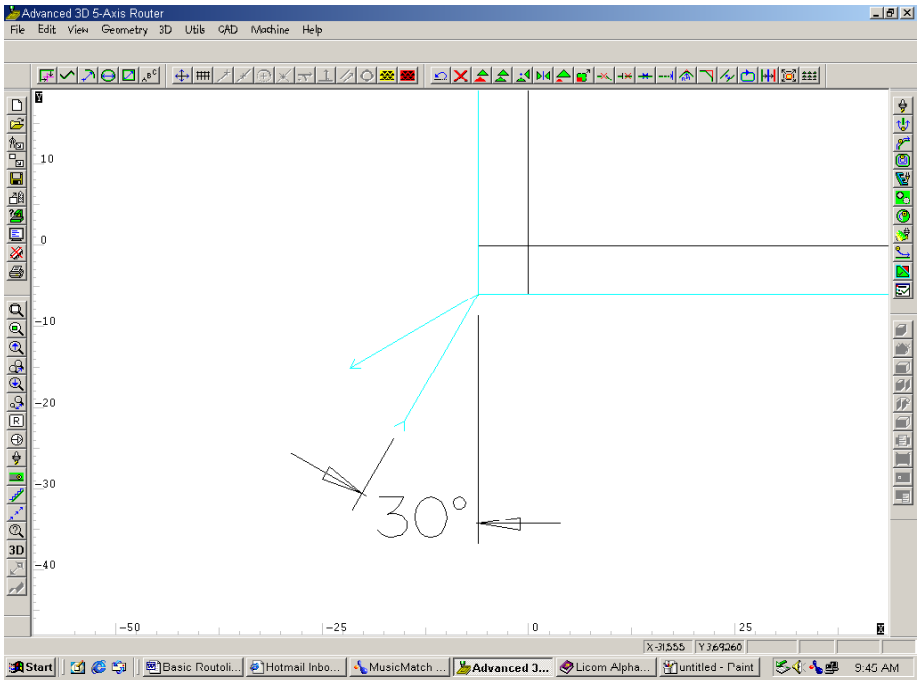


Lead-In/Lead-Out

Lead In/Lead-Out is a SECONDARY process that we apply to the toolpath once that the machining method has been applied. We may choose NOT to Lead-In/Lead-Out our toolpaths and the result is a direct plunge on the Z-axis, however, most materials require adequate Lead-In/Lead-Out to prevent chip-out or damage to the edges.



Below, a detail of the previous exercise, but with a Lead-In/Lead-Out of 30°.

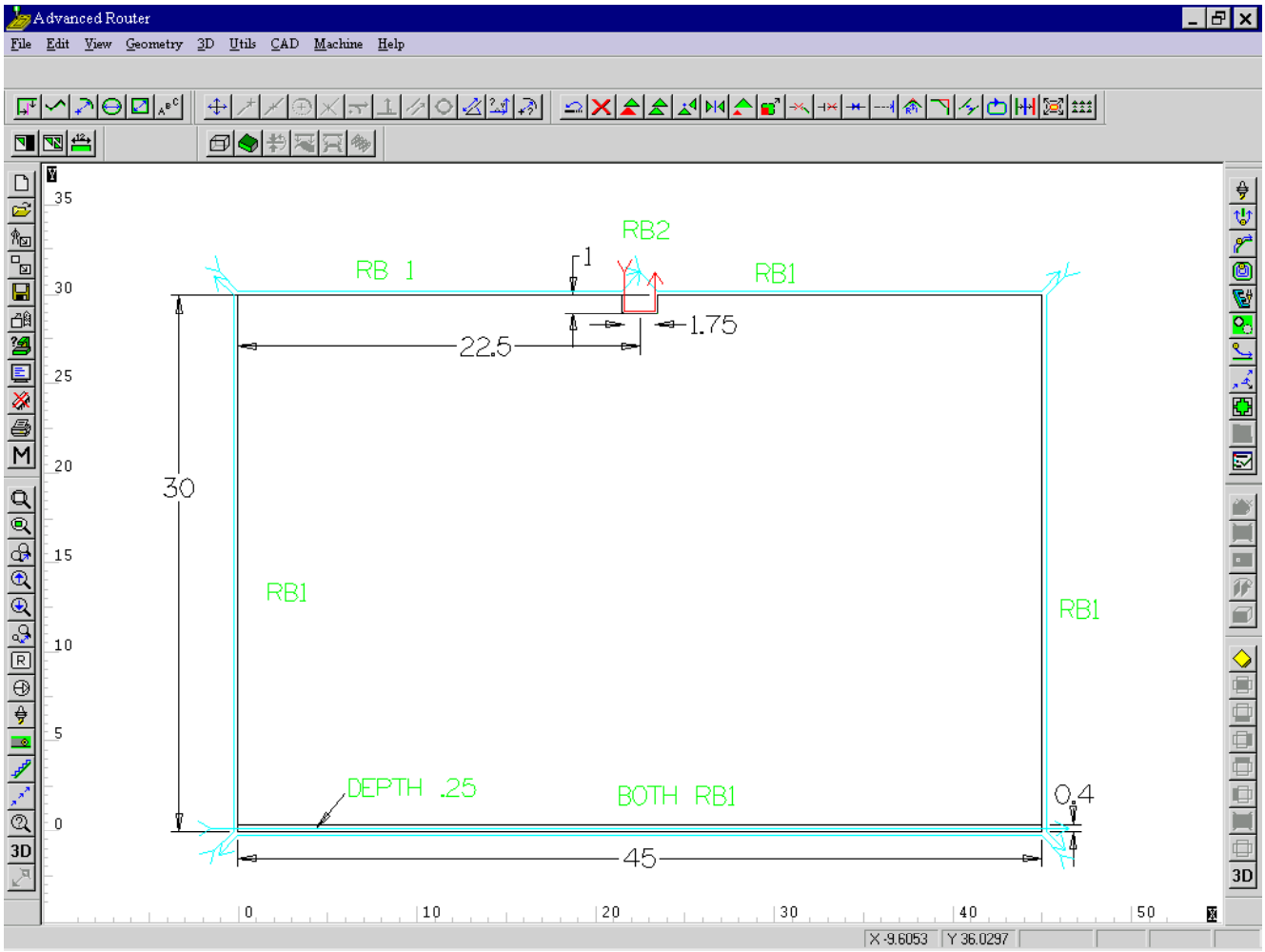


Partial Cutting In Rough/Finish

Sometimes it is desirable to only to use a tool on a specific PORTION of a closed geometry. Take the following part as an example:

We have the need to trim the part, but the minimal radius accepted at the notch is only ⅛" so, it would be a smart idea to cut the whole part with a bigger tool and finally use a ¼" to get that specific radius at the notch. Note how the whole part as been trimmed using ONLY partial cuts.

RB1 is a ½" Cutter with a depth of ¾". Except where indicated otherwise
RB2 is a ¼" cutter with a depth of ¾"
Apply Lead-In/Lead-Out at 45° for the trimming and 0° for the Rabbet



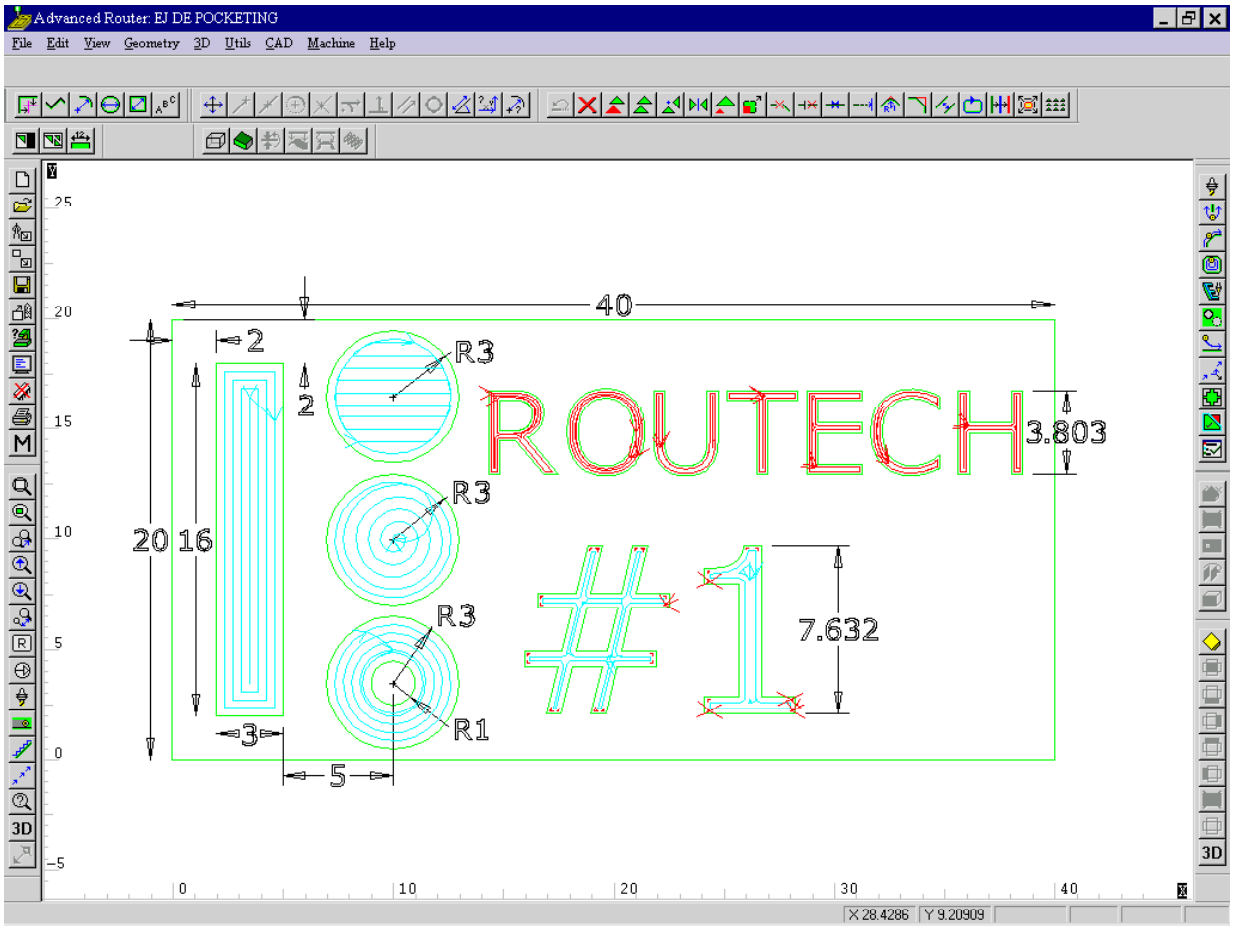
Pocketing

Pocketing is the function that allows us to remove heavy quantities of material. It is automatic for the most part however, the Tool Direction must always be set up manually and this happens to be the most important factor when applying pocketing.

Let us consider the following example:

Note how the larger portion of the material (whenever the application allows) will be removed by the ½" cutter, whereas the lettering which requires something more delicate is pocketed with a ¼" cutter.

The Blue toolpath is a ½" cutter with a depth of ½"
The Red toolpath is a ¼" cutter with a depth of ⅛"

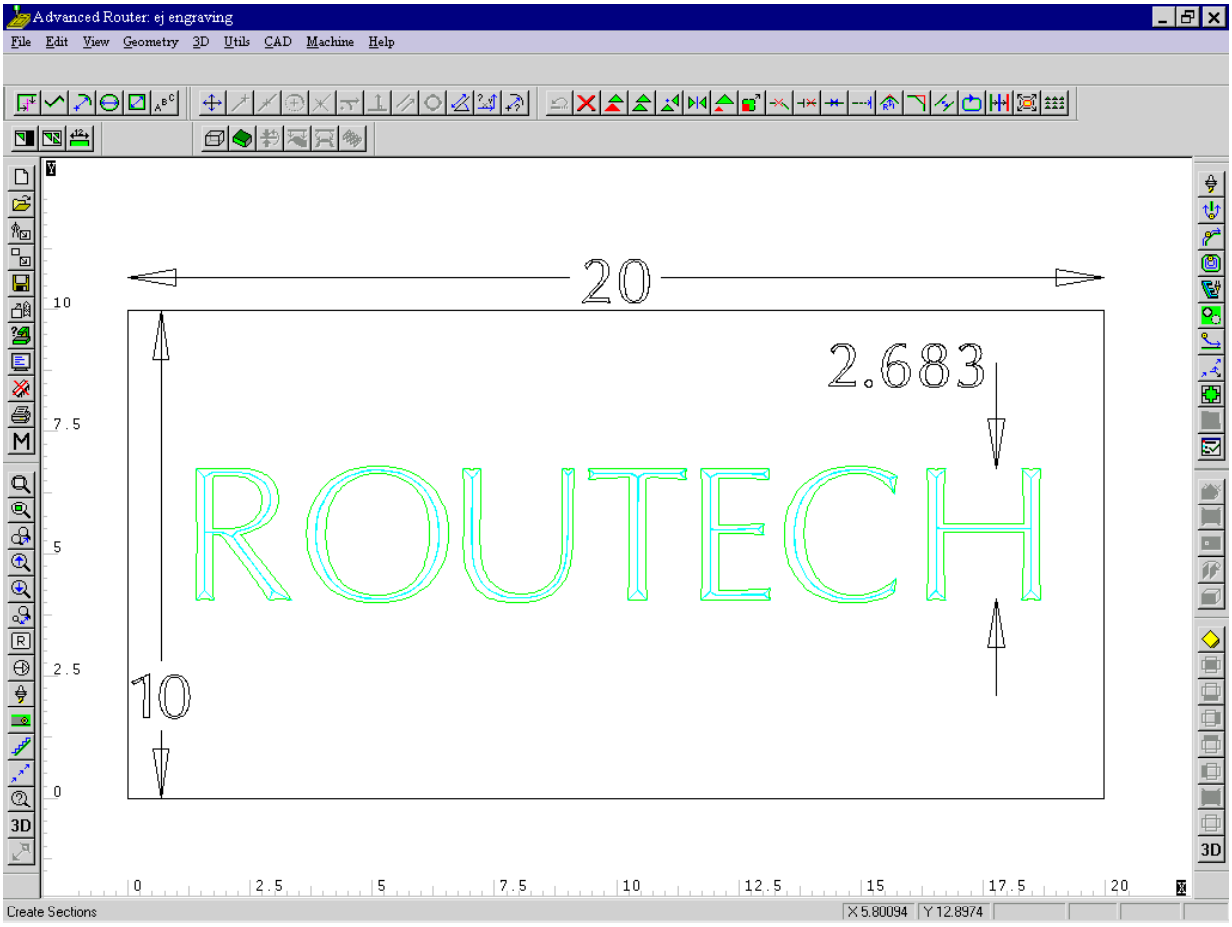


Engraving

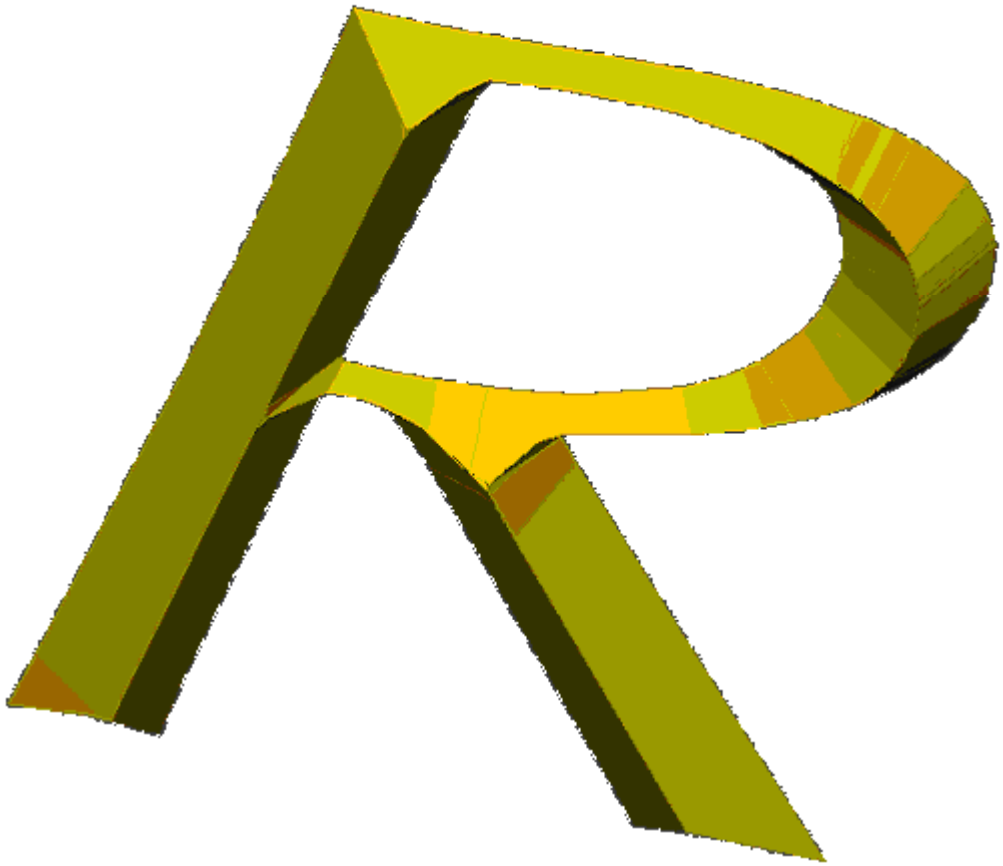
Engraving is the third machining method that we will address. However, the following things should be kept in mind:

a) The tool MUST be a user-defined tool. If the tool is not perfectly symmetrical, the 3D simulation may not work properly.
b) Engraving is an ITERATION process, the higher the resolution the more accurate it will be, but it will also generate more code.

Let's consider the next example:



Detail of engraving generated by the Advanced 3D Simulator.



Nesting

Nesting is the process of fitting different parts on a full sheet (either 8'x4' or 10'x4') to obtain better yield from the sheet stock; it is also a considerable timesaving routine that expedites the generation of single items. Since we are able to cut different parts and different shapes on the same sheet, we can generate short runs without going through the panel saw.

There are three know ways for Nesting:

a. Manual Nesting
b. Automatic (selecting the parts from the screen)
c. Nesting List (using a pre-generated nest-list)

The following is an example of a complex part successfully nested by the algorithm: