E-FCS

GraniCAD Manual

TOPFIBRA

EFFECTIVE FILAMENT WINDING® PIONEERS

E-FCS

GRANICAD MANUAL

TABLE OF CONTENTS

1. GENERALITIES

GraniCAD is a program of CAD/CAM that integrates into a single application the stages of drawing, creation on the way of cutting with specific properties for each line and generation of programs of numerical control for a machine to cut stone with diamond wire.

Your main characteristics are:

- It permits draw drawing straight, three circumference types, five types of the arc of circumference and text.
- It forces references to dots to facilitate the design: Grid, endpoint, halfback, quadrant, intersection and next dot.
- It permits working with several units already configured: millimetres, centimetres, meters, inches, feet and a definable unit for the user.
- It permits multiple levels of undoing and redoing.
- It disposes of special operations of drawing: moving, rotating, dividing stretching, lengthening and symmetry.
- Cutting path generation assisted by the user, which may indicate the order of strokes and modify the speed, side to respect, final pause and move the wire for each stroke.
- Selection of diameter for the wire.
- Possibility of an edition of the orders of numerical control generated and save the changes.
- It permits print drawing, cutting traces and orders of numerical control.

2. MAIN SCREEN

When starting the program it will appear always on the following screen.

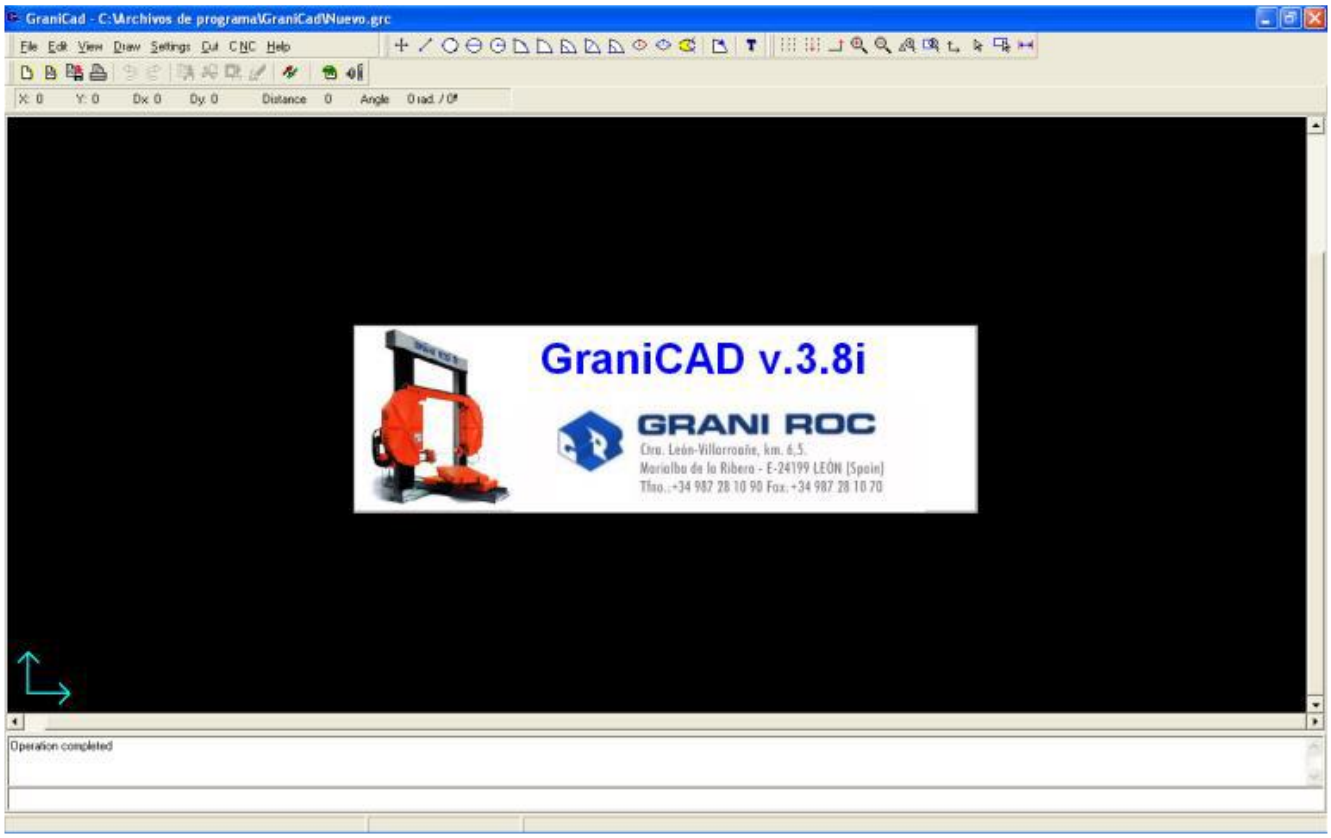

Figure 1

On the screen you can distinguish four areas of work:

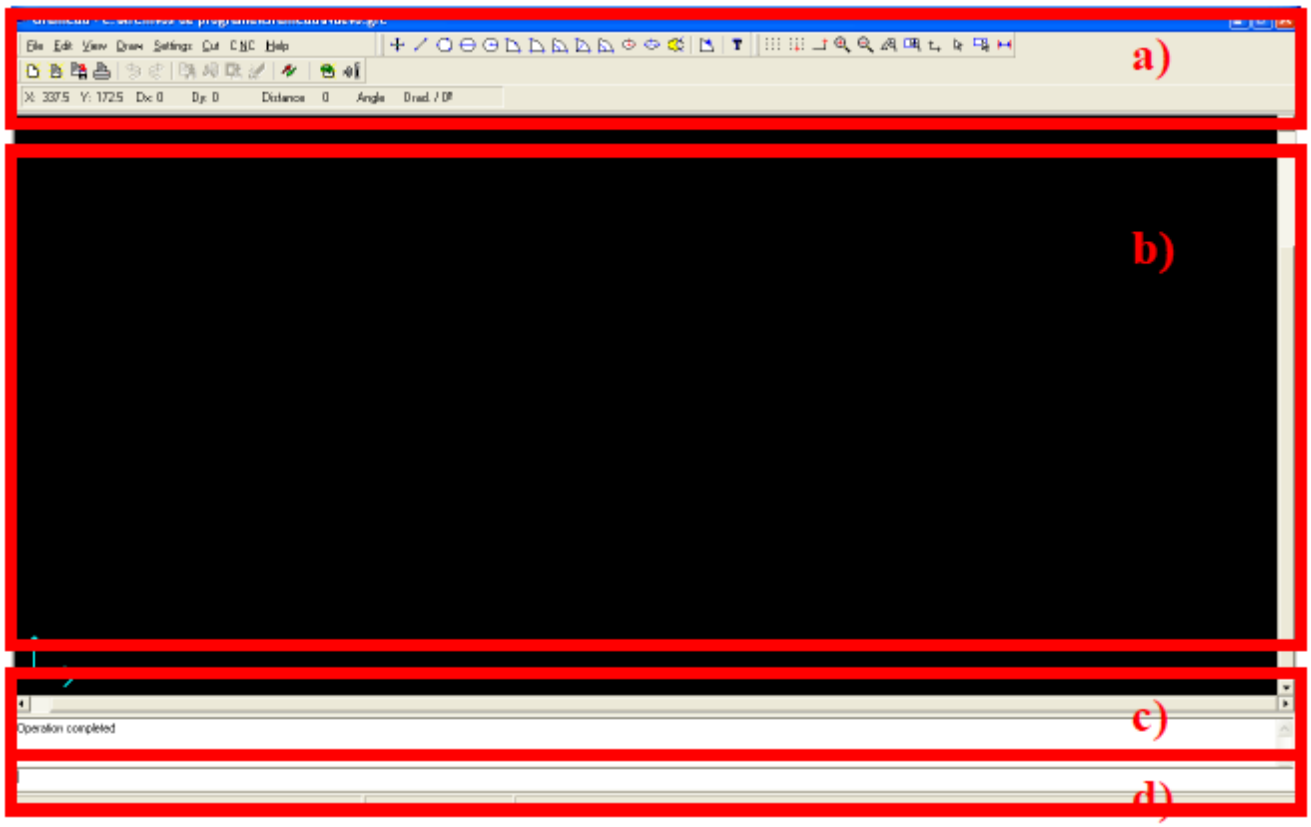

Figure 2

a) Area of the toolbars. In it, you can find the bar of the menu, which contains the main orders. Also, it disposes of the toolbars that serve as shortcuts for the orders of the menu. The bars can be moved and placed in the way you prefer, retaining that position for the next session.

b) Drawing area. It will define the characteristics of the piece, both geometrically: defined section of it, and from the point of view of the cut: defining the cutting parameters.

c) Command Line. You enter the data required by the program and the different orders of the program in text mode.

d) The line of text. The program shows here a brief description of each one of the shortcut icons to go over them with the mouse cursor.

3. COORDINATES

The program works in a two-dimensional plane so that any point is defined by two values, the coordinates X and Y. You can use different units specified in the menu of options. All values are entered using a decimal point as a separator and not a comma (0.52 and 0.52).

The basic types are the absolute coordinates (refer to the origin 0.0), and relatives (refer to the last point) shown in different colours. The active point, there is normally when you create an object that requires more than a point to be defined. Having introduced the first point, this may be used as a reference for the second and so on. The relative coordinates begin with an "R" or "@". Then is inserted horizontally and vertically displacement to the base point.

You can enter coordinates using a keyboard relative distance and direction generated by the mouse.

The coordinates can also be introduced in a polar (magnitude and angle). The format is "module <angle."

The angular values can be introduced in selected units in options, radians or degrees.

If you want to use a different unit can be selected to add the suffix "gra" for degrees or 'rad' for radians.

It allows the use of the word PI for angle values, with the following restrictions: It can only appear once in the expression.

Operations are not supported: PI / 2 is wrong, what is proper 0.5PI.

4. MENU

4.1. Menu

Figure 3

In this menu, you can find the different commands that present GraniCAD related to the creation and manipulation of files.

By deploying it, see commands **New, Open, DXF Import, Save, Save As, Print, Print Preview, Page Setup, and Exit.**

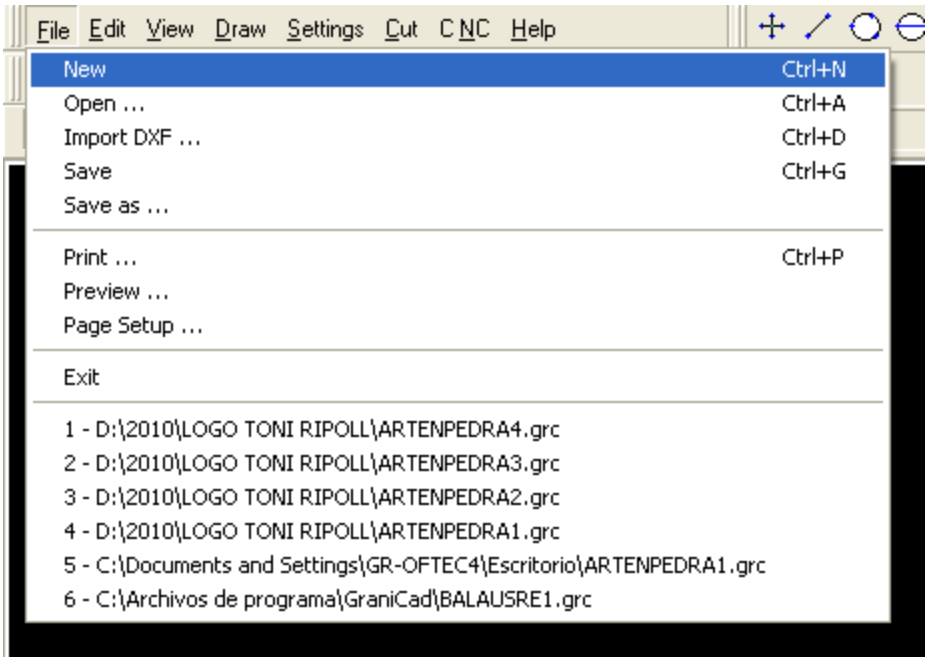

Figure 4

Selecting any of them, a dialogue appears that you will be explaining each particular case.

4.1.1. New

This command permits the creation of a new project. When executing it, will open a template of a new drawing.

Also, you can access this command with the sequence of CTRL+N keys.

If at the time that you open a new project is another project already opened (that has not been saved), the program will ask if you want to save the changes made in that earlier project.

4.1.2. Open

Opens an existing project, either in the computer's internal hard disk or any removable memory.

Running this command displays a dialogue box, where through a browser you can select the file location.

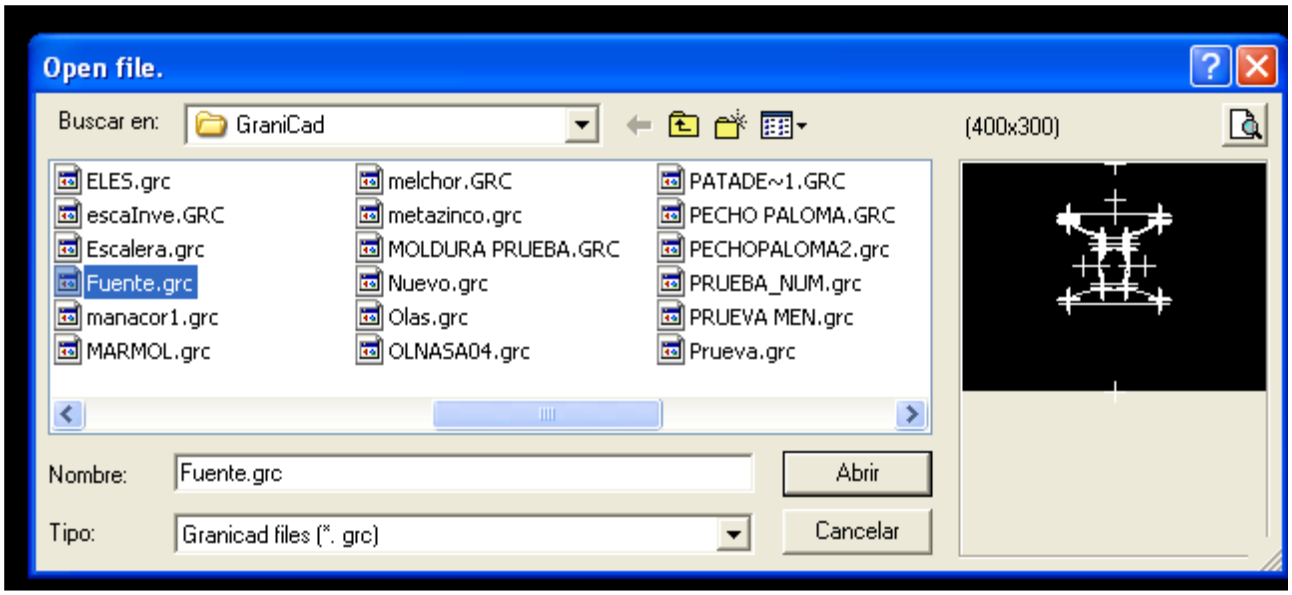

Figure 5

If the user does not remember the name of the file he is looking for, the right of the displayed dialogue box shows a preview image of the document.

It can open any type of file with. Grc extension.

You can also access this command with the key sequence CTRL + A and/or writing to the "Command Line" OPEN.

4.1.3. Import DXF

This command allows the use of files with the .dxf extension in the program. With this option, the user can import drawings created with most programs, and computer-aided design on the market.

* Always check that the imported drawing is a closed object. GraniCAD works only with closed objects and only with dots, lines and arches.

When you run "Import DXF ..." opens a window where you can search for the desired file.

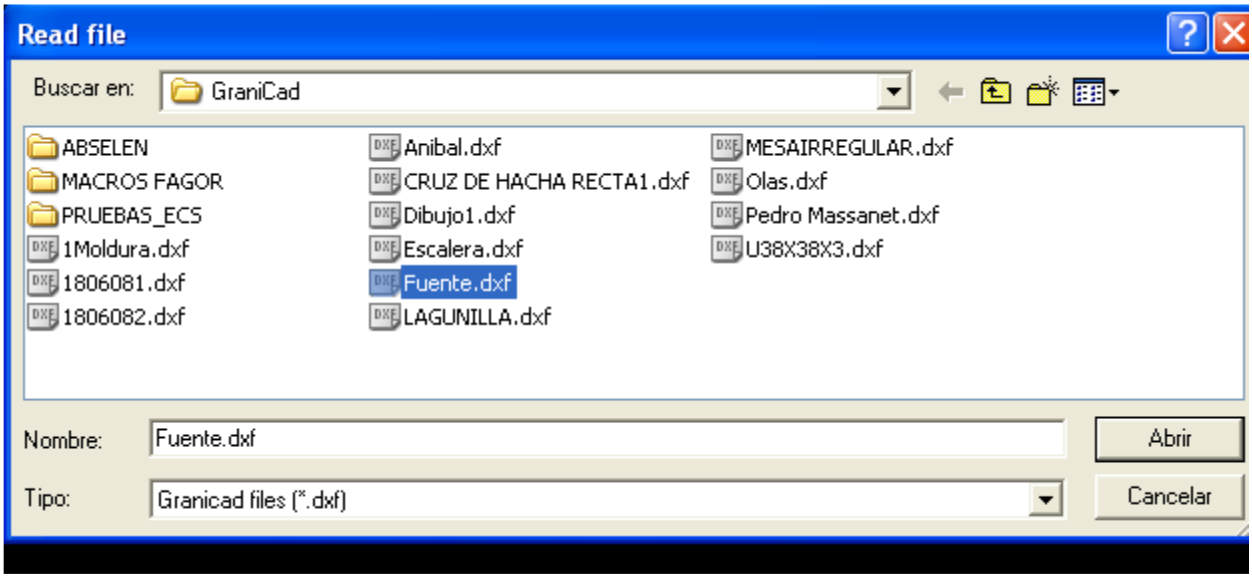

Figure 6

Also, you can access this command with the sequence of CTRL+D keys and/or by writing in the "Command Line" IMPORT.

4.1.4. Save

Allows overwrite the latest changes made to the previously saved version. If this is the first time the file is saved and had no previous name, the program will be assigned a name and location through some dialogue boxes such as the following figures.

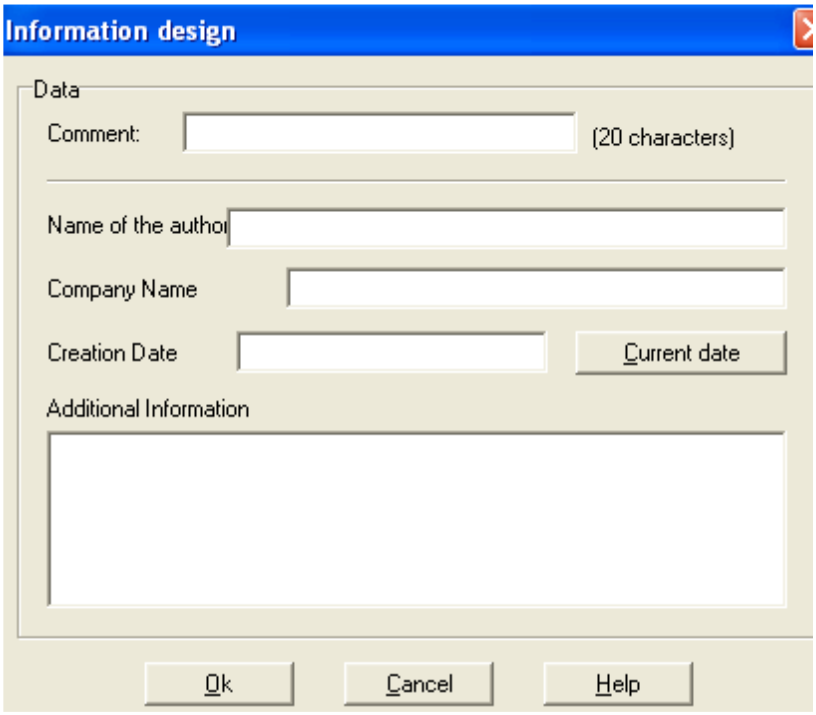

Figure 7

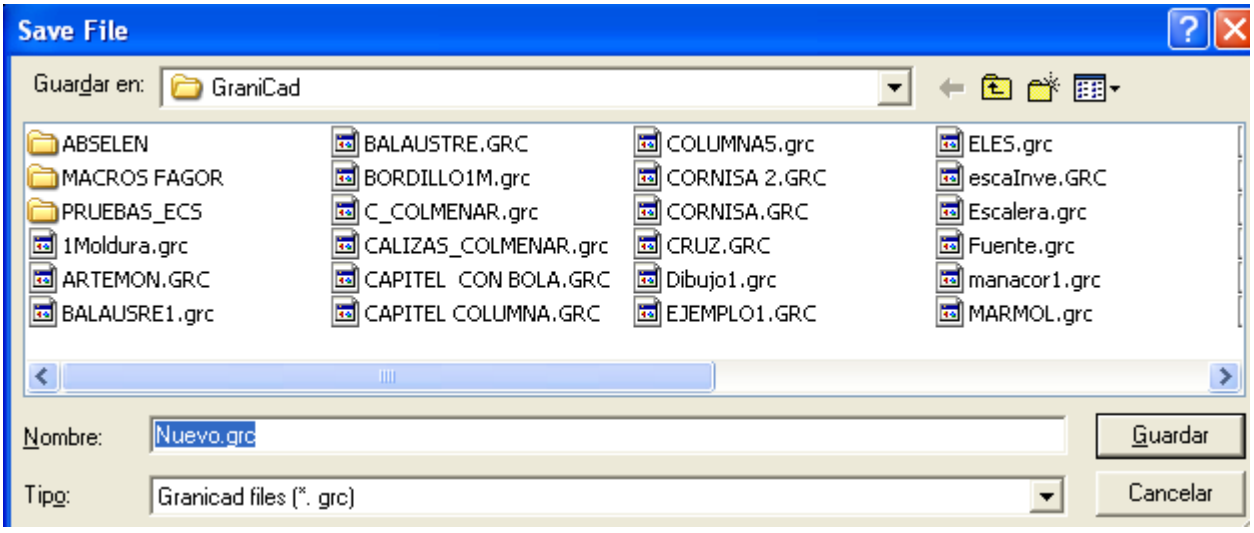

Figure 8

In the first dialogue box, you can enter the name of the figure, the author's name, the company name, the date of creation, and the comments under consideration. To edit or view any of the information in the dialogue log can be accessed from the SETTINGS menu and OPTIONS.

In the second box, the user can assign the desired name to the project that is developing or will develop and define the location of the file.

The GraniCAD files are saved with the extension. GRC

You can also access this command with the key sequence CTRL + G and/or writing to the "Command Line" GU or SAVE.

4.1.5. Save as

Allows the user to assign a name and save the project that is developing in a particular location. In the execution of this command two dialog boxes appear as explained in 2.1.4 device for the instruction store.

You can also access this command with the key sequence CTRL + G and/or writing to the "Command Line" SAVEAS.

4.1.6. Print

It allows us to obtain a paper copy of the work done, the trace cut or both. When you print your entry will also appear in the Information dialog box drawing.

In the print dialogue box, you can select the desired printer, properties, print range, and number of copies...

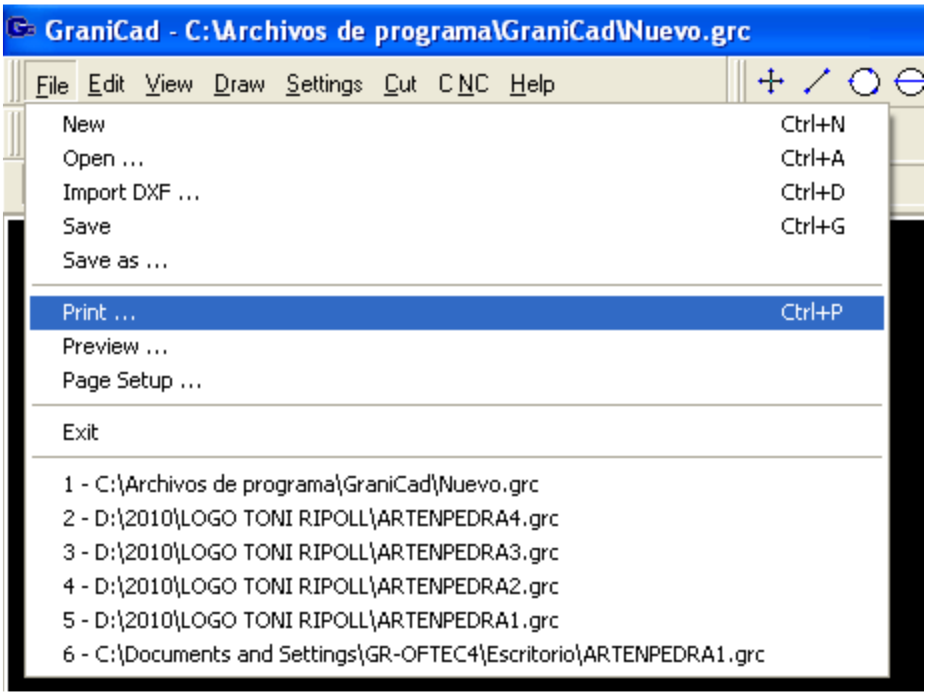

Figure 9

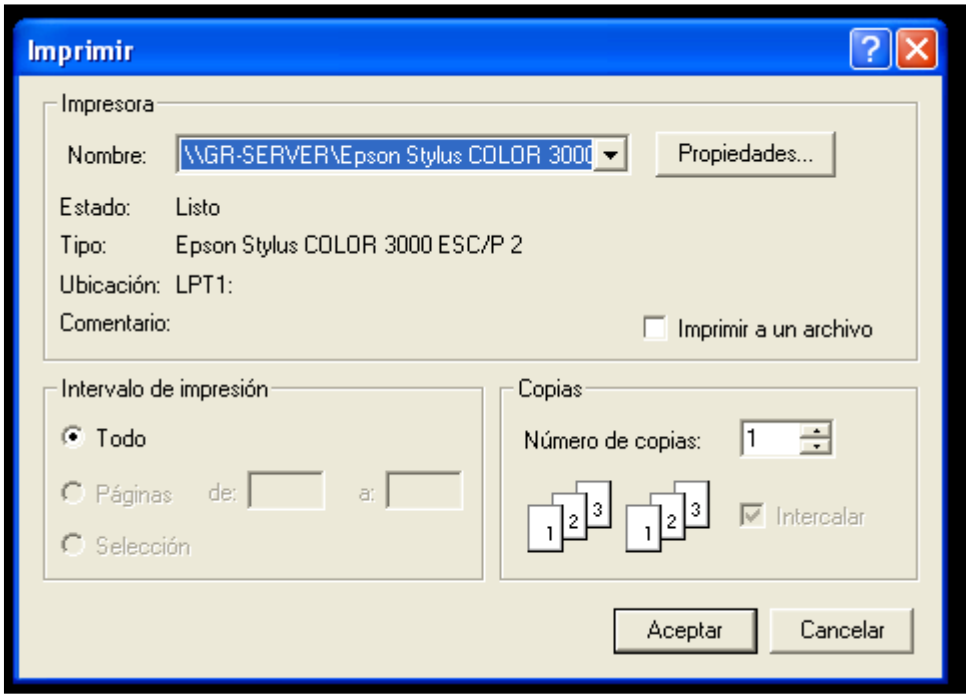

Figure 10

Also, you can access this command with the sequence of CTRL+P keys and/or by writing in the "Command line" IMP or PRINT.

4.1.7. Preview

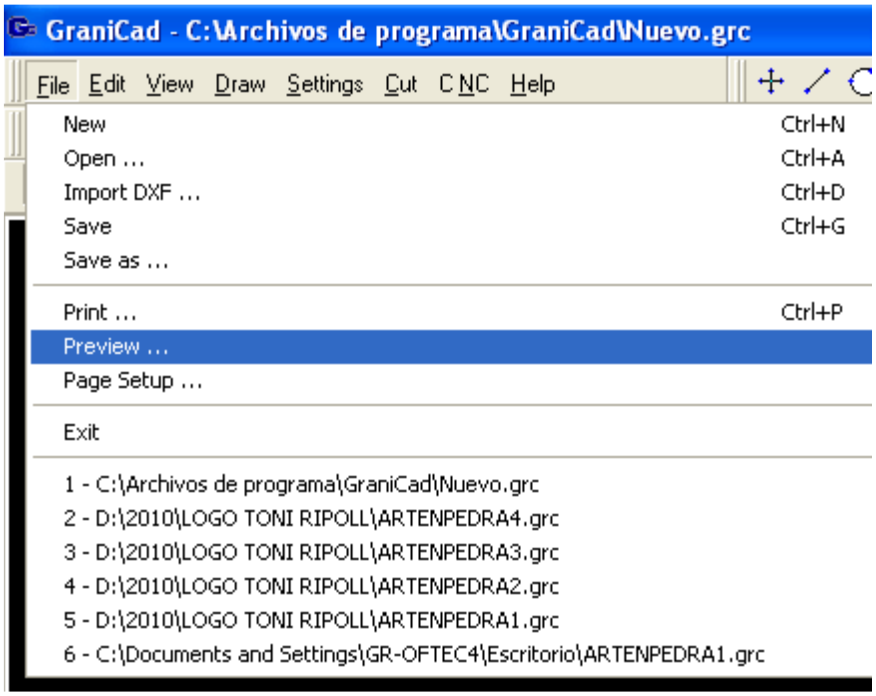

Figure 11

Running this command will see a dialogue box like the one shown below, which shows the approximate print design and the path held court specified by the user.

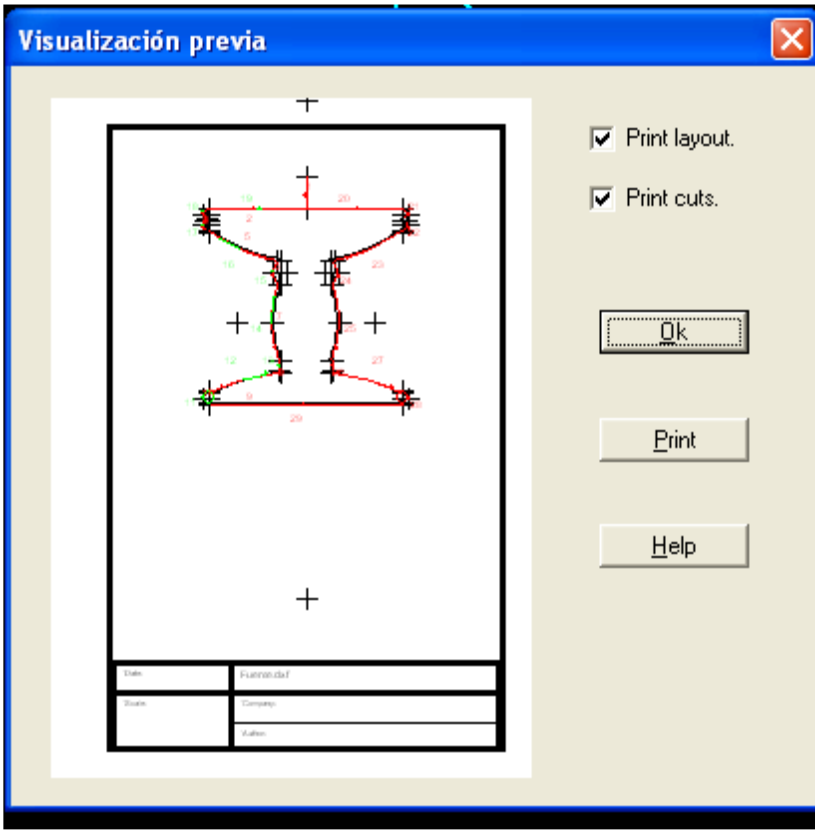

Figure 12

Cells PRINT LAYOUT AND PRINT CUTS sets how much the project will be printed. The change of state in these settings is automatically saved and will affect all designs made then, and not just the design in progress.

Pressing the OK button closes the dialogue box, but there is no printing, for this purpose you can use the PRINT button.

You can also access this command by typing in "Command Line" PREVIEW.

4.1.8. Page Setup

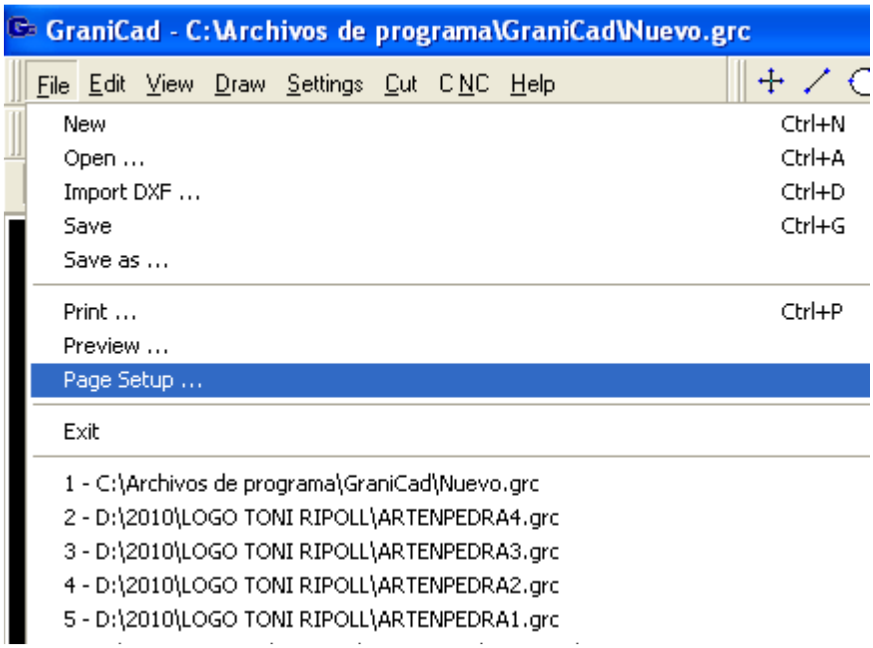

Figure 13

Selecting this option shows a dialogue box that allows us to modify the parameters relating to the printing of the project, such as selecting the printer, paper size and orientation.

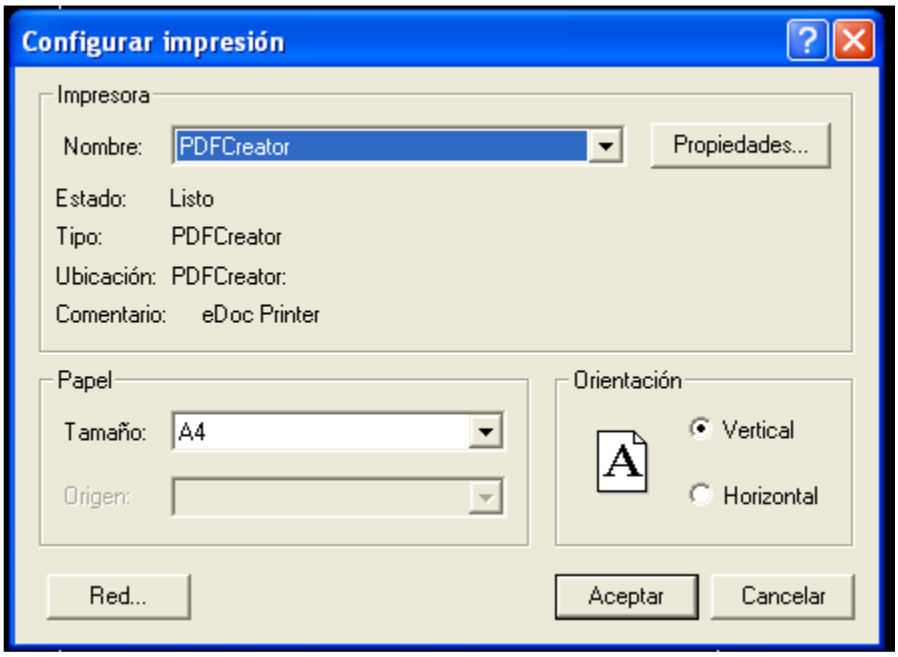

Figure 14

If you click on the Properties option gives us the ability to select the number of copies, and choose between black and white...

You can also access this command by typing in "Command Line" or CONFPAG.

4.1.9. Exit

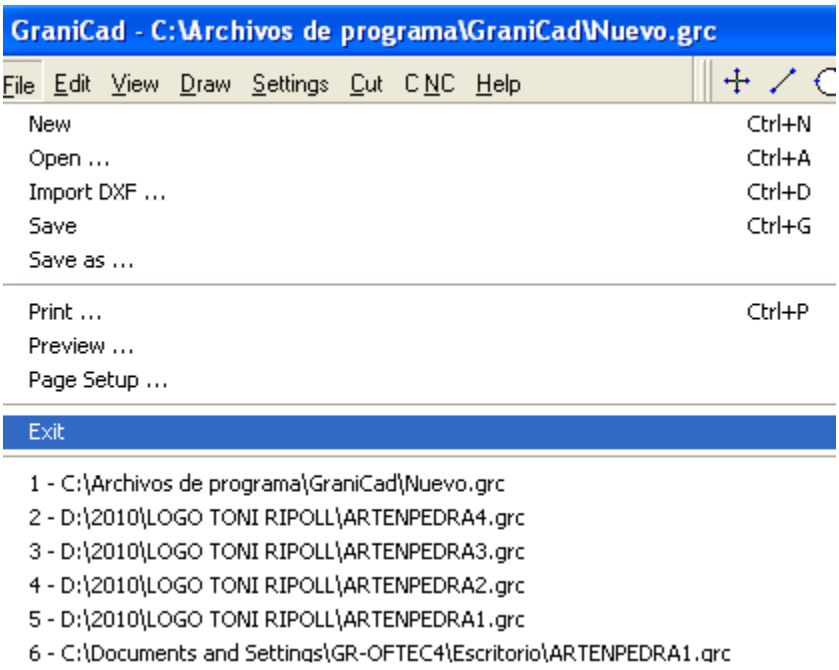

Figure 15

This command allows the user to close the program. If there is a drawing without saving, or changing, the program informs the user about this fact, allowing saving itself from the computer disk or external device memory before exiting.

You can also access this command by typing in "Command Line" FIN.

4.2. Settings Menu

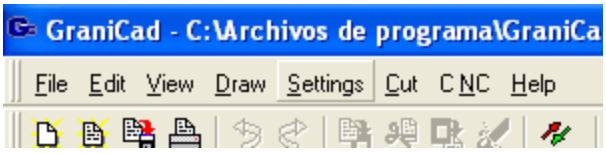

Figure 16

This menu allows the user to modify various aspects of the operation of the program, and define various parameters of work, adapting to your preferences. These variables can be modified in each individual work, or after installation of the program, preserving its value changed from one job to another.

By deploying this option are four buttons that you can customize the various parameters, which are detailed below.

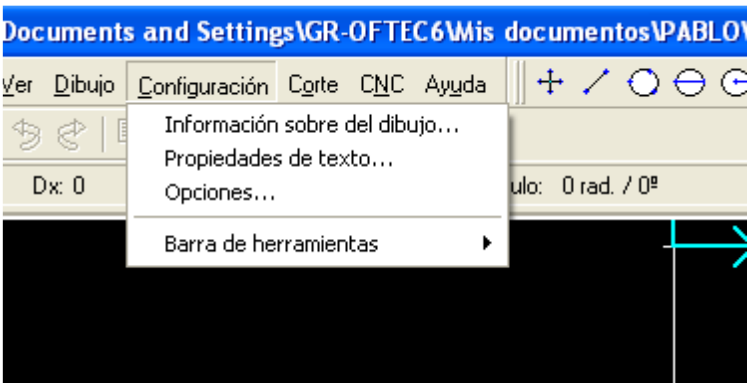

Figure 17

4.2.1. Information on the drawing

The user can enter data on the design drawing that is saved in the file generated by GraniCAD.

This same dialogue box appears when you run the Save As command (see section 2.1.4).

The different areas are detailed below:

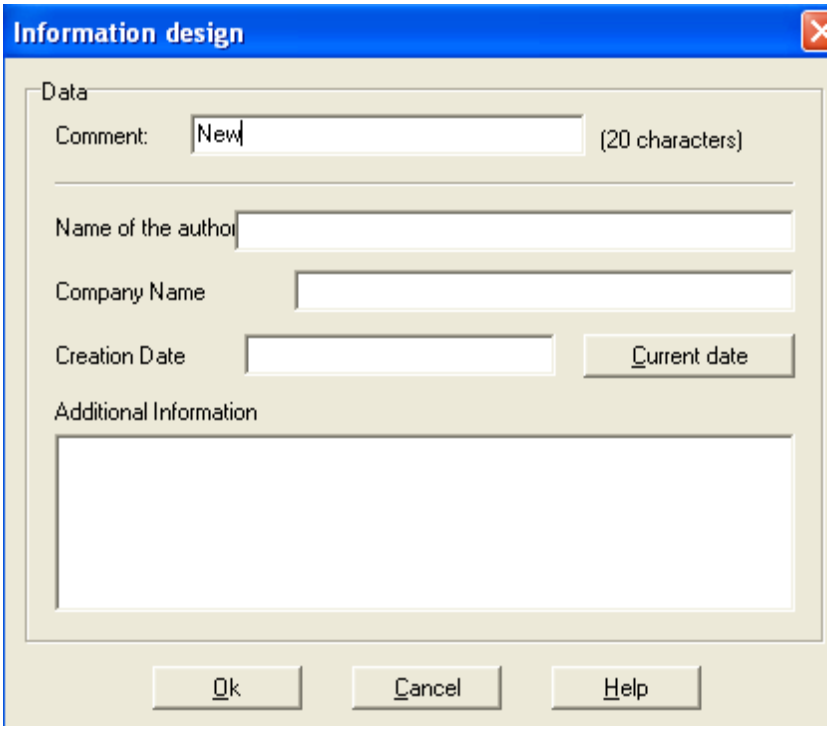

Figure 18

- **COMMENT**: You can write any comments describing the design in development or in any other that you wish to be useful for the definition of design. This comment is used when transferring programs to the CNC.
- **NAME OF AUTHOR**: write the author's name or department to which it belongs.
- **COMPANY NAME**: name of the company that is creating the design or the company that has made the task of drawing.
- **DATE OF CREATION**: You can enter the date on which initiated the creation of the drawing, the date of the order, or any other reference data to provide the user with the task of identification of the design. If this date coincides with the current date, the user can currently use the button to enter in this field the date of the current day.

* Fields Author's name and company name are stored, by default, session to session.

You can also access this command by typing in "Command Line" INFODIBUJO or INFO.

4.2.2. Properties of the text

When executing this command will appear in screen the following dialogue box.

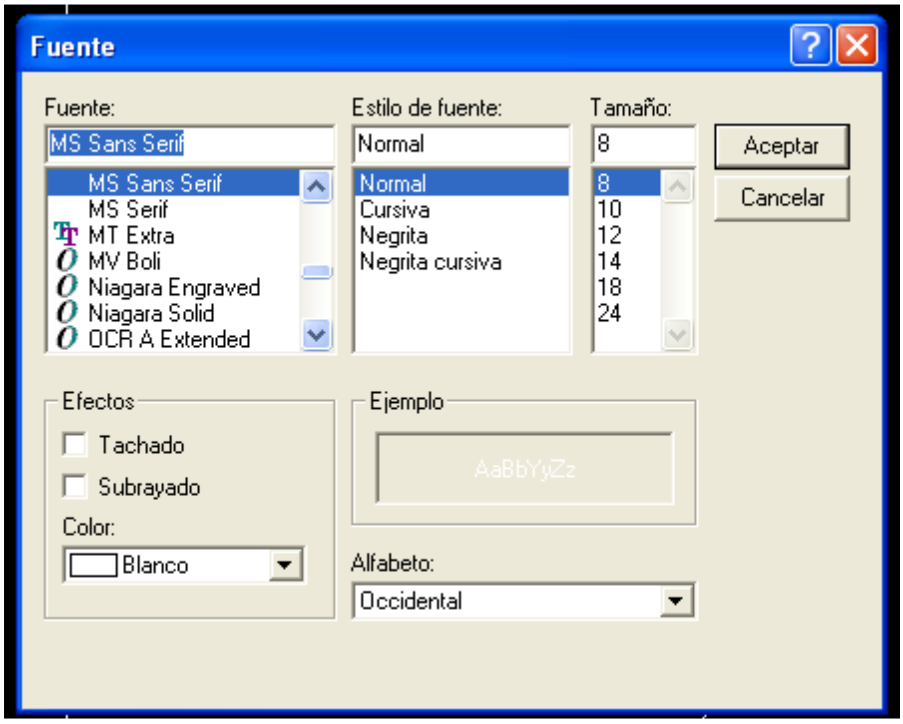

Figure 19

You can change the size, colour or font type used.

* TrueType fonts are recommended (see your name along with a symbol TT) since these are the only ones that can apply a scaling factor to maintain the same appearance both on screen and on paper when printed.

You can also access this command by typing in "Command Line" PROPIEDADESTEXTO or PROPTXT.

4.2.3. Options

When you click will appear a screen that is divided into five folders: SCALE, UNIT / GRILLE, DRAWING, GENERAL AND MACHINE.

The modification of any of these parameters has no effect until you click OK. Pressing CANCEL will not apply the changes made to the above parameters.

- SCALE: In this option, two parameters can be modified.

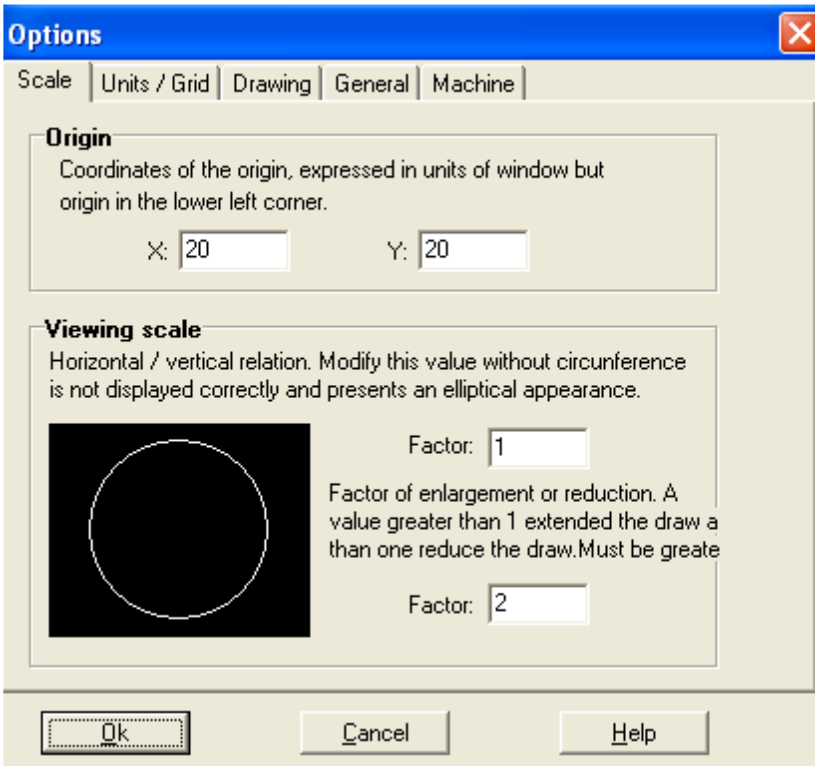

Figure 20

- Origin: You can modify the position by default of the source of global coordinates. This aspect is practical when a drawing requires negative coordinates for your definition. By scrolling the source is been able to work in a more comfortable way using exclusively coordinate positives.

- Viewing scale: It can modify the factor that relates the horizontal coordinate with the vertical circle. Also, you can modify the factor of enlargement or reduction.

- UNITS/GRID: It has three parameters to modify.

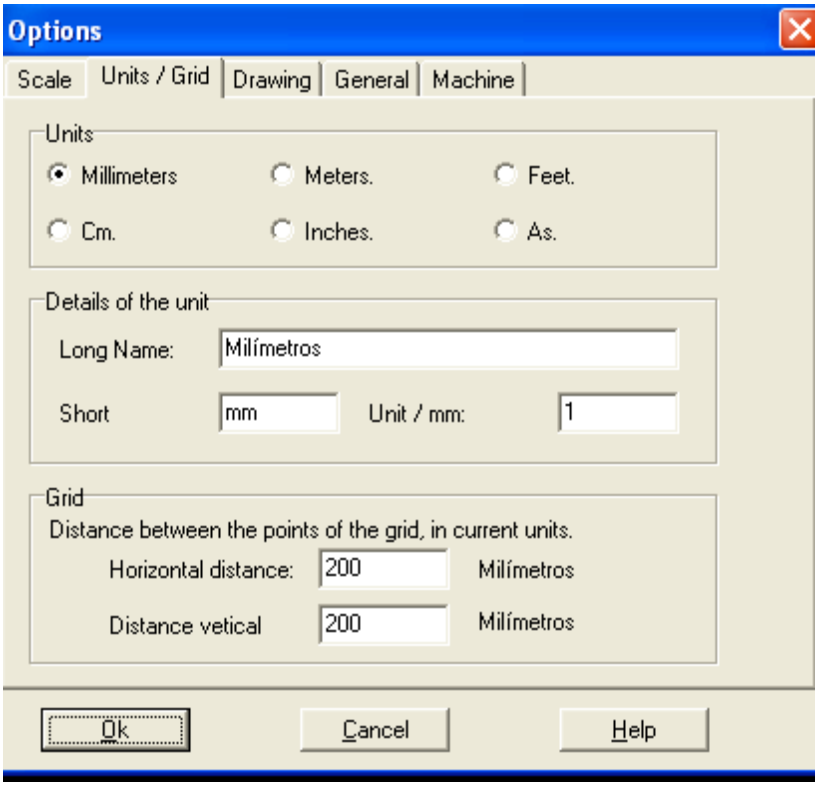

Figure 21

o Details of the unit: can you define the long name, short name and the number of millimetres that contains a unit.

- Grid: GraniCAD presents the user with the option of defining a grid reference that can serve as a geometric guide when developing the design required. It is necessary to define the horizontal and vertical distance between the grid points of reference, always expressed in the units of measure selected.

- Units: Select the unit of measure you are using. * If you use a unit other than those available, you will mark the option "As." If you select this option allows you to change the parameters that appear in the "Details of the unit."

- DRAWING: There are three main parameters to be modified.

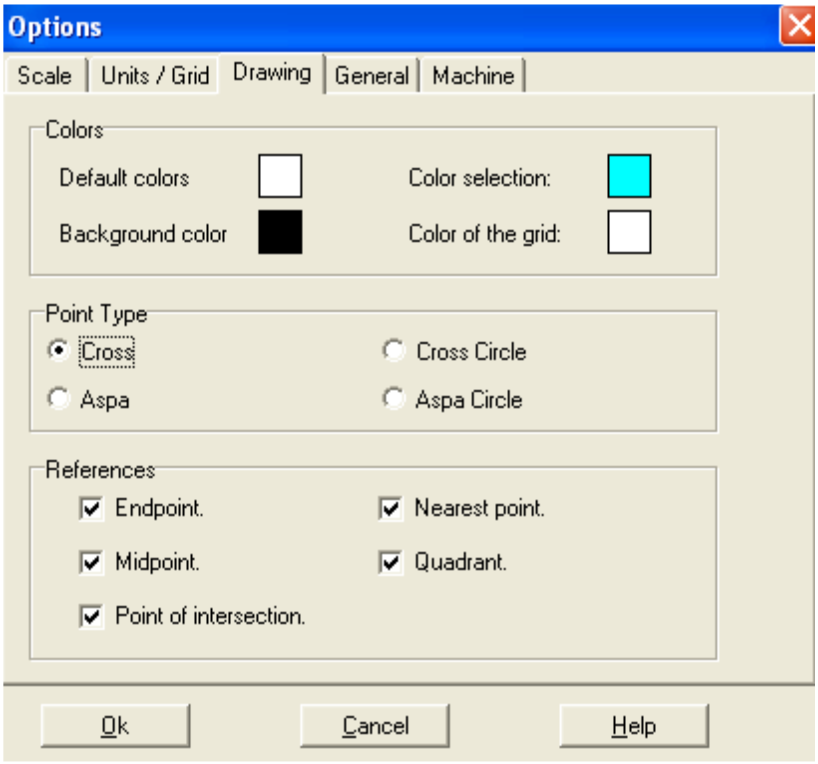

Figure 22

- Colours: you can change the background colour of the grid, which will colour the selected objects and the default colour. To change colour has to click on the box and choose a colour that you deploy.

- Type point: You can choose the shape of the points of the drawing. Changing the type of point affects all existing points.

- Reference: A tracking tool and reference to key points of objects to help draw quickly and accurately. As a tracking or reference you can activate the following key points: Endpoint (select an endpoint of a line, arc ... when you put the mouse close to it. The cursor shows as a square), Midpoint (select a midpoint of a line, arc ... when you put the mouse close to it. The cursor is shown as a triangle), point of intersection (select a point of intersection between straight lines, arcs ... when you place the mouse near the intersection. The cursor is displayed as an X), the nearest point (select a point when you place the mouse near it. The cursor is shown as a double triangle attached at one end) and Quadrant (The cursor appears as a quarter circle). Using these references can be

drawn more accurately without the need to specify the exact coordinates of these key points and do not make complicated calculations.

- GENERAL: There are three main parameters to be modified.

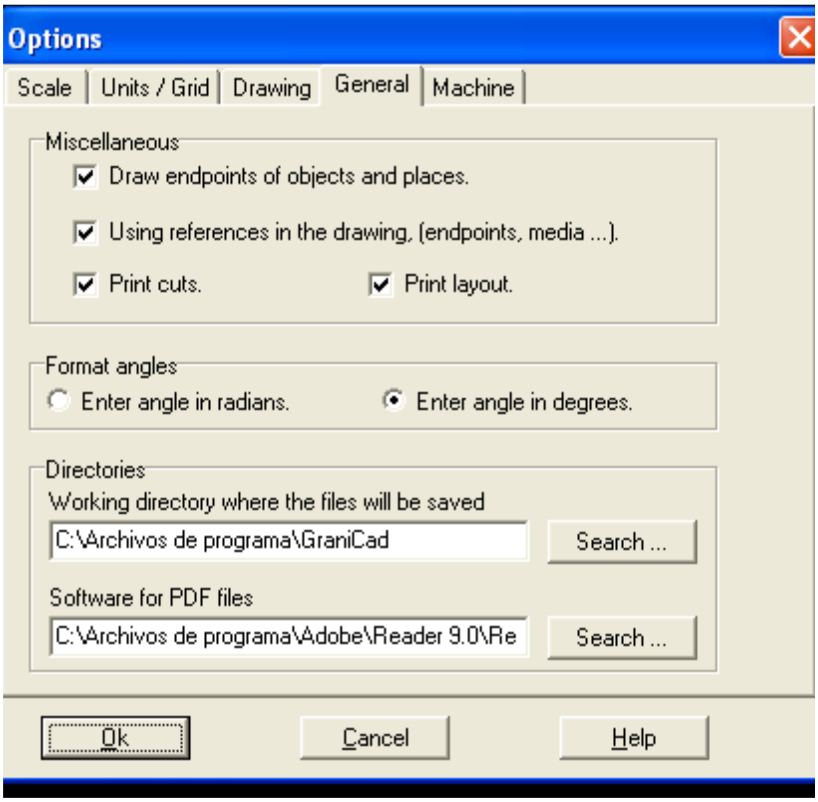

Figure 23

  - Miscellaneous: you can select whether to draw points at the ends of lines, squares or centres of arcs and circles if you are to use the reference tool while you draw if you print the cutting path or if you print the made design.
  - Format of angles: Choose the angle unit.
  - Directories: select the working directory where you saved projects.

- MACHINE: There are three main parameters to be modified.

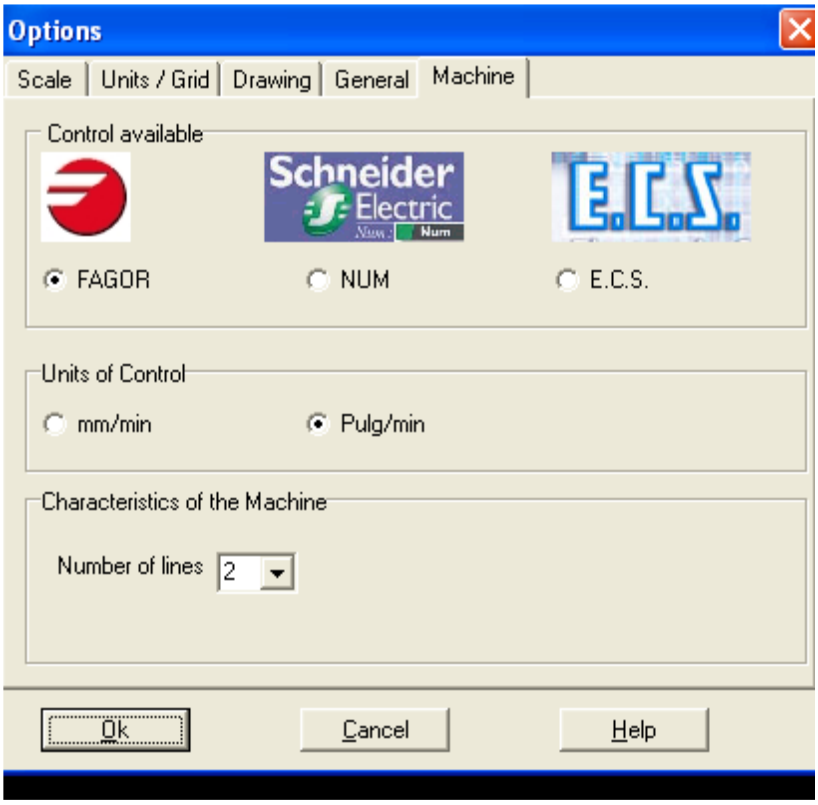

Figure 24

- Available control: it is selected the numerical control that is installed in our machine.
- Units of control: you can choose the work units.
- Characteristics of the machine: fixed the number of axis of our machine.

4.2.4. Toolbars

The GraniCAD program has three toolbars positioned with the shortcut icons to the main commands:

- Toolbar FILE.
- Toolbar ENTITIES.
- Toolbar VIEW.

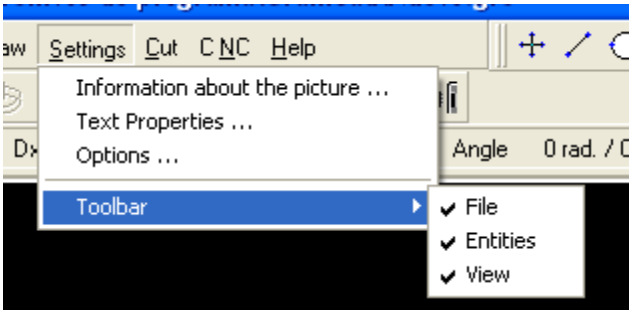

Figure 25

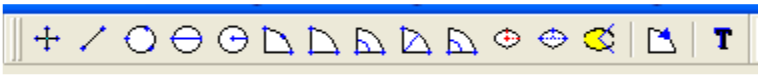

Figure 26: Toolbar ENTITIES

Figure 27: Toolbar FILE

Figure 28: Toolbar VIEW

IN NEXT CHAPTERS WILL DEFINE EACH OF THE FAST-ACCESS ICONS THAT CONTAIN EVERY ONE OF THEM.

4.3. Draw Menu

Deploying this menu is a series of commands that can be considered in two groups: drawing commands and commands to modification.

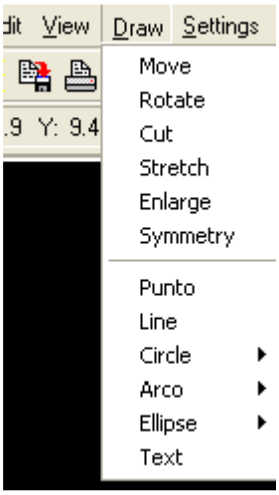

Figure 29

The first group of commands is those of modification and the second is those of drawing.

4.3.1. Dot

When you run this command, draw a dot on the screen. The point position is defined by its coordinates X and Y.

When you run this command you can fix it, by indicating its position with the mouse or by typing in the line of text where you want to put the point. If for example, you want to draw a point X0 Y0 write the position in the text line 0.0.

The shortcut icon to the command POINT is located on the bar Entities and is shown in the figure below.

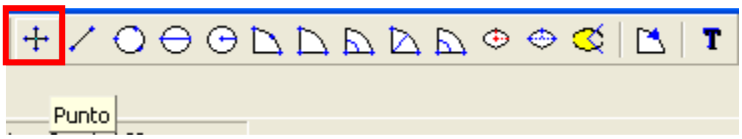

Figure 30

Also, you can access this command by writing in the “Command line” PUN or PUNTO.

4.3.2. Line

The execution of the command "line", is drawn on the screen in a straight line between two points. The introduction of the coordinates of the points can be made by absolute coordinates, relative (type @ in front of the X and Y coordinate) or by selecting a point of reference for an existing entity.

There is the possibility of creating a series of contiguous line segments so that after writing the second point of the first segment, it will act as the first point of the second segment. This will facilitate the implementation of connected straight. To end the operation press ESC, ENTER or the right mouse button.

The shortcut icon to the command POINT is located on the bar Entities and is shown in the figure below.

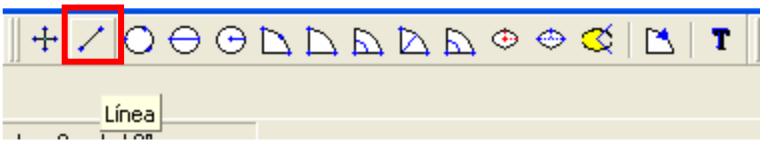

Figure 31

Also, you can access this command by writing in the "Command line" LIN or LINEA.

When you run this command can position the line, by indicating its position with the mouse or typing the text where you want to place the start and end points of the lines.

If for example, you draw a horizontal line 100mm in length originating from X0 Y0 next steps are:

1. Click the line icon (or type in the command line or LINE LIN);
2. The program asks that you indicate the starting point;
3. Write in the command line and press ENTER 0.0.
4. Then the program prompts you to indicate the end of the line, type 100.0 and press ENTER.
5. As you do not draw any other line will press ESC to end this command. If you draw more lines would write the following.

4.3.3. Circle

The program GraniCAD gives the user the ability to draw circles defined in three ways:

- Circle passing through three points (Circle 3P).
- A circle that passes through two points (Circle 2P).
- Circle defined by its centre and radius (Circle CR).

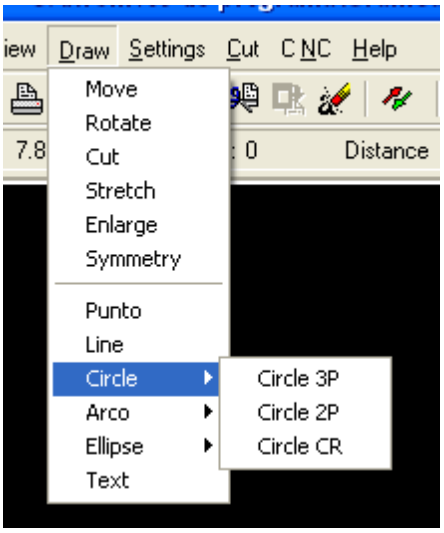

Figure 32

4.3.3.1. Circle 3P

Define a circle by introducing three unaligned points through which it passes. The shortcut icon is shown in the figure below:

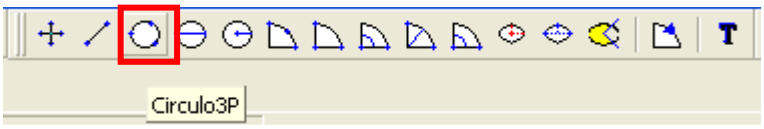

Figure 33

The text-based command to access this statement is CIRC, CIRCULOS, CIRC3P or CÍRCULO3P.

When you click on this icon, the program asks that you indicate the first point through which passes the circle, you introduce that item and press ENTER. Then ask us you indicate the second point through which it passes, introduce the value and press ENTER. The same with the third point. You can also introduce the crossing points with the mouse.

4.3.3.2. Circle 2P

Define a circle by introducing two points that define the diameter of the circle. The circle thus defined has the centre at the midpoint of the line joining the two points entered.

The shortcut icon is shown in the figure below:

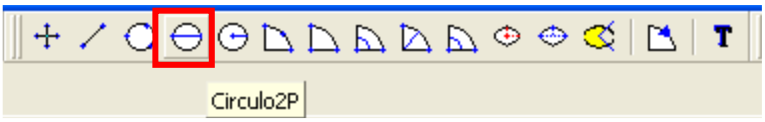

Figure 34

The command in text mode access to this instruction is CIRC2P or CÍRCULO2P.

When you click on this icon, the program asks that you indicate the first point through which passes the circle, you introduce that item and press ENTER. Then ask us you indicate the second point through which it passes, introduce the value and press ENTER. You can also introduce the crossing points with the mouse.

4.3.3.3. Circle CR

You can draw a circle defining the centre and the length of the radio.

The shortcut icon is shown in the figure below:

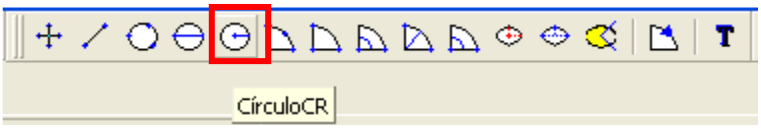

Figure 35

The command in text mode access to this instruction is CIRCCR or CÍRCULOCR.

When you click on this icon, the program asks that you indicate the centre of the circle, you introduce that item and press ENTER. Then ask us you indicate the radius, you introduce the value and press ENTER. You can also introduce them to the mouse.

4.3.4. Arc

The program gives us the opportunity to draw arcs of circumference defined in five different ways.

1. Arco passing through three points not aligned (Arco 3P).
2. Arc is defined by its starting point, its endpoint and the centre of the circle. (Arco IFC).
3. Arc is defined by its starting point, its endpoint and the angle that runs between them. (Arco ICA).
4. Arc is defined by its starting point, its endpoint and the radius of the circle. (Arco IFR).
5. Arc is defined by its starting point, its endpoint and the angle you travel. (Arco IFA)

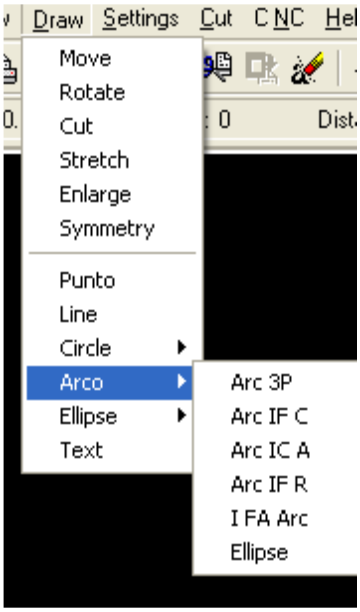

Figure 36

4.3.4.1. Arc 3P

Defines a circular arc passing through three points not aligned. The sense of establishing the arc is given by the order in which points are created.

The shortcut icon is shown in the figure below:

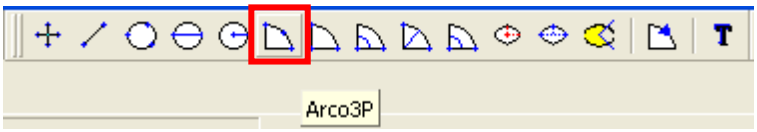

Figure 37

The command in text mode access to this instruction is AR, ARCOS, ARC3P or ARCO3P.

When you click on this icon, the program asks that you indicate the first point through which passes the arc, you introduce the point (for example -540, -500) and press ENTER. Then ask us you indicate the crossing point, enter the value and press ENTER (for example -575, -415).

Finally, you asked the endpoint, to enter the value and press ENTER (for example -650, -630). You can also introduce the crossing points with the mouse.

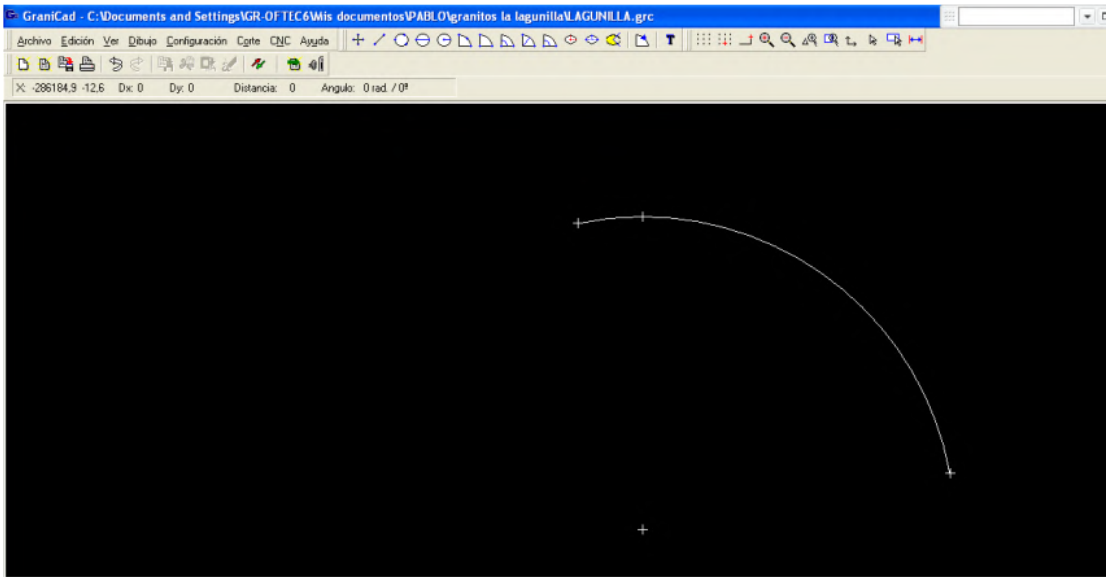

Figure 38

4.3.4.2. Arc IFC

Defines a circular arc through the introduction of the start and end points of the arc and centre. The sense in establishing the arc is given by the state of the Sense of the arc button located on the toolbar Entities, which is explained at the end of this point.

The shortcut icon is shown in the figure below:

Figure 39

The command in text mode access to this instruction is ARCIFC or ARCOIFC.

The program will ask, in this order the following: start point, endpoint and centre. You introduce these data in the text line by typing each item you INTRO. You can also introduce the crossing points with the mouse.

4.3.4.3. Arco ICA

Defines a circular arc through the introduction of the starting point and centre of the arc and the angle you travel. The sense in establishing the arc is given by the state of the Sense of the arc button located on the toolbar Entities, which is explained at the end of this point.

The shortcut icon is shown in the figure below:

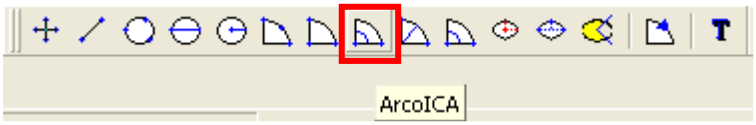

Figure 40

The command in text mode access to this instruction is ARCIC or ARCOICA.

The program will ask, in this order the following: starting point, center and angle. You introduce these data in the text line by typing each item you INTRO. You can also introduce the crossing points with the mouse.

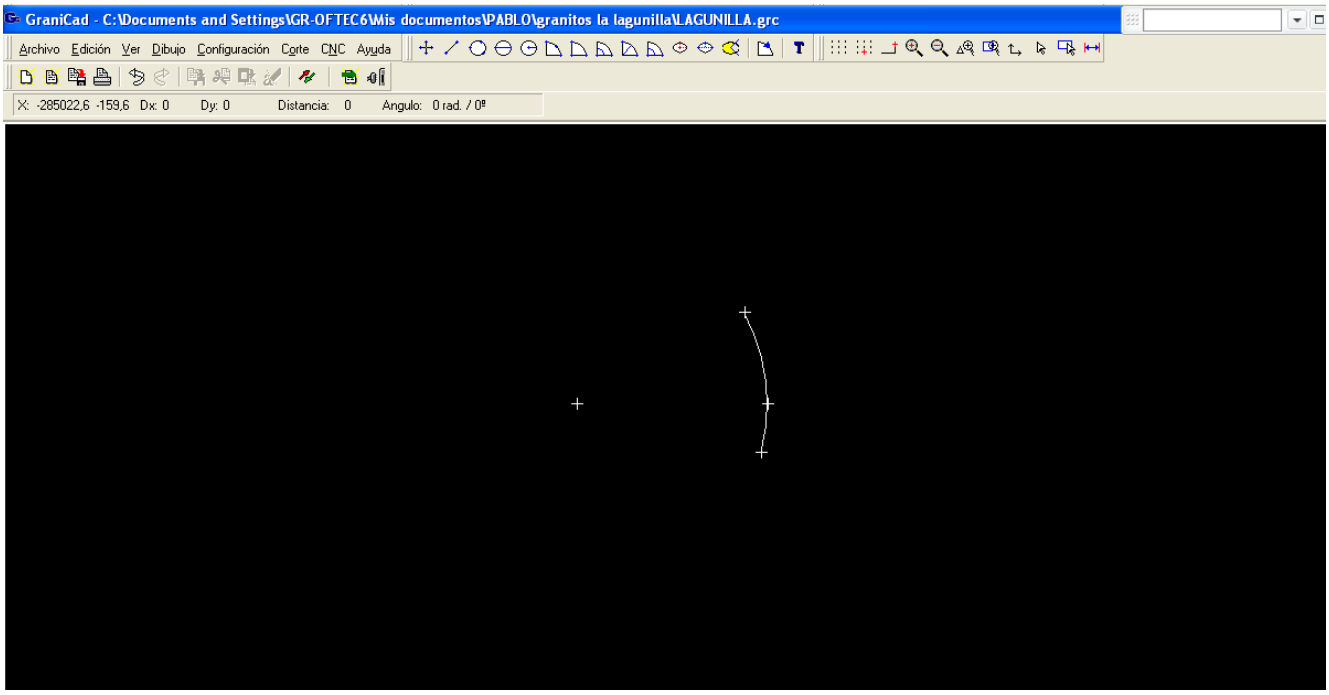

Figure 41

4.3.4.4. Arco IFR

Defines a circular arc through the introduction of the start and end points of the arc and radius. The sense in establishing the arc is given by the state of the Sense of the arc button located on the toolbar Entities, which is explained at the end of this point.

The shortcut icon is shown in the figure below:

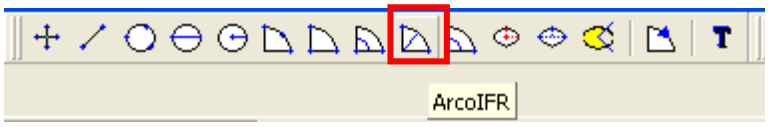

Figure 42

The command in text mode access to this instruction is ARCIFR or ARCOIFR.

The program will ask, in this order the following: start point, end point and radio. You introduce these data in the text line by typing each one you INTRO. You can also introduce them to the mouse. When the bow is drawn using the mouse, the centre is chosen that is closest to the mouse position at that time.

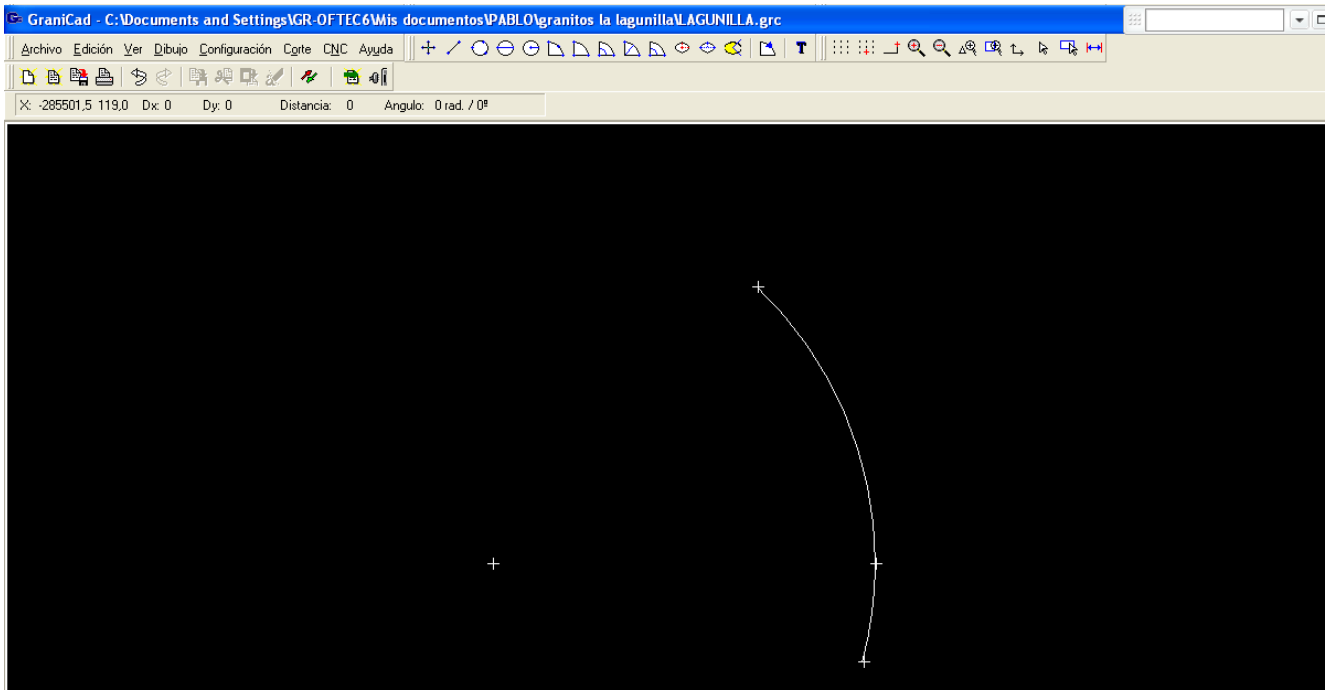

Figure 43

4.3.4.5. Arco IFA

Defines a circular arc through the introduction of the start and end points of the arc and the angle described. The sense in establishing the arc is given by the state of the Sense of the arc button located on the toolbar Entities, which is explained at the end of this point.

The shortcut icon is shown in the figure below:

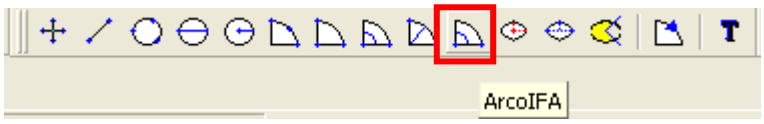

Figure 44

The command in text mode access to this instruction is ARCIFA or ARCOIFA.

The program will ask, in this order the following: start point, end point and angle. You introduce these data in the text line by typing each one you INTRO. You can also introduce them with the mouse, but the angle can only be made through the text line.

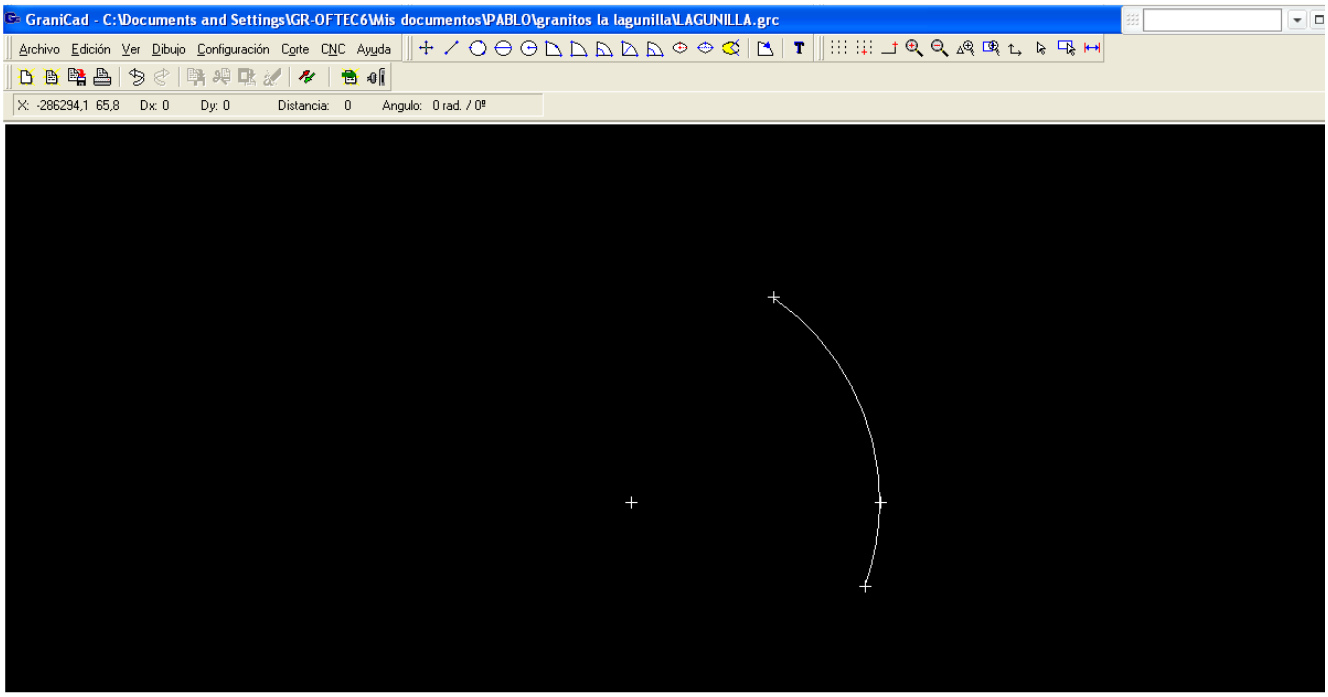

Figure 45

Determination of the sense of turn

It permits defining in which sense will generate the previously explained arcs.

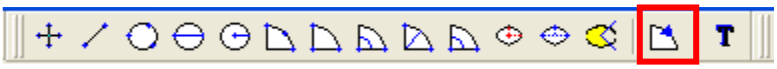

Figure 46

If this button finds to him selected, the sense of creation of the arcs will be the |anticlockwise|, while in the case of is not enabled the sense by default will be |clockwise|.

4.3.5. Ellipse

The program allows us to create defined ellipses in two different manners:

- By defining the centre and the axis;
- Defined the position of the axis.

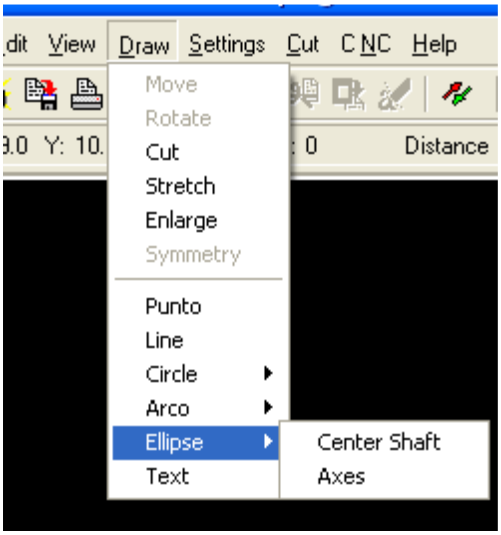

Figure 47

4.3.5.1. Centre and Axis

Allows represent an ellipse approximated by arcs of circumference. The entire ellipse is approximated by eight arches. The input data are the focus of the ellipse, which is introduced by a point, the end of one of the axes also introduced by a point and the length of the other axis.

The program will ask, in this order the following: the ellipse centre, end of an axis and end of the other semi. You introduce these data in the text line by typing each one you INTRO. You can also introduce them to the mouse.

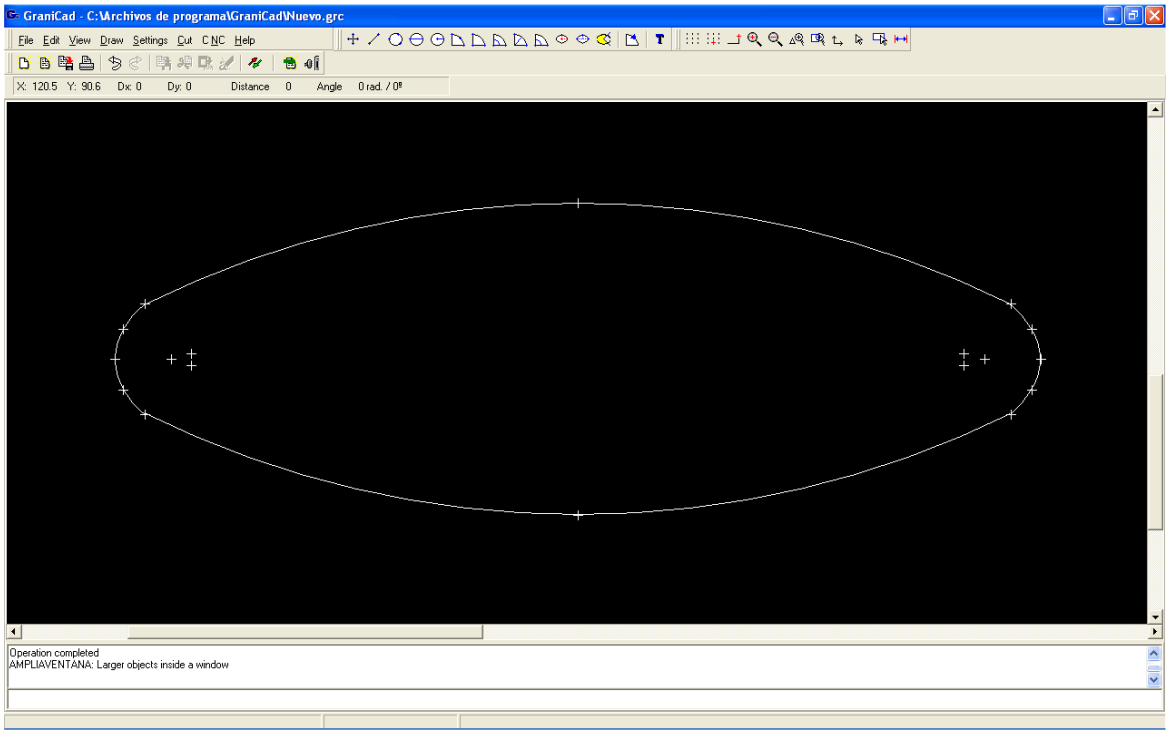

Figure 48

- Axes: allows represent an ellipse approximated by arcs of circumference. The entire ellipse is approximated by eight arches. The input data are the ends of one axis of the ellipse, which are introduced by points and the length of the r-axis.

  The program will ask, in this order the following: first point of the shaft end, shaft end point and length of the other semi. You introduce these data in the text line by typing each one you INTRO. You can also introduce them to the mouse.

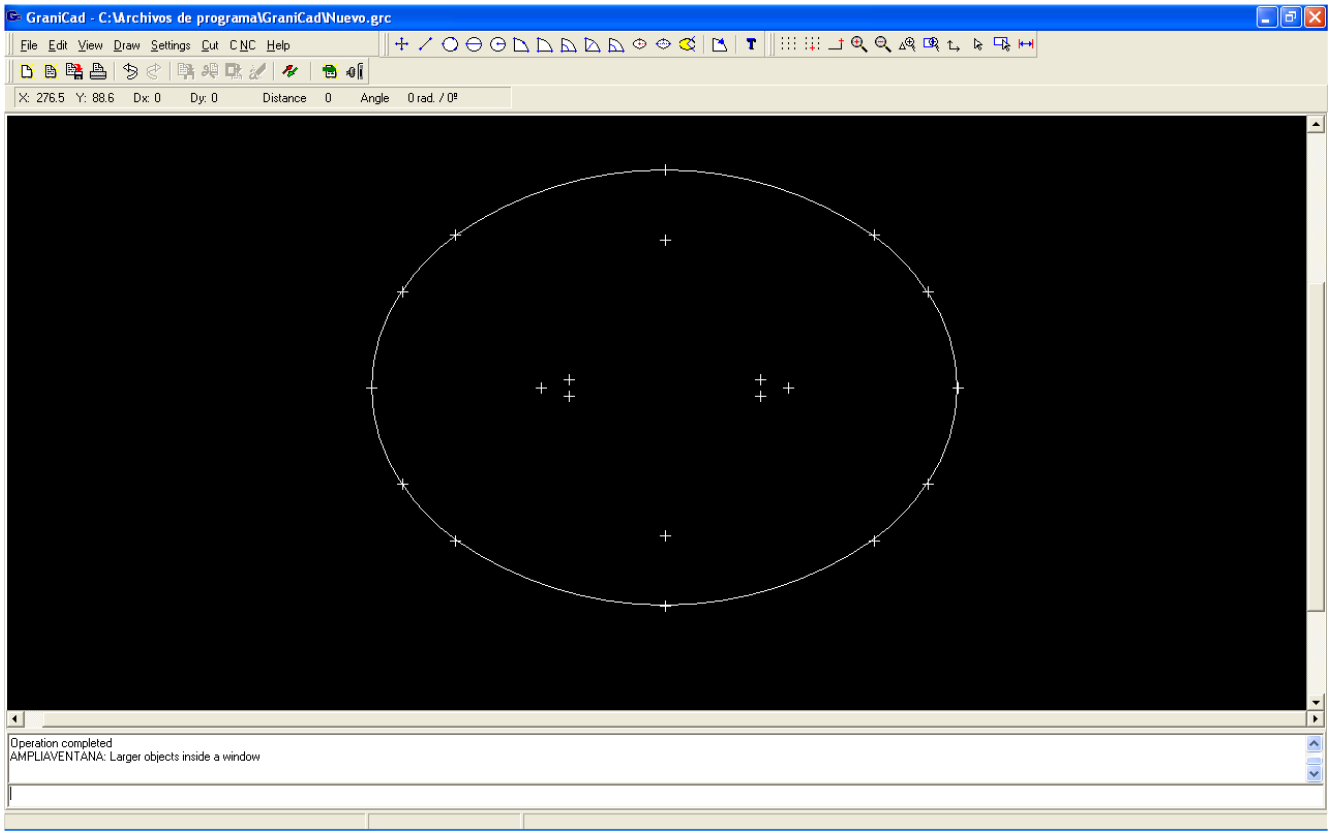

Figure 49

- ELLIPSE ARC: allows represent an arc of ellipse approximated by arcs of circumference. Initially, data is entered into the entire ellipse: the centre, and a shaft length of another axis. Then you introduce the initial and final angles, both measured from the centre.

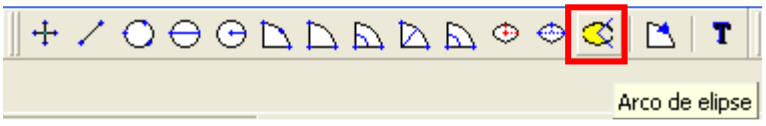

Figure 50

4.3.6. Text

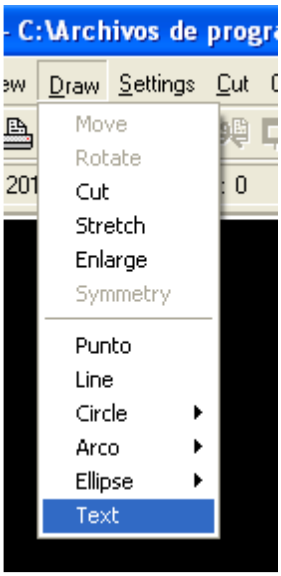

Figure 51

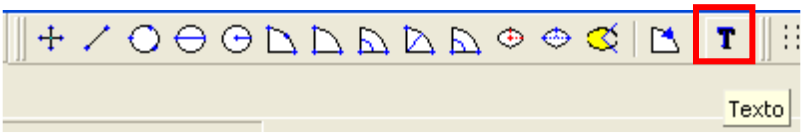

Figure 52

By this order, you can enter a message or clarification in the drawing. To indicate the position of the text is necessary to make a point.

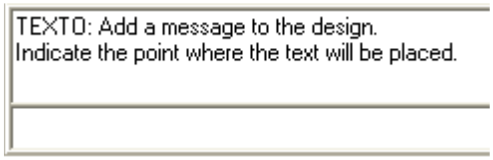

Figure 53

This will be in the top left corner of the text box. After defining this point, the content of the text box is written to the command line.

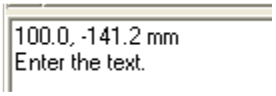

Figure 54

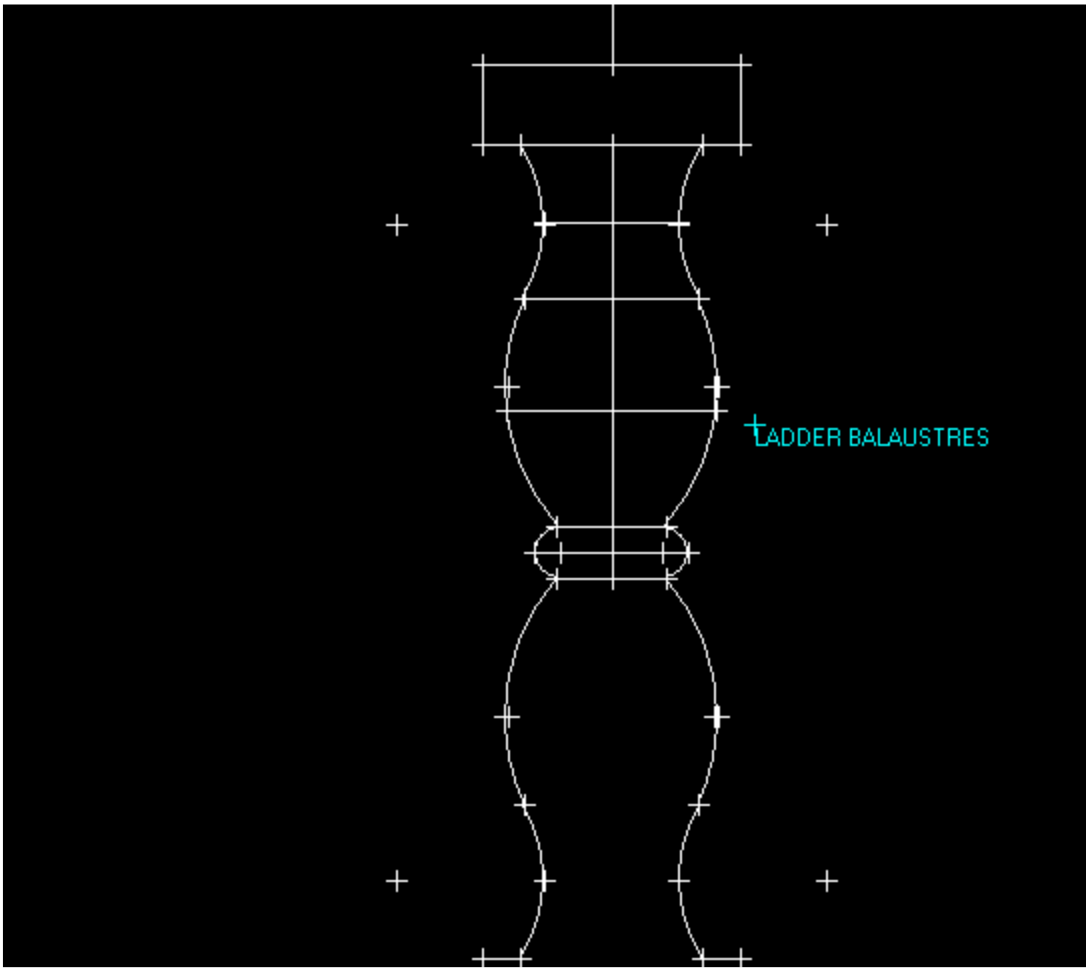

Figure 55

This command can also be done by typing in the text bar TEXT or TEX.

4.3.7. Move

☒Implementation of the MOVE command gives the user the option to move previously selected objects without changing their orientation or size and can do so with precision using coordinate values and reference objects. When you run the command, the program requires the introduction of two points: one point serving as a source or basis of displacement and a second point which serves as the end point of displacement. The first point to introduce the basis of displacement will move us to the coordinates that you introduced as the end point of displacement.

Example: Move the part of the figure below, so that the base point you take him to the point 100.100.

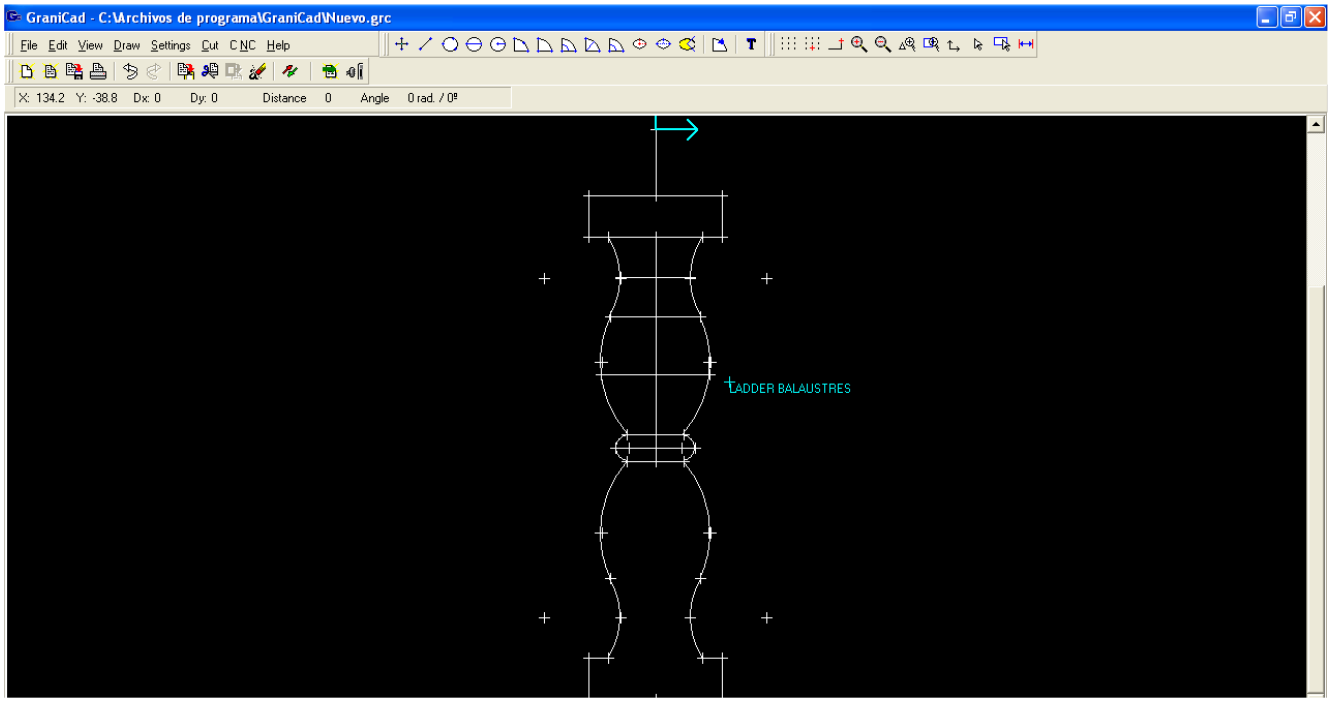

Figure 56

You click Edit and select ALL (or the key sequence CTRL + E) and then DRAW, MOVE.

The program will be asked to indicate the origin point.

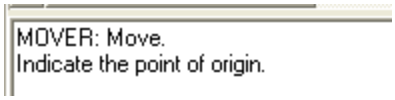

Figure 57

And select with the mouse our base point. Next, you write our destination point.

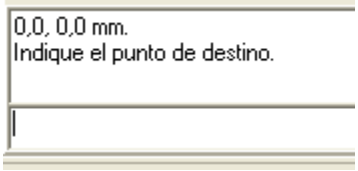

Figure 58

And typing our destination 100.100 and press ENTER. Granic will have moved our picture, based on the origin point you select, to position 100.100.

This command can also be done by typing in the text bar or MOVE MOV.

4.3.8. Rotate

Allows the user the option to turn previously selected objects without changing their orientation or size, and can do so with precision using coordinate values and reference objects. When you run the command, the program requires the introduction of two parameters: the rotation centre point (around which to make the turn) and the angle rotated. The latter can be assigned by typing in the command line or through the signing of two points, the initial and final, which together with the rotation centre defines the required angle.

Example: Turn the piece of a figure below 45 ˚ angle taking a center of rotation indicated.

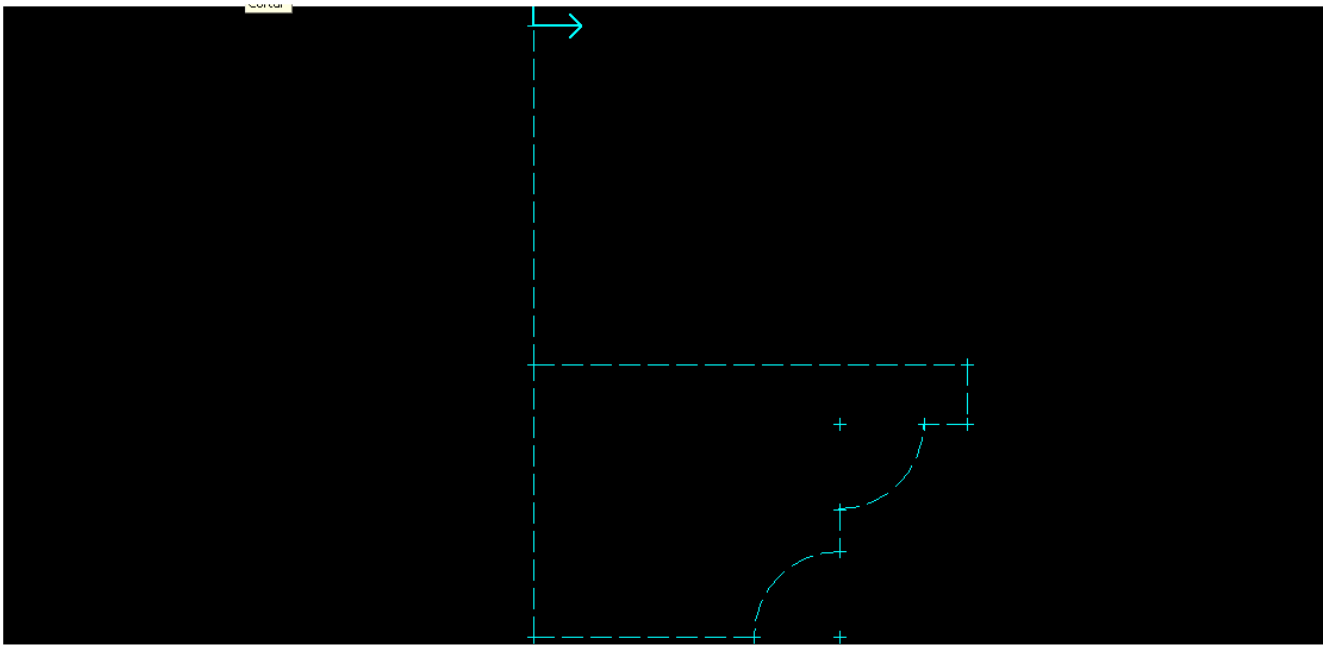

Figure 59

You click Edit and select ALL (or the key sequence CTRL + E) and then DRAW, ROTATE.

The program will ask you to indicate which is in the centre of rotation:

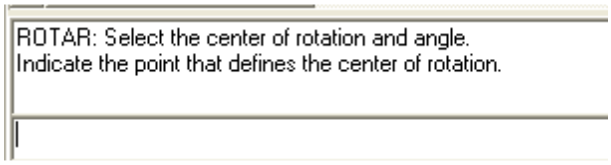

Figure 60

You indicate our point with the mouse, or you could write the coordinates of that point if you know them. Now you are asked to indicate the point where the angle starts to rotate, which in this case is the same as the origin of rotation.

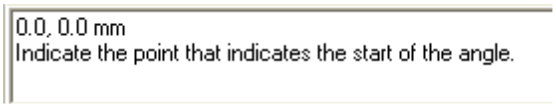

Figure 61

Then you have to indicate the angle to rotate:

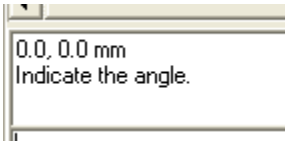

Figure 62

Write 45, or you can indicate if you know two points which are 45 each.

The figure has been rotated as wanted:

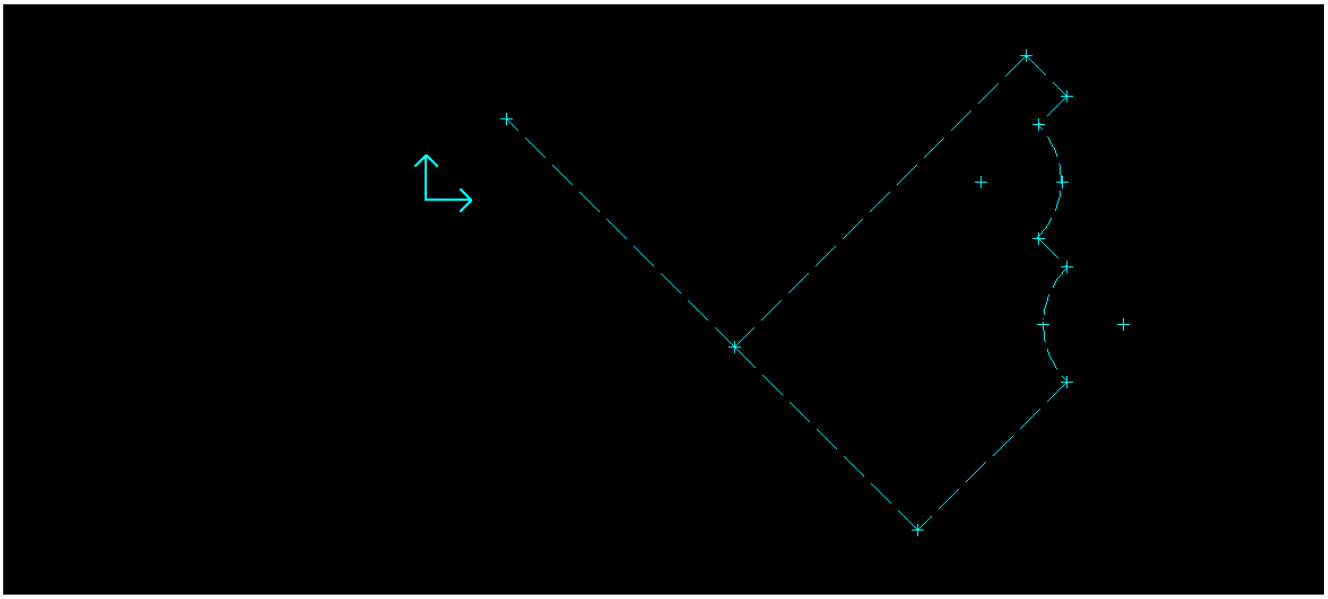

Figure 63

This command can also be done by typing in the text bar ROT or ROTATE.

4.3.9. Cut

You can divide an object into two or three parts. The division points must belong to the initial object.

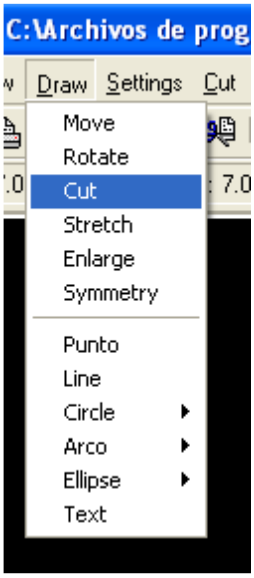

Figure 64

When you run the command from the program always requires the introduction of two breakpoints. If you want to divide an object into two parts, one must show in the same spot twice.

Example: You want the line indicated in the drawing in two equal parts.

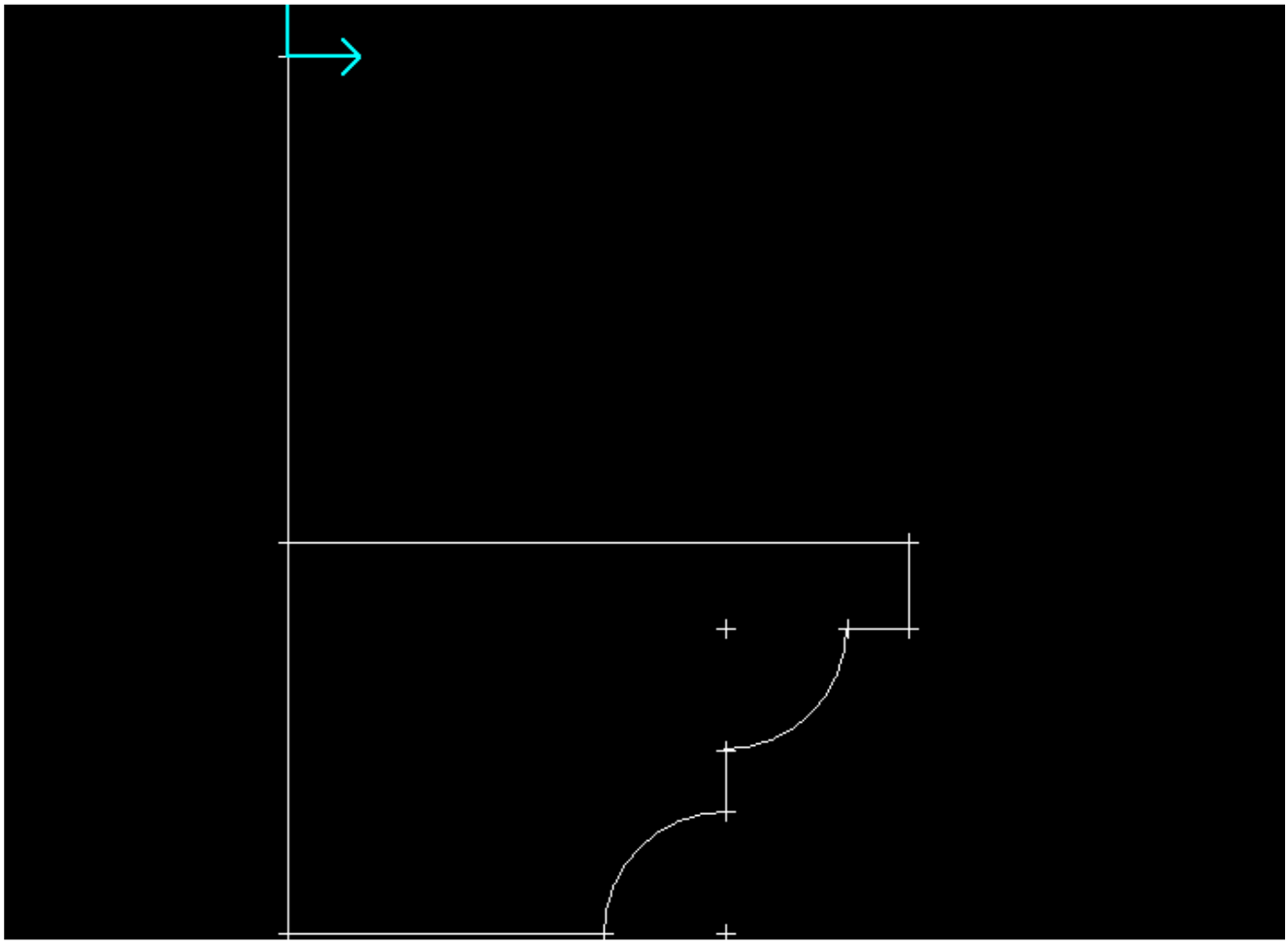

Figure 65

Press DRAW, CUT. The program tells us that you select the object from.

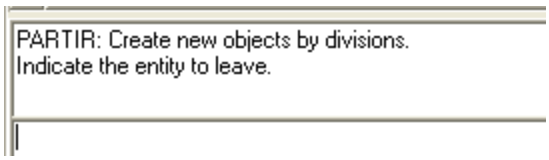

Figure 66

Select the desired line and you have selected the screen:

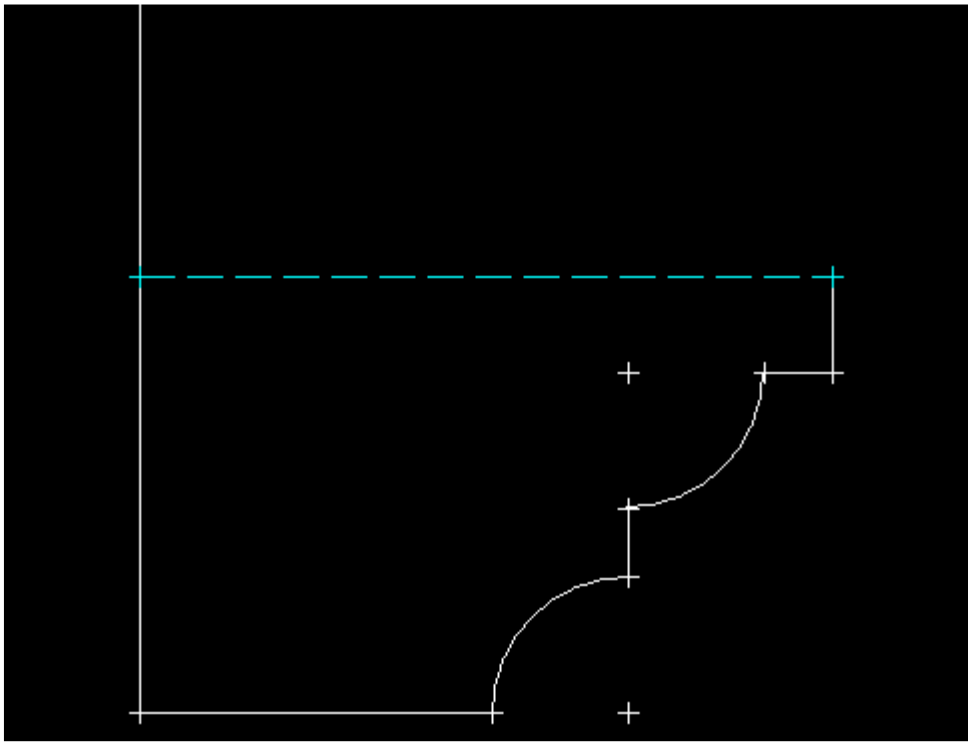

Figure 67

The program asks from the first point where:

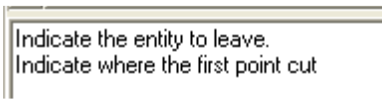

Figure 68

Indicated by the mouse (or write in the command bar if you know the coordinates) the point.

The same with the second point. As you split into two equal parts denote the same point as the first.

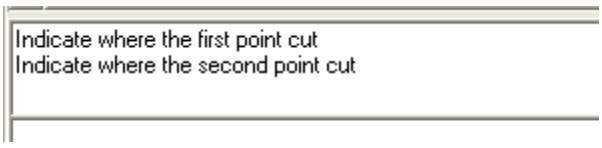

Figure 69

The line you have split in two.

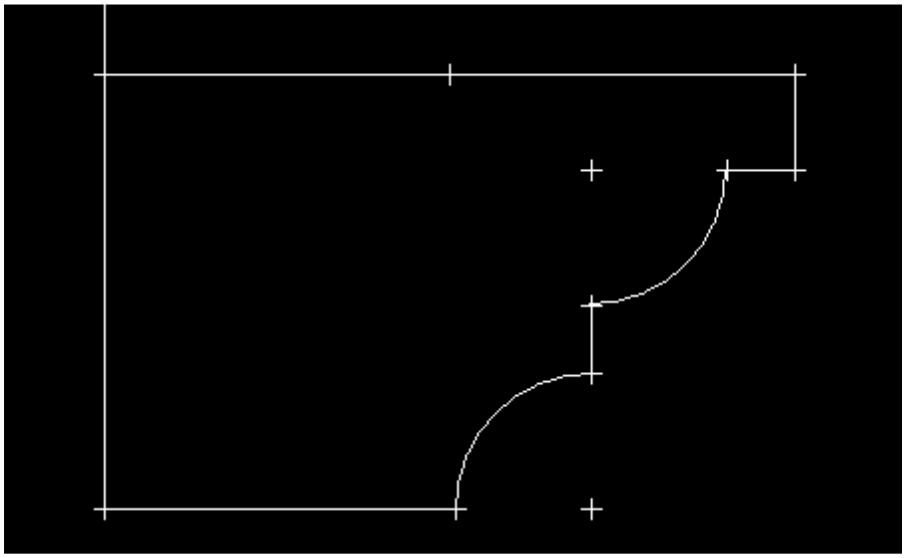

Figure 70

This command can also be done by typing in the text bar PAR or PARTIR.

4.3.10. Stretch

The STRETCH command permits scrolling objects and lengthens lines. The execution of the command stretch requires the definition of colons to indicate the window of selection, the dot of source and the endpoint of the shift. The lines contained partially are stretched so that the exterior dot to the selection square remains similar but the inside is scrolled.

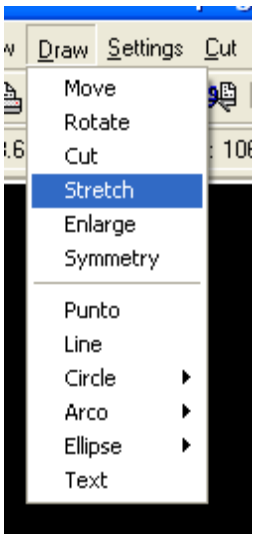

Figure 71

Example: You want to stretch the left corner of the drawing.

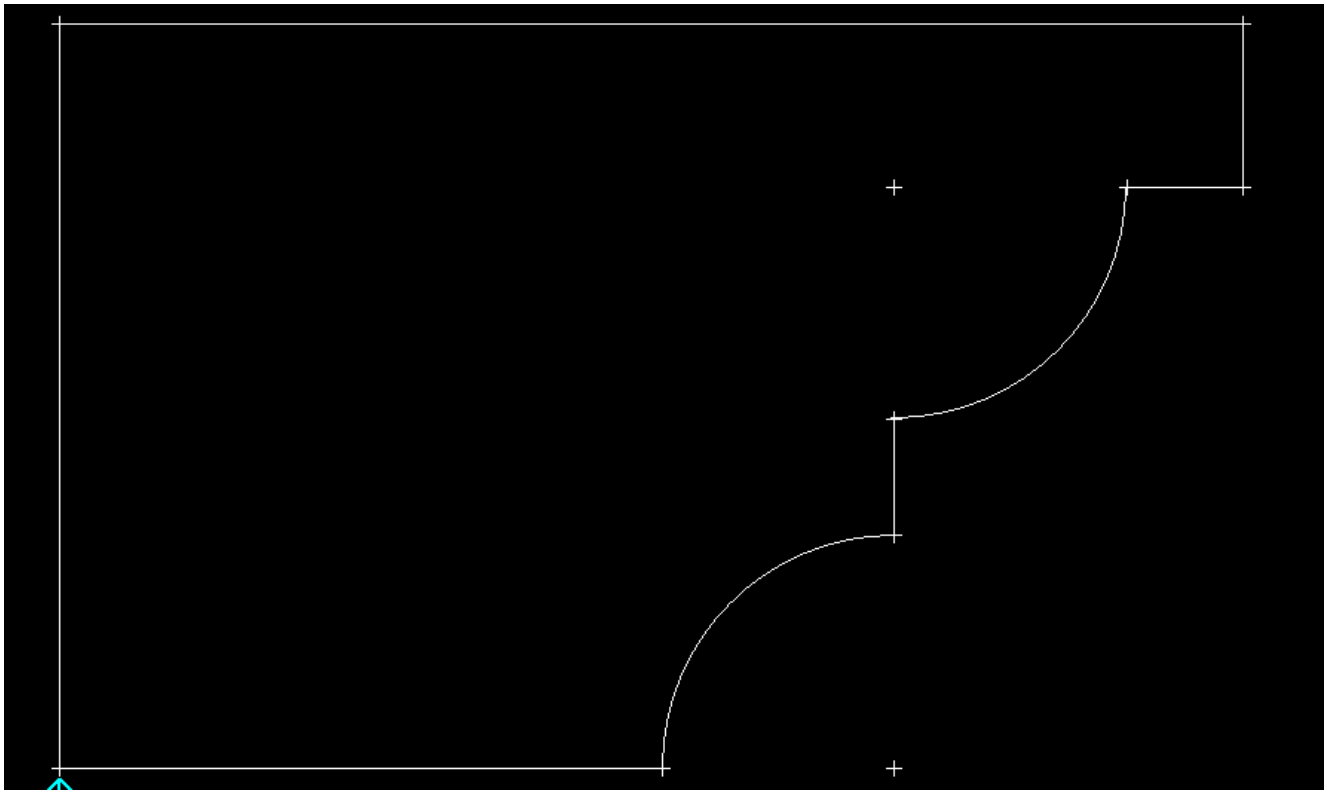

Figure 72

You press DRAW, STRECH. The program indicates that select the object to stretch.

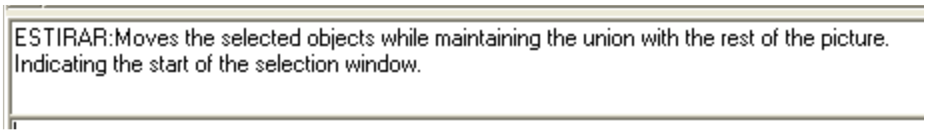

Figure 73

The mouse indicates the beginning of the selection window.

The program asks us to denote the opposite corner of the selection window.

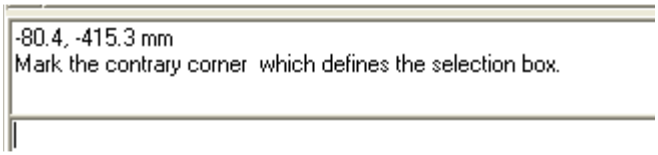

Figure 74

Then you asked to indicate the origin point of stretching. (Indicated with the mouse or by its coordinates if known).

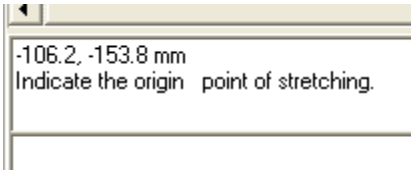

Figure 75

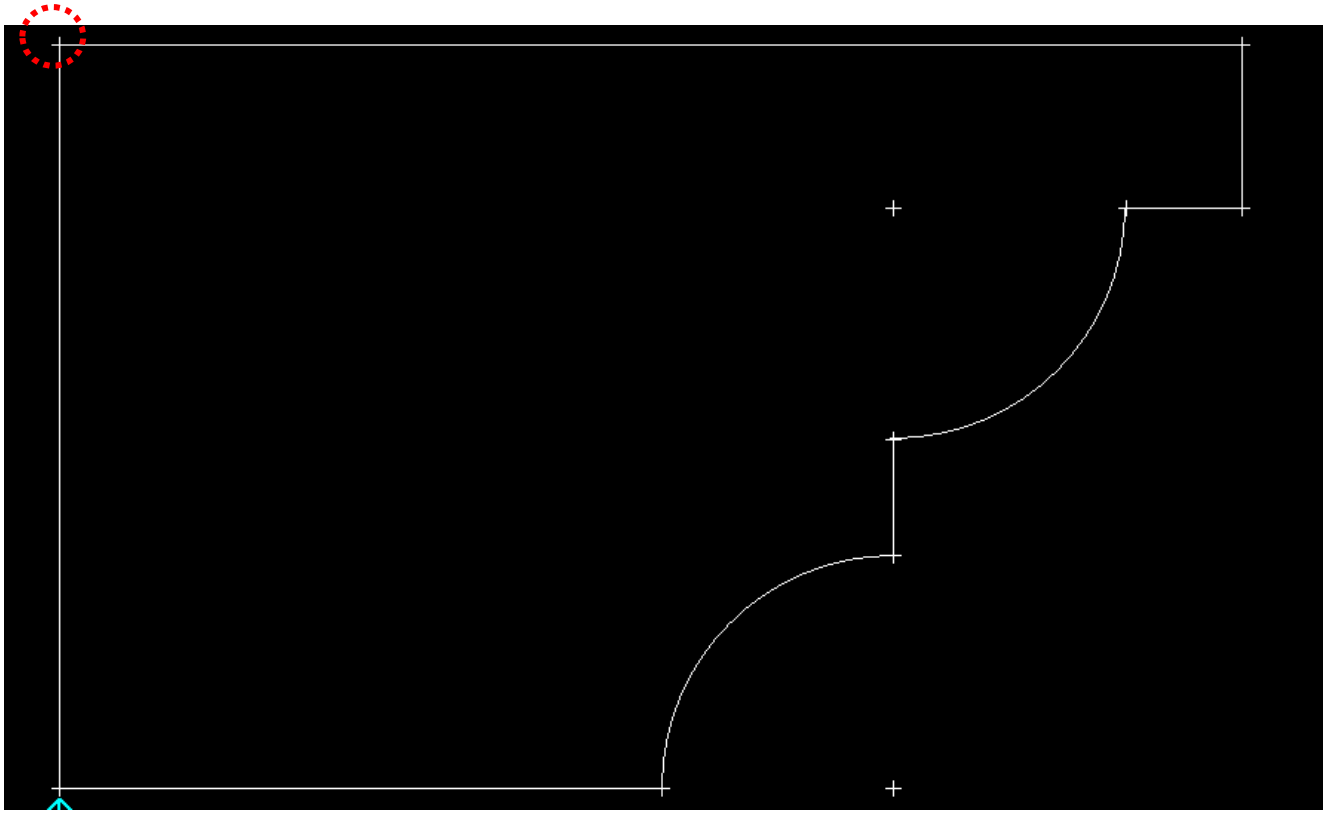

Figure 76

Indicate the destination. (Indicated with the mouse or by its coordinates if known).

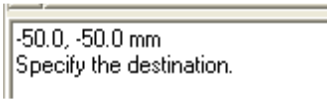

Figure 77

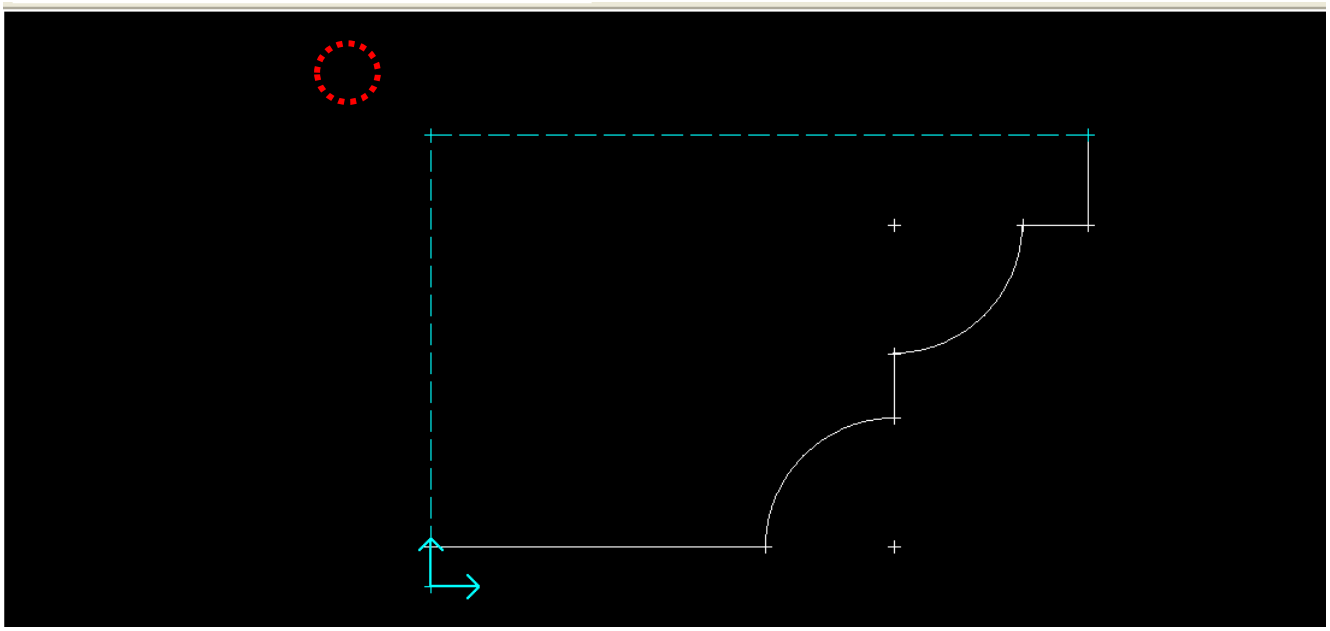

Figure 78

The selected corner stretches us such and as trying to get.

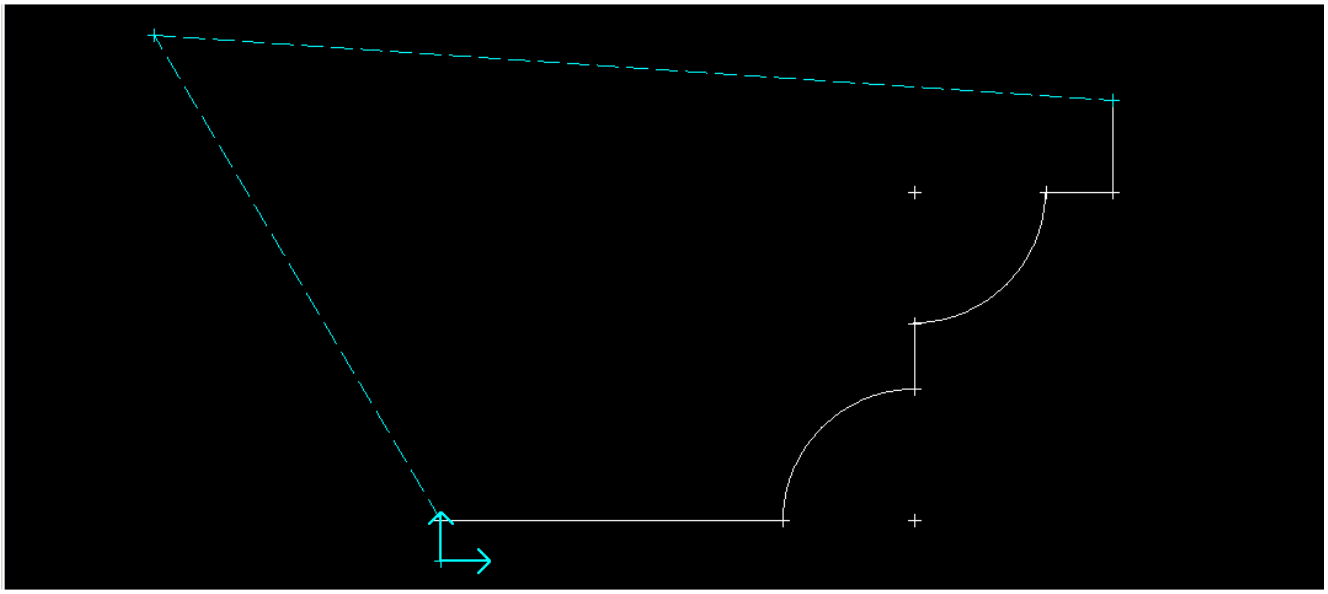

Figure 79

4.3.11. Enlarge

The execution of this command extends an object so that it finishes exactly at the boundaries defined by other objects. The program first requires the selection of the entity you want to lengthen and secondly the selection of the entity to which you want to extend the first.

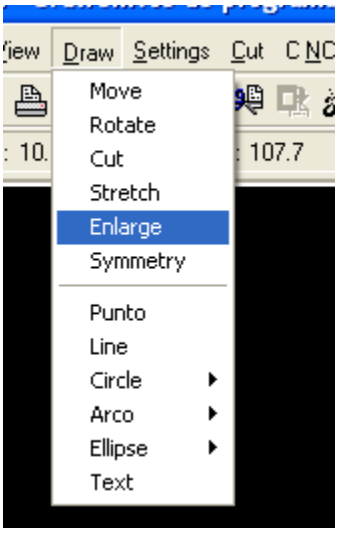

Figure 80

Example: You want to lengthen the superior line of the drawing to form a closed outline.

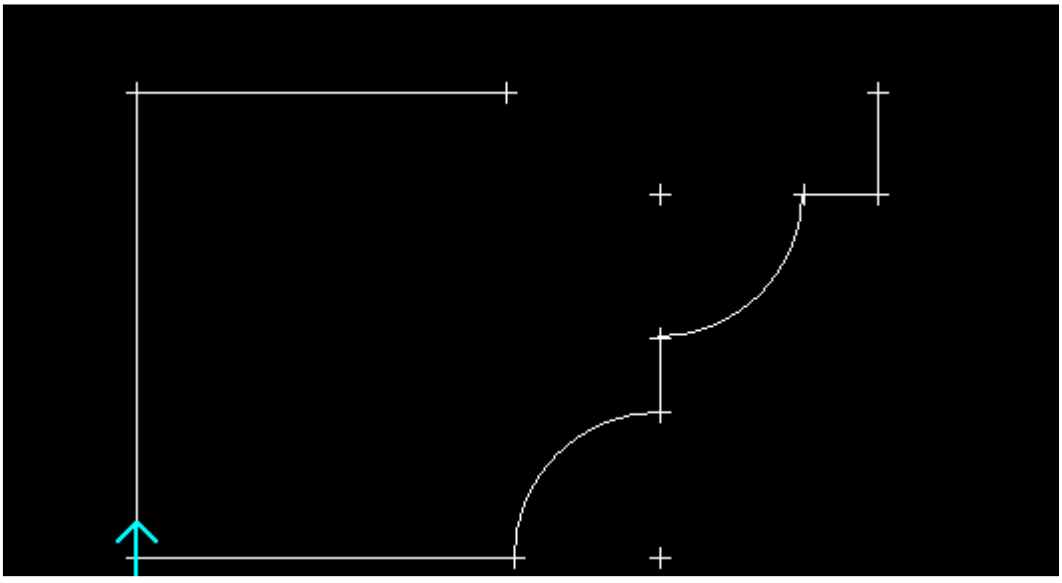

Figure 81

You press DRAW, ENLARGE. The program indicates that select the object to lengthen.

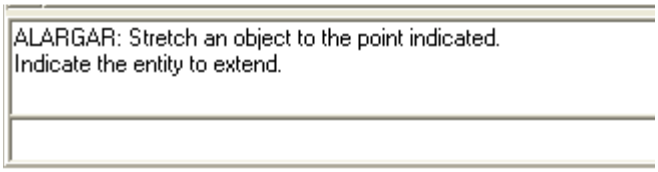

Figure 82

You select with the mouse the desired entity:

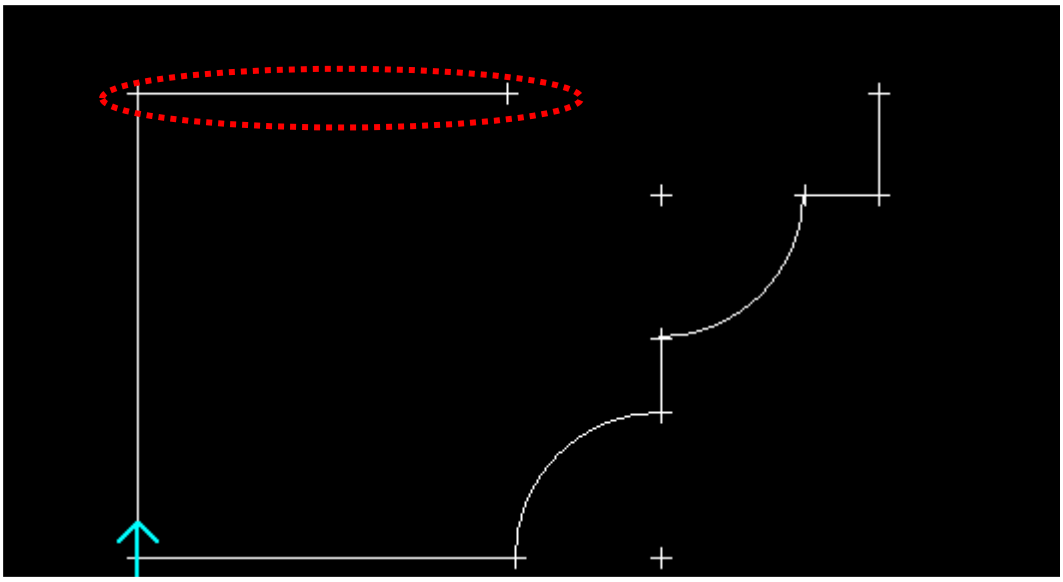

Figure 83

Then you asked to indicate the entity to which it extends.

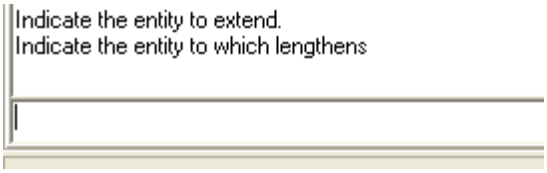

Figure 84

You select with the mouse the desired entity:

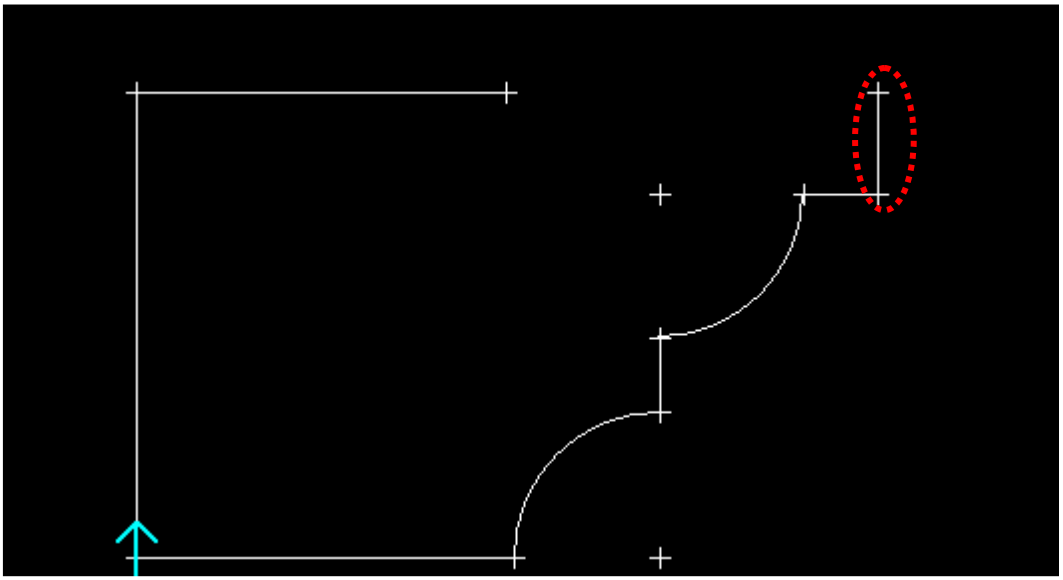

Figure 85

The line lengthens us such as trying to get.

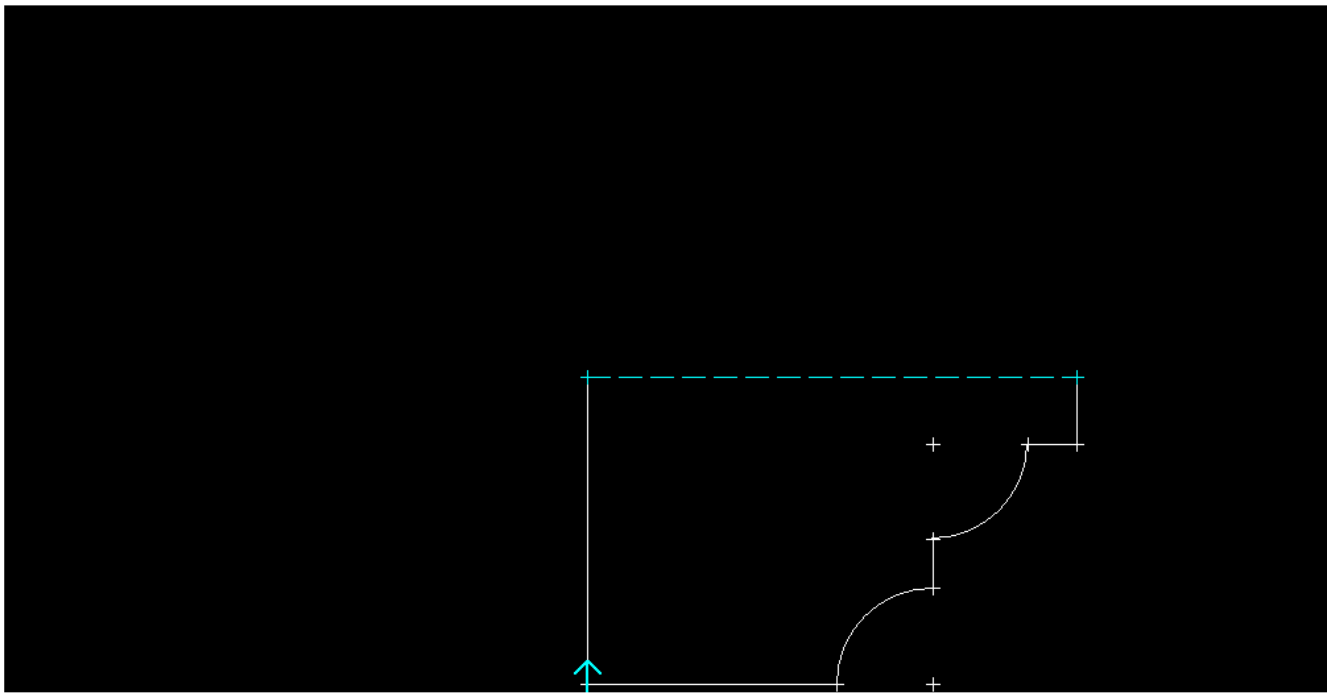

Figure 86

4.3.12. Symmetry

SYMMETRY command allows copies of objects previously selected symmetrical about an axis called the axis of symmetry. The objects that you want to back symmetry must be selected and the axis of symmetry should exist in the drawing before executing the order. In the event of any text within the selected objects, it is not reflected and only shows the symmetry point which indicates your position.

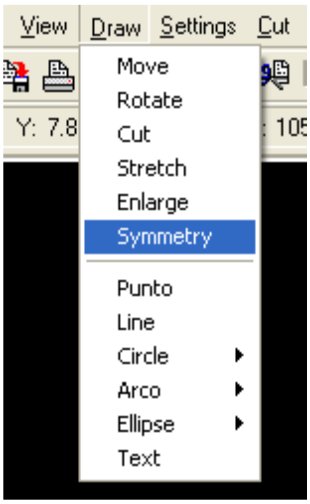

Figure 87

Example: You want to do a symmetrical piece like the figure which a symmetrical axe is his left side.

As our axis of symmetry is the left side, you need not draw previously. If the axis of symmetry is not what we had, you should draw it before executing the order SYMMETRY.

You select one piece that wants to apply symmetry.

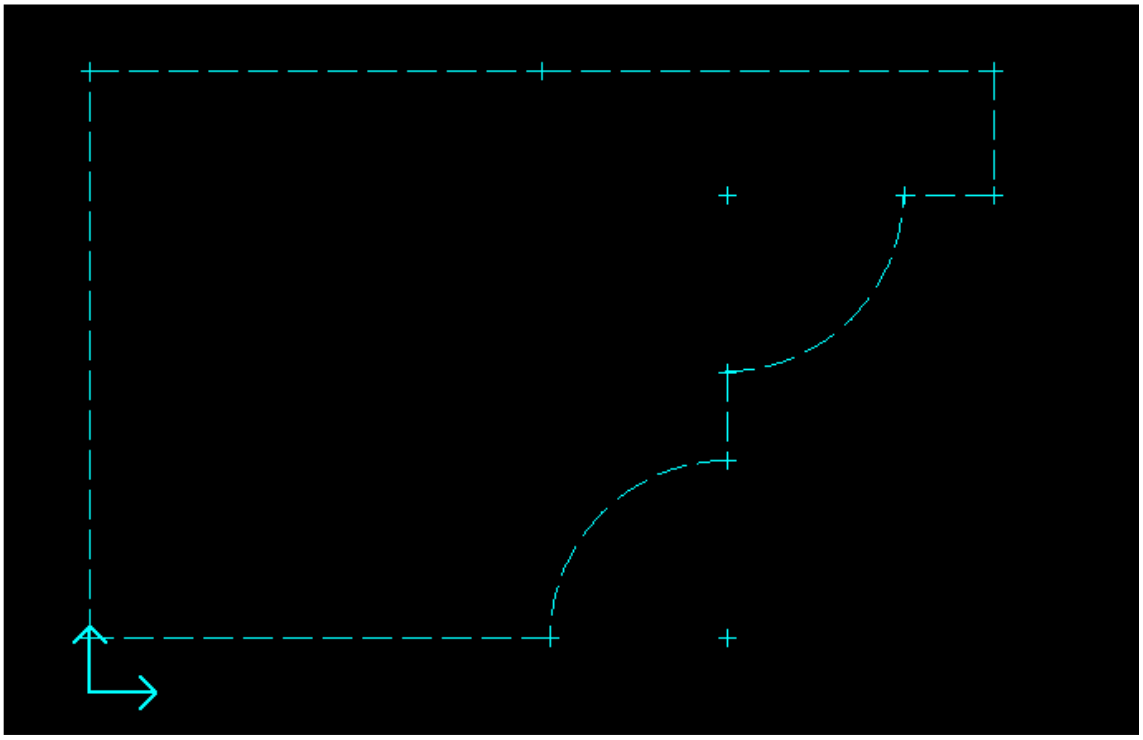

Figure 88

The program asks us that indicate the straight it will serve as an axis of symmetry (select it with the mouse).

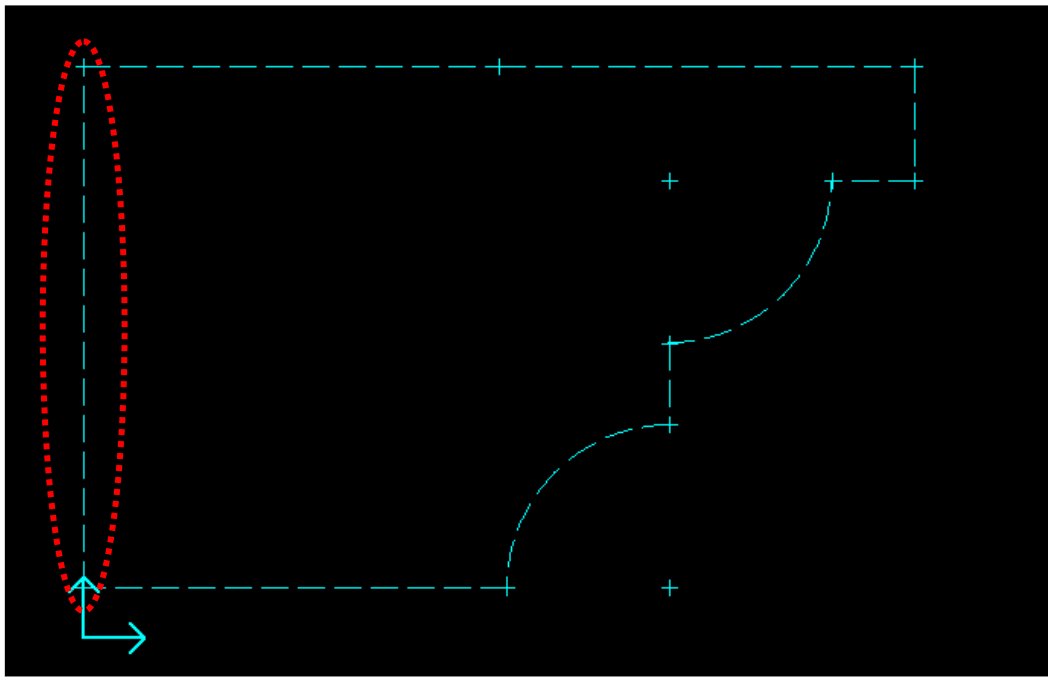

Figure 89

GraniCAD will do the symmetrical copy.

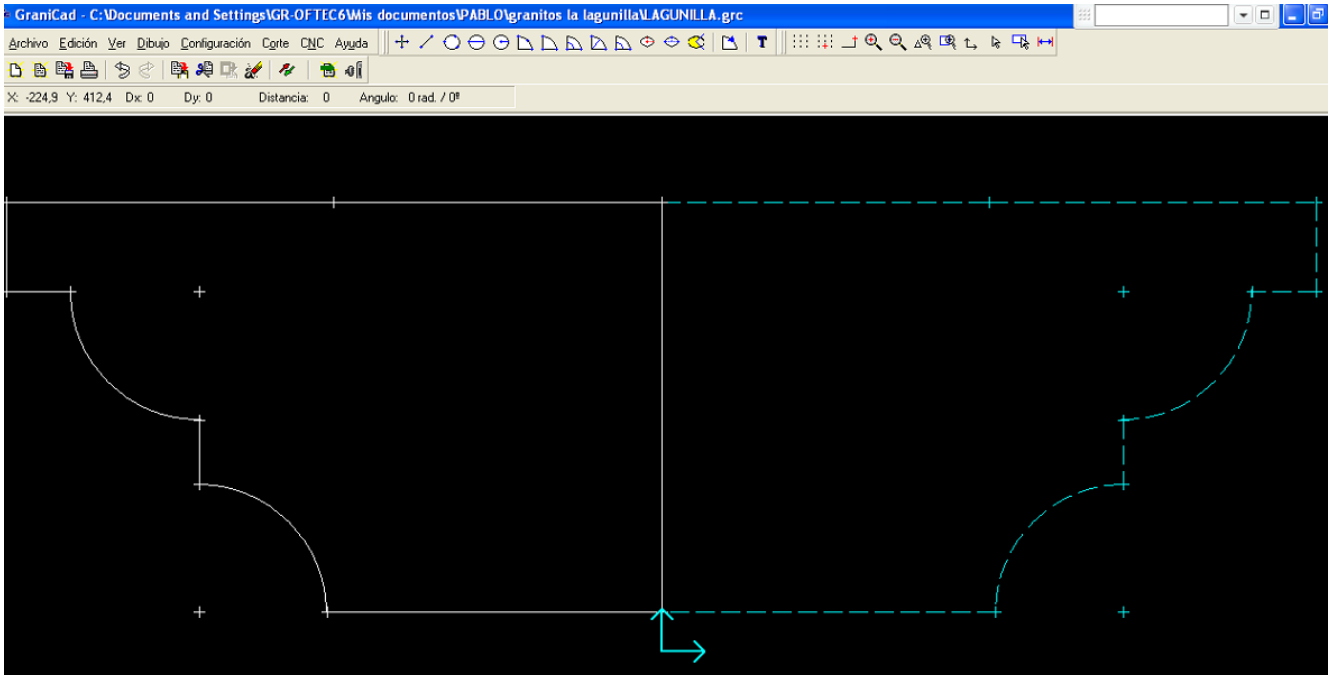

Figure 90

4.4. Menu EDIT

When you open the Edit menu, the user can execute a series of commands that can be classified into two groups: one object selection commands, and on the other hand, editing commands.

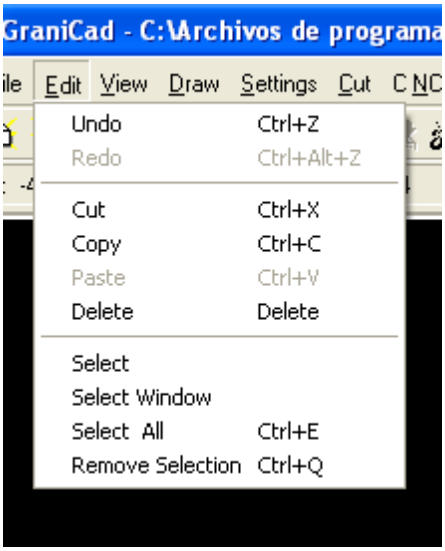

Figure 91

Within the editing, commands are: undo, redo, cut, copy, paste and delete.

Within the selection, orders are: select, select window, select all and remove the selection.

4.4.1. Undo

The Undo command allows the user to return to the situation before the last operation is performed. This command supports being executed several times in a row, so you can return to the state of drawing several operations ago. Most will be able to undo operations until the last time you ran the command SAVE.

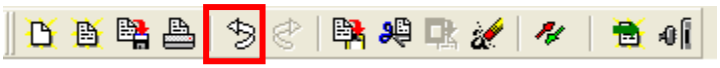

Figure 92

Your text-based command is DES or DESHACER.

Your fast-access key combination is CTRL +Z.

4.4.2. Redo

Command REDO reruns the last operation on the UNDO has been applied. UNDO, supports multiple levels, to re-run all the operations which have been previously wasted.

The shortcut icon REDO command is shown in the figure below:

Figure 93

Your text-based command is RE or REHACER.

Your fast-access key combination is CTRL + ALT + Z.

4.4.3. Cut

The Cut command deletes selected entities' drawings or drawings and stores them in a temporary space. In this space, the drawings are available to be pasted into another drawing.

The shortcut icon to the command CUT is shown in the figure below:

Figure 94

Your text-based command is COR or CORTAR.

Your fast-access key combination is CTRL +X.

4.4.4. Copy

The COPY command allows the user to copy the design's selected drawing entities and stores them in a temporary space where they can be retrieved and inserted into another drawing with the paste command.

The shortcut icon to the COPY command is shown in the figure below:

Figure 95

Your text-based command is COP or COPIAR.

Your fast-access key combination is CTRL+ C.

4.4.5. Paste

The Paste command inserts into the drawing objects or entities selected in the last Copy or Cut operation. Inserted entities can move indicating a point of origin and a destination. The vertical and horizontal distances between the points are introduced as initial and final point displacement to be applied to objects.

The shortcut icon PASTE command is shown in the figure below:

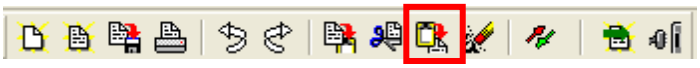

Figure 96

Your text-based command is PEG or PEGAR.

Your fast-access key combination is CTRL +V.

4.4.6. Delete

The DELETE command can delete drawing objects or entities selected. The shortcut icon to the DELETE command is shown in the figure below:

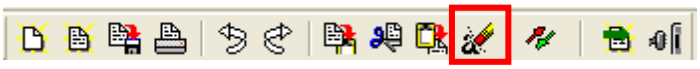

Figure 97

Your text-based command is BOR or BORRAR.

Your fast-access key combination is SUPR.

4.4.7. Select

The SELECT command allows you to select objects on the screen. To mark a particular object is to bring the mouse cursor to it, when the graphical cursor changes appearance, you can click the left mouse button to confirm the selection. To include more than one object individually, the user must hold down the CTRL key on the keyboard while selecting with the mouse.

If you want to delete an object from among those selected, must click on it again. If more than one object is selected, you can deselect by holding down CTRL while selecting the object to deselect.

The colour that shows the selected objects can be changed in the window: OPTIONS - DRAWING SET.

The shortcut icon to the SELECT command is shown in the figure below:

Figure 98

Your text-based command is SELE or SELECCIONA.

4.4.8. Select Window

With the command window selection, you can select all objects that are wholly or partially included in a graphic chart defined by the user. This table is created with the mouse indicating two opposite corners thereof.

The shortcut icon to the command window selection is shown in the following figure.

Figure 99

Your text-based command is SELE or SELECCIONA.

4.4.9. Select All

With the Select All command the user can select all objects in the drawing.

Its combination of shortcut keys is CTRL + E.

4.4.10. Remove Selection

With the command Remove Selection the user deselects all objects previously selected.

Its combination of shortcut keys is CTRL + Q.

4.5. Menu VIEW

The commands that appear when you open the View menu can be classified into three groups: help commands for drawing, viewing and command order "Distance."

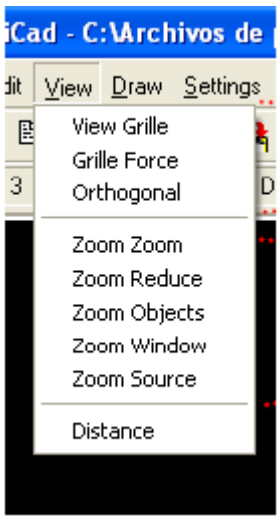

Figure 100

4.5.1. View Grid

View Grid command displays or hides a grid of points that the user can use as a reference and aid for the drawing. The horizontal and vertical distances and the colour of grid points can be modified in-DRAW- SET OPTIONS (See section 2.2.3 of this manual).

The shortcut icon to view the grid is shown in the figure below:

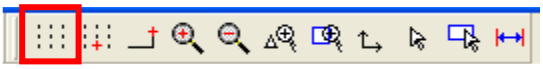

Figure 101

Your text-based command is REJ or REJILLA.

The aspect that presents GraniCAD when enabling the grid is the following:

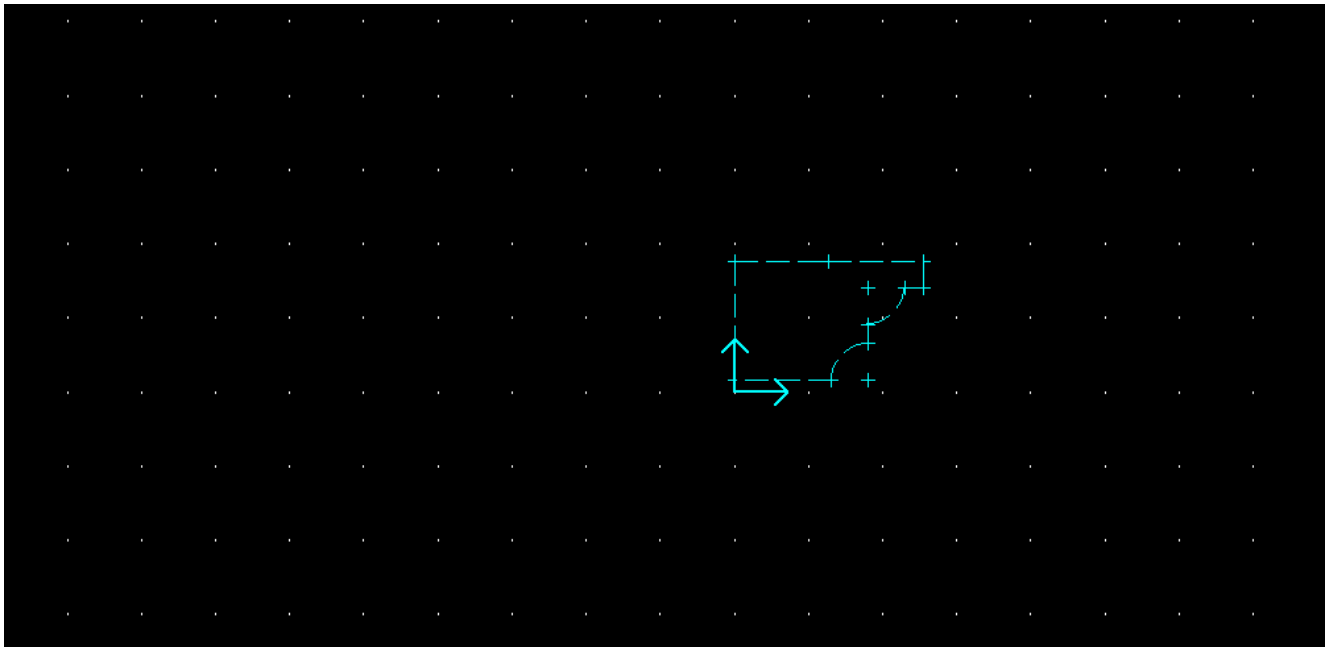

Figure 102

4.5.2. Grid Force

GRID FORCE command allows the user to use the grid points as a basis so that only can be used either grid points or the reference object (end, centre, intersection, etc.) when requested coordinates a point. This limit only applies when using the mouse, using the command line you can enter any value.

The shortcut icon to force the grid is shown in the figure below:

Figure 103

4.5.3. Orthogonal Mode

Orthogonal activation MODE allows the user to draw straight lines to form each 90 °, thereby facilitating the delineation of vertical and horizontal lines.

The shortcut icon is shown below:

Figure 104

Your text-based command is ORTO or ORTHOGONAL.

4.5.4. Zoom Reduce

ZOOM command increases the zoom factor display screen drawing to 0.5, thus allowing part of the picture to see a bigger size.

The shortcut icon is shown below:

Figure 105

Your text-based command is AMP or AMPLIAR.

4.5.5. Zoom Reduce

Unlike the extended zoom command, the command ZOOM REDUCE decreases the drawing factor display screen in 0.5, allowing to have more screen objects.

The shortcut icon is shown below:

Figure 106

4.5.6. Objects Zoom/See All

In the implementation of the ZOOM OBJECTS command, the program selects the zoom factor necessary to adjust the picture on the screen completely. After clicking on the screen automatically see everything You have drawn.

The shortcut icon is shown below:

Figure 107

4.5.7. Window Zoom

The ZOOM WINDOW command execution requests the user to enter the coordinates of two points that define a window to open and then the enclosed part of the picture in that window.

The shortcut icon is shown below:

Figure 108

4.5.8. Zoom Source

ORIGIN ZOOM command adjusts the zoom display on the screen until the origin, point 0.0, is visible.

The shortcut icon is shown below:

Figure 109

4.6. Cut Menu

Once you create a profile that is desired, the next step is to generate the cutting path, for which you need to define various parameters for it to be perfectly defined.

Through the short menu, the user can access three commands:

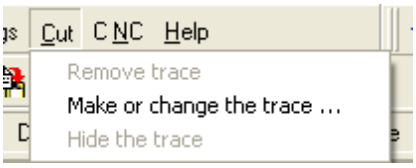

Figure 110

4.6.1. Creating or modify the trace

When you run the command CREATE OR MODIFY THE TRACE displays a dialog box.

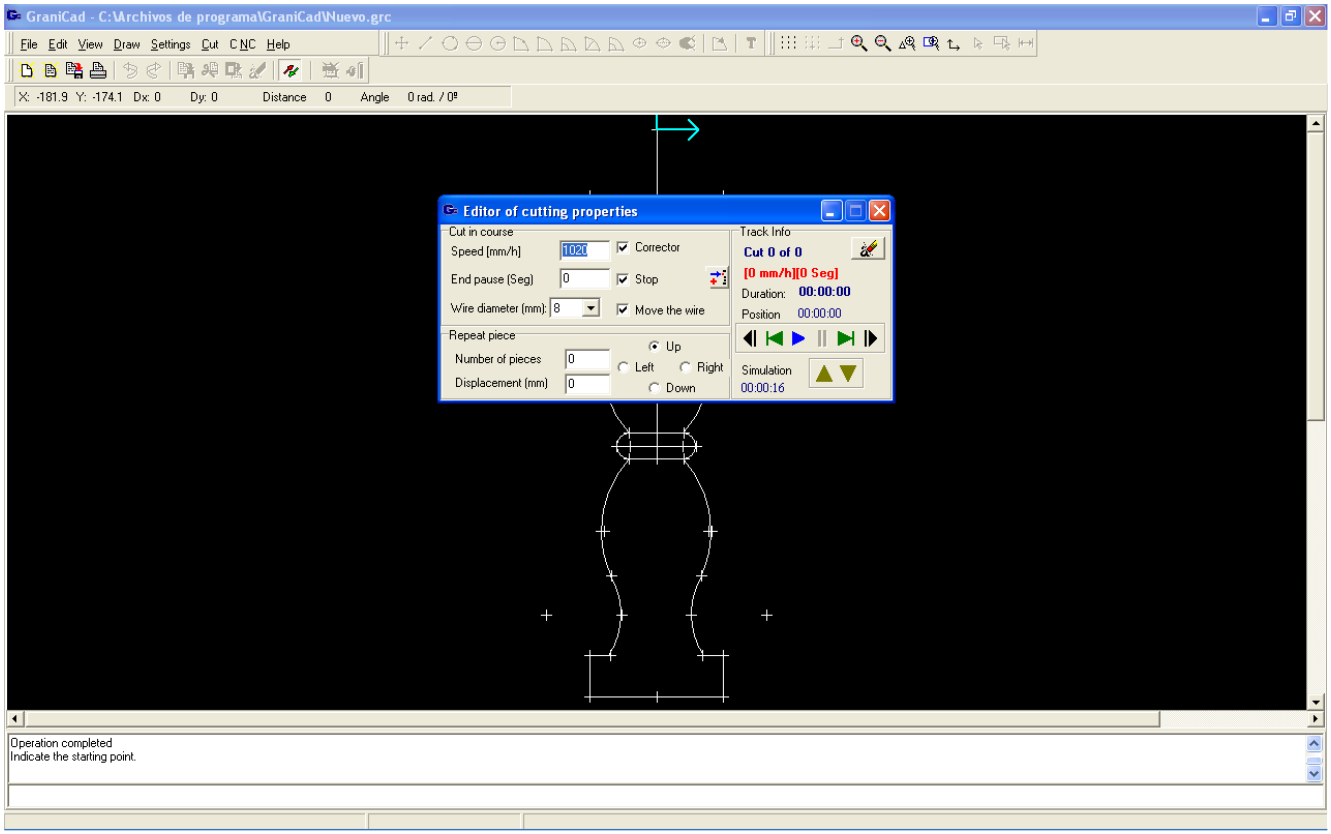

Figure 111

This table contains all parameters to be defined and represents the conditions under which to make the cut of the figure that has been previously designed. This dialogue box also lets you change the parameters already defined for a given stroke. The table can move and minimize out of the way in the work area during use.

You consider it divided into four zones:

- CUT PROCEEDING Area;
- PART REPEAT Area;
- LAYOUT INFORMATION Area;
- SIMULATION Area.

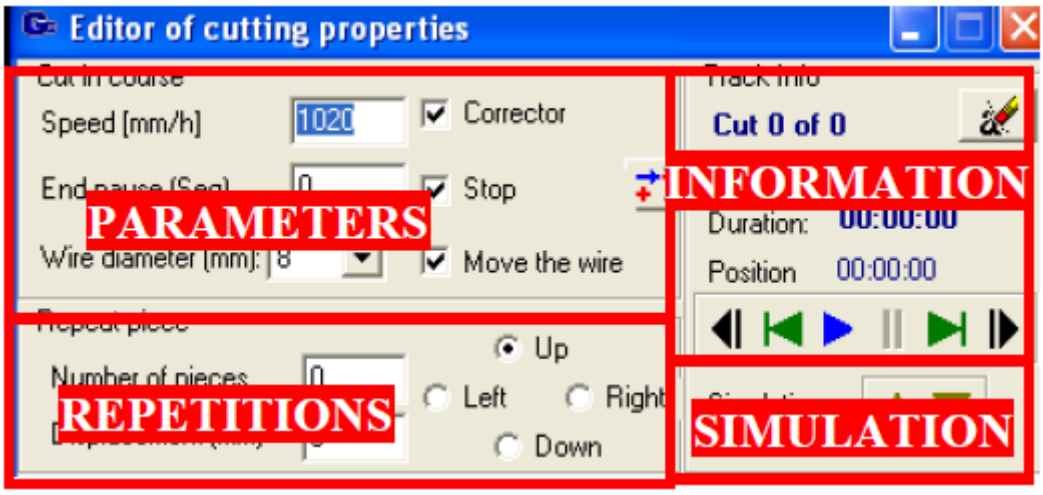

Figure 112

Your text-based commands are CREACAM, CREARCAMINO, MODCAM or MODIFICAR CAMINO.

4.6.1.1. Cut in Course

In the area of CUT IN COURSE can define to him or modify the following parameters:

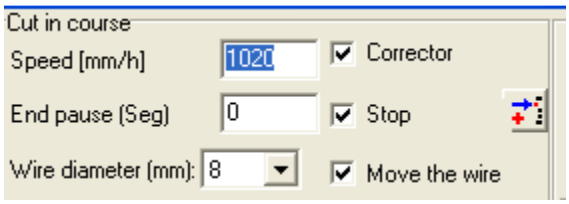

Figure 113

- ADVANCE (mm / h): In this box, the user must enter the speed of the cut. The value can be defined in inches/hour.
- End pause (sec): This defines the run time pause to take place after reaching the endpoint of the cut. If the entered value is 0, should be delayed pause. a
- Wire diameter (mm): Specify the wire diameter that you can use. The value entered is unique to the whole piece, not being able to define different diameters for different sections.

- CORRECTOR: When defining the cutting path, the user can indicate which side of the geometry created will move the thread, to respect and maintain the desired dimensions. Checking this box, the program generates the necessary orders to take into account the correction defined. If not check this option, the thread made the journey through the stages set up.

- STOP: If this option is checked, the wire does not make stops in the movement of the wire defined at the end of each section.

- MOVING THE WIRE: This box allows the user to indicate whether that stretch will be done cutting, moving the floss for that purpose, or treading a path already cut or a dead stone to carry any particular position.

To assign values as explained above you use the highlighted button in the picture:

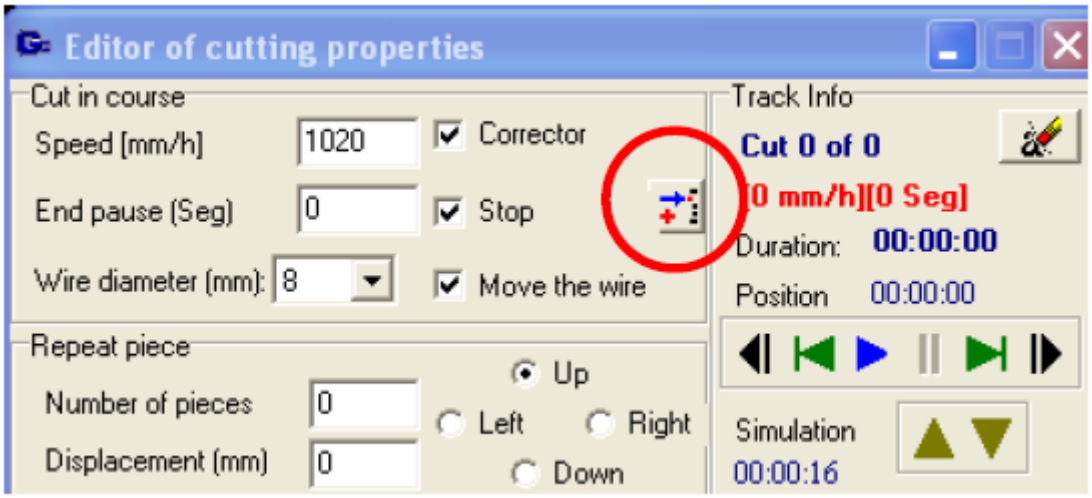

Figure 114

Clicking this icon is assigned to the section in which is located the cutting parameters that you defined.

4.6.1.2. Repetition of Pieces

REPEAT Area PARTS, the user may specify the number of identical parts you want to perform. This is to indicate the number of pieces that are to be cut, the displacement from the final position of the court, and the sense of repetition: vertical or horizontal.

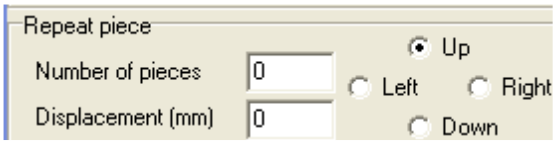

Figure 115

4.6.1.3. Information on the Cut

The options in the area located LAYOUT INFORMATION user can see and move about the different sections of cut and created:

Figure 116

- TRACE an OF b: It is shown in the piece of appointed to window INFORMATIONS the features of cut of the section to of a total of sections b.
- C cm/h ; d seg: it indicates between brackets the speed of cutting allocated c to the section under discussion. Also, it shows between brackets the time of d duration of the cut of that section.
- DURATION: Shows the approximate time it will take to make the full cut of the piece (adding all the sections) taking into account the parameters defined for each section. In case of repetition of parts, the right time must be multiplied by the number of pieces.
- POSITION: Sampling in values of time the position in which is found the cut; that is, the employee time until the moment; by permitting how long approximate it takes a long time in reaching to a certain dot of the cut.
- Button ERASE.

Figure 117

Eliminates cutting sections from the stage set in which you find when you press.

- CURSOR POSITION: Enables the user to fast forward, rewind, start a simulation, stop it, go to the end of a section or the entire court, go to the top of a section or the entire court, etc.

4.6.1.4. Simulation

The program GraniCAD gives the user the ability to simulate on-screen cutting off the piece with all its features: feed rates, downtime, and movement of the thread. The duration of this simulation can be controlled by the two arrows that are highlighted in the following figure:

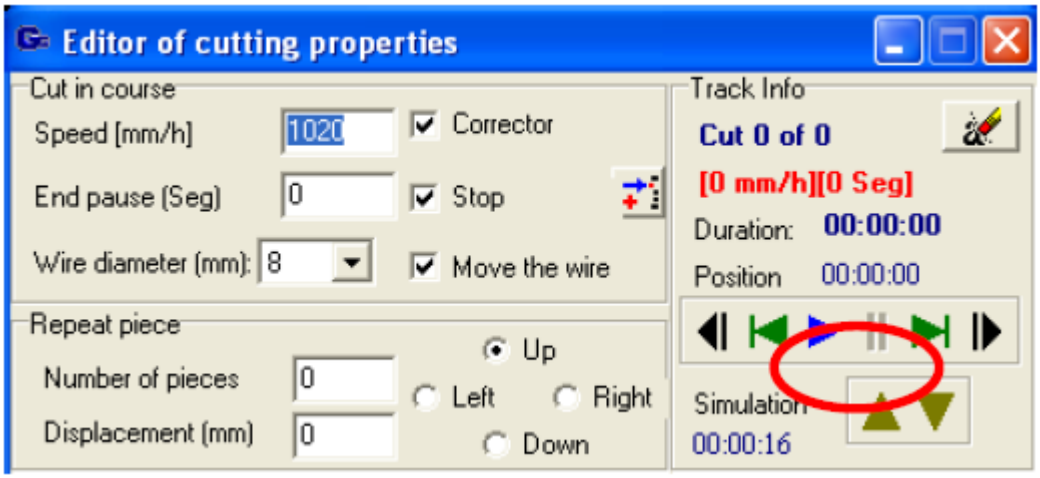

Figure 118

Through these arrows the user can increase or reduce this time, assigning to the simulation from a few seconds until the actual time of cutting.

Once generated the road a piece you would have to look like the example shown below:

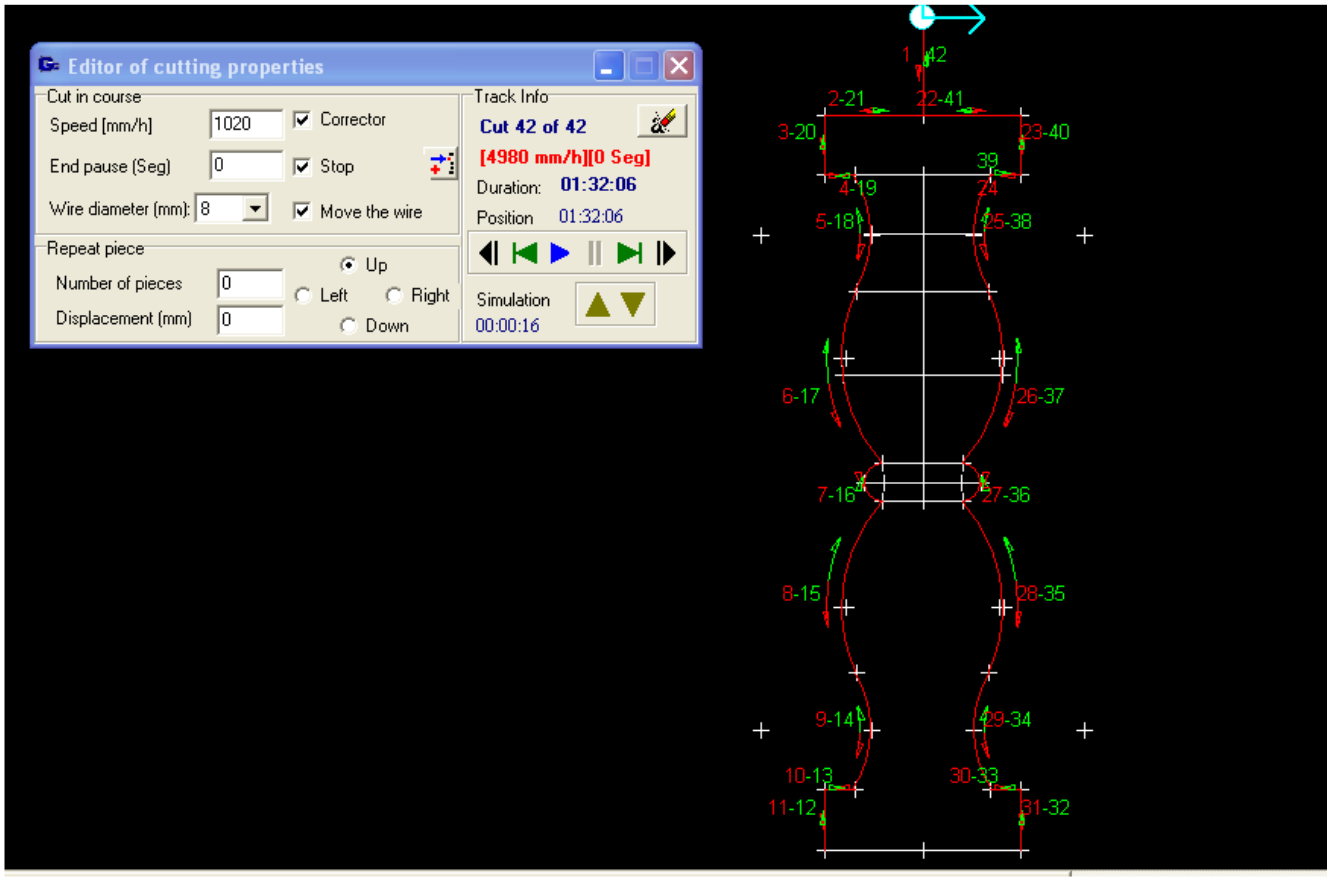

Figure 119

For each trace shows an arrow pointing to the direction that crosses the wire and is drawn on the side from which it travels. Respecting the right side, the arrow is drawn on the left if the left is respected, it must do the right and if not used the spell, will be drawn on the track. When you create a line that runs counter to an existing path is drawn in green to make it more visible and distinguishable from the original.

4.6.2. Remove Trace

With this command you can delete all profiles created as a way of cutting, leaving clean to re-define it again.

4.6.3. Hide Path

Allows the disappearance of the icons on the screen generated during the cutting path definition.

4.7. CNC Menu (Numeric Controller)

Once created the cutting way the next step will create the code for the CNC. This can be done from this menu.

Once you display de CNC menu, there are two orders relating to the generation of numerical control files and communication with the machine.

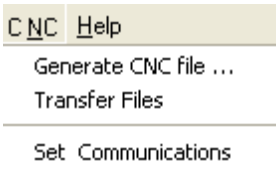

Figure 120

4.7.1. Generate CNC File

By GENERATE CNC FILE command generates the CNC program commands understood by the CNC for the design and pre-defined cutting path. The instructions are displayed in a box that allows editing, printing and saved as ASCII code extension. XPI.

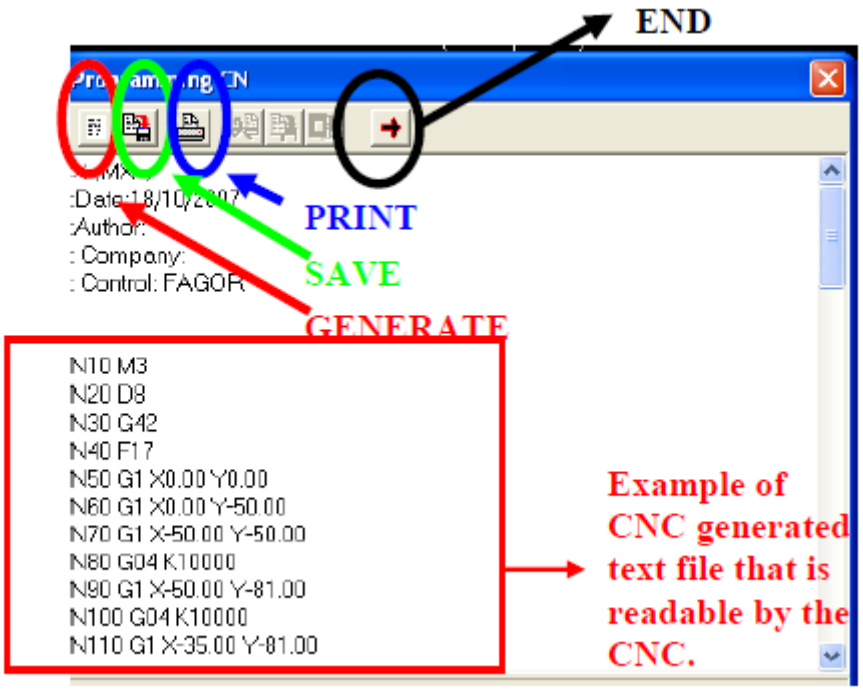

Figure 121

To generate the CNC file which it's going to be transferred to NC, you must have previously defined the cutting path and then click Save on the screen that shows the previous figure. Note that you'll be able to send only the programs stored on your computer whose name is composed of six numbers. It is important to bear this in mind when working with any external editor, GraniCAD will force you to save your files under this classification.

The shortcut icon CNC FILE GENERATE command is shown in the figure below:

Figure 122

The instructions for the text-mode command are **GENCN** or **GENERARCN**.

5. PART DESING EXAMPLE

You will explain step by step and in detail, the design of a concrete form in a block of stone, as an example and thereby facilitate the work of understanding this manual to the operator.

Suppose you want to make the next cut in a block:

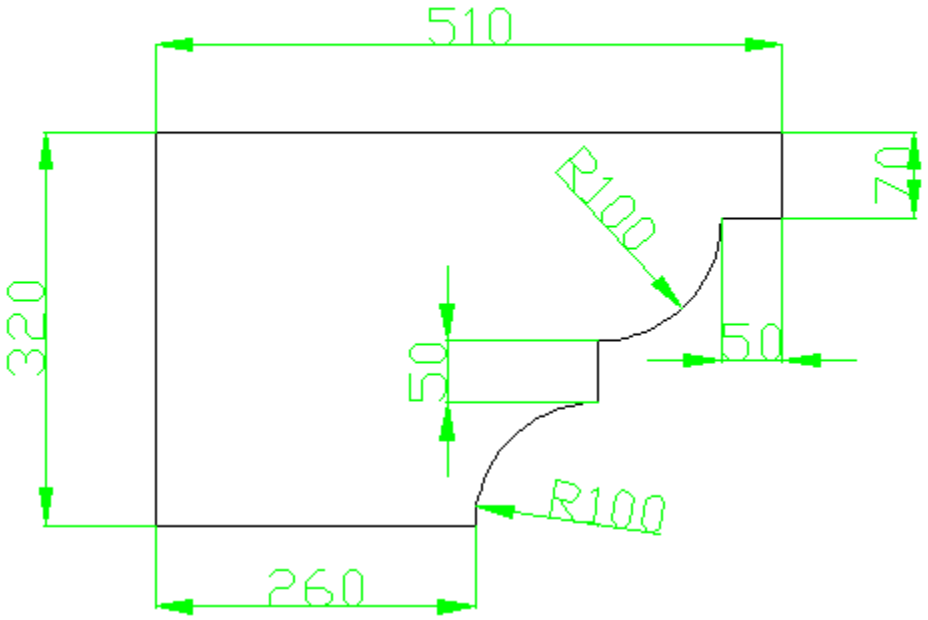

Figure 123

The steps are:

- Draw the trace;
- Drawing the block entryway;
- Locate the drawing in the origin;
- Generate the cutting way;
- Generate the CNC File;
- Generate the Cut Profile.

5.1. DRAW THE TRACE

You run the GraniCAD program.

Figure 124

You started drawing, for example, a lower horizontal line measuring 260 mm. To do this click with the mouse on the command line:

Figure 125

or type in the text-line **LIN** or **LINEA**. Then the program asks us to indicate the starting point. You define it by clicking the left mouse button or by typing the coordinates of a point. You take for example the point 0, 0.

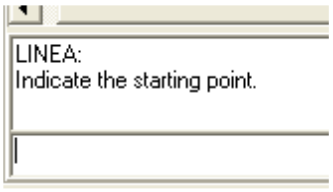

Figure 126

Press ENTER, and the program will ask us to indicate the endpoint of the line. As you know the length of that line it'll be easier to work with relative coordinates (you will consider as a

reference the starting point of that line). Therefore you write **@ 260.0** or **R260, 0**. Press ENTER twice to validate the command and end the command line. It will be shown on the screen.

Figure 127

Figure 128

Then you draw, for example, the vertical line on the left measuring 320 mm. To do this, click with the mouse on the command line:

Figure 129

or type in the text-line LIN or LINEA. You'll be asked to indicate the first point of the line, and with the mouse by clicking on the first point, since the reference object allows us to indicate with precision by placing the mouse close to the beginning of the line you drew earlier. You may also be indicated by typing text as the starting point 0, 0. It asks for the end-point line, and he indicated by relative coordinates @ 0,320 or R0, 320. Press ENTER twice to validate the command and end the command line. It will be shown on the screen.

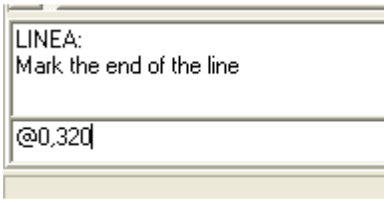

Figure 130

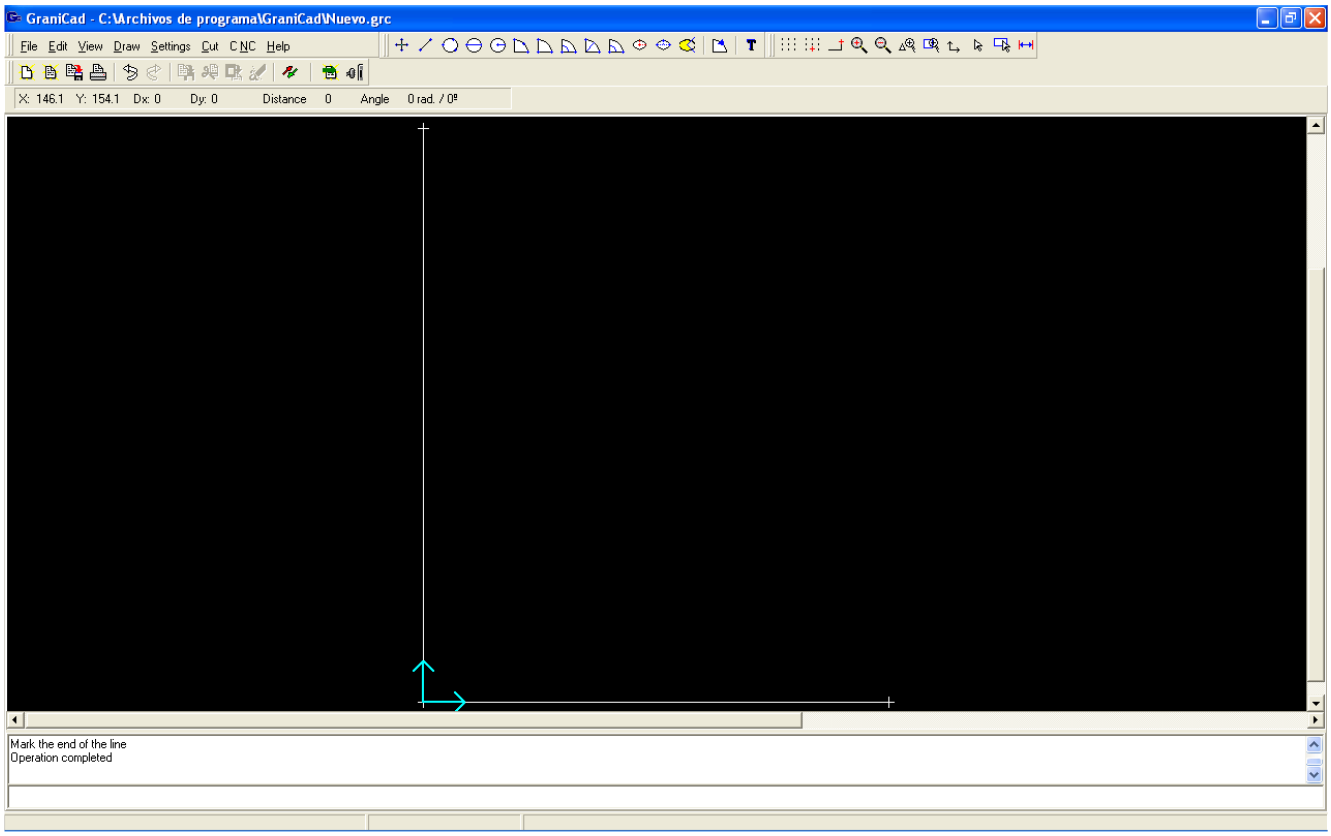

Figure 131

The next point that is drawn is the top horizontal line measuring 510 mm. Click with the mouse on the command line:

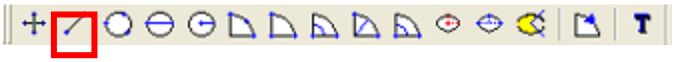

Figure 132

or write in the text-line LIN or LINEA. You'll be asked to indicate the first point of the line, and with the mouse clicking on the first point since the reference object allows us to indicate with precision. You can also indicate via text if you know the exact starting point in absolute coordinates of that line, which in our case this point corresponds to 0,320, so you would write as the starting point Section 0,320. The endpoint of this line is to write it again and it will be relative coordinates @ 510, 0 or R510, 0. Press ENTER twice to validate the command and end the command line. It will be shown on the screen.

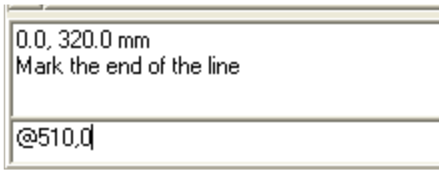

Figure 133

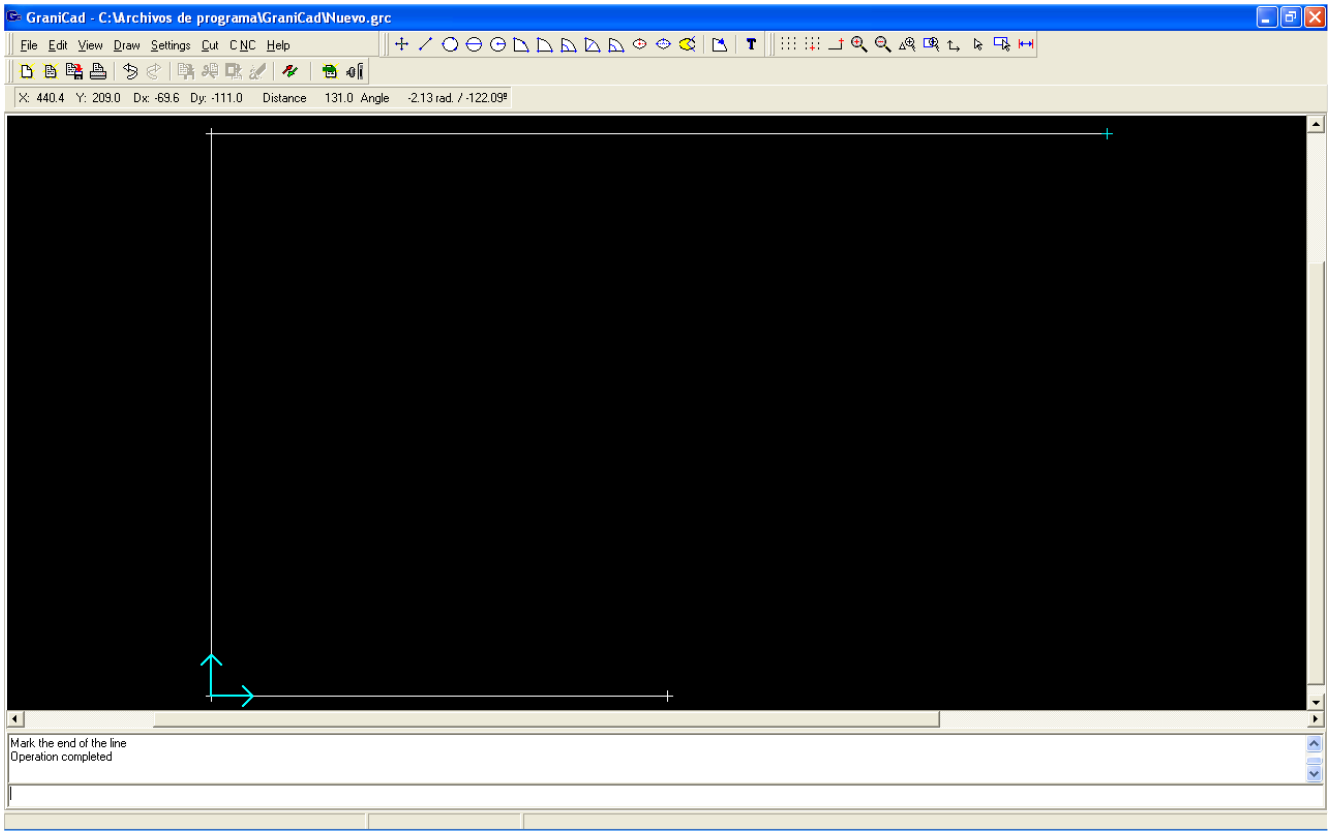

Figure 134

The next part of the figure is going to be a right vertical line of 70 mm long. To do this click with the mouse on the command line:

Figure 135

or write in the text-line LIN or LINEA. You'll be asked to indicate the first point of the line, with the mouse by clicking on the first point, since the reference object allows us to indicate with precision. You may also indicate it by typing the starting point **510,320**, which is corresponding to that point's absolute coordinates.

It asks for the end-point line, and he indicated by relative coordinates **@ 0,-70** or **R0,-70**. Press ENTER twice to validate the command and end the command line. It will be shown on the screen.

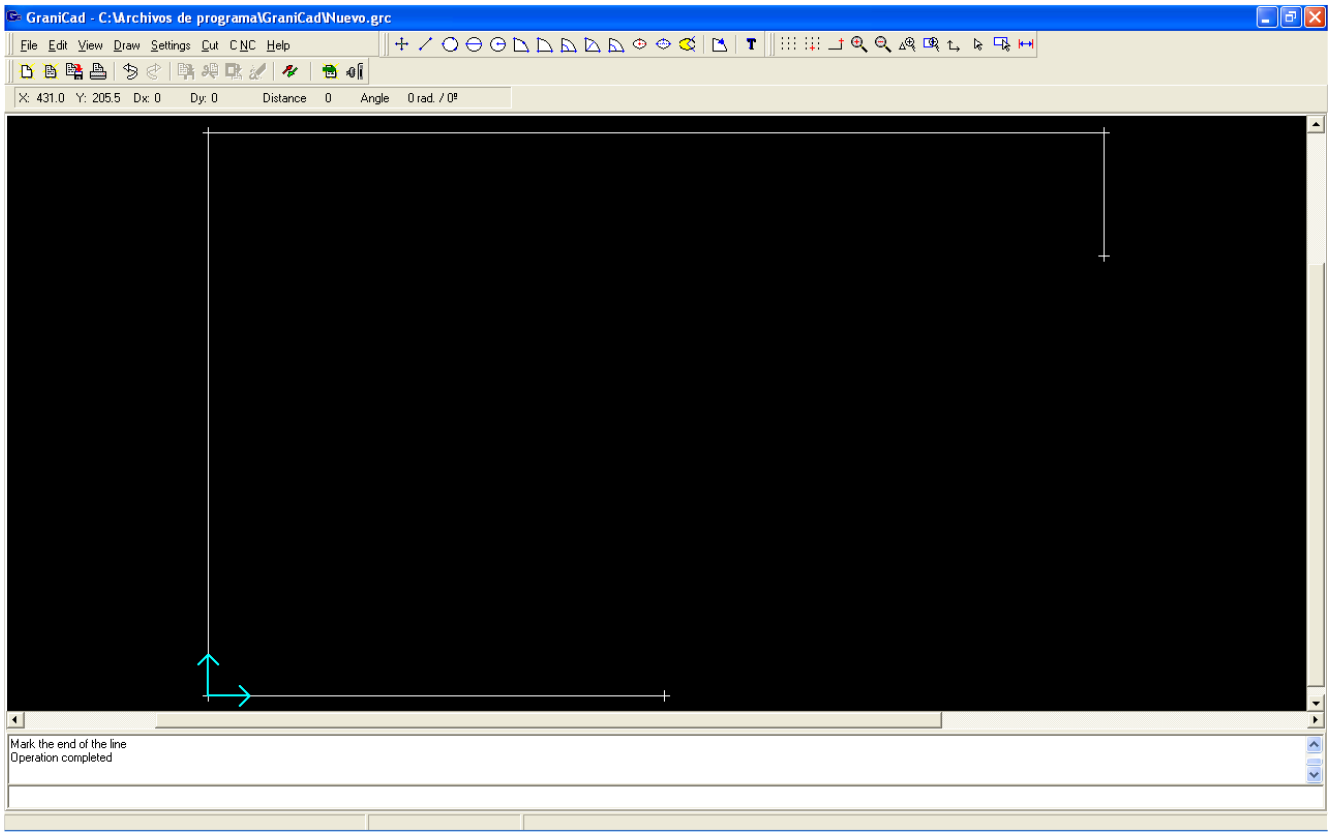

Figure 136

Now you'll draw the horizontal section of 50mm.

Click with the mouse on the command line:

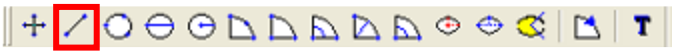

Figure 137

or write in the text-line LIN or LINEA. You will be asked to indicate the first point of the line, and with the mouse clicking on the first point since the reference object allows us to indicate with precision. You may also be indicated by typing text as the starting point 510,250, which corresponds to that point in absolute coordinates. It asks for the end-point line, which will be indicated by relative coordinates @ -50, 0 or R50,0.

Press ENTER twice to validate the command and end the command line. It will be shown on the screen.

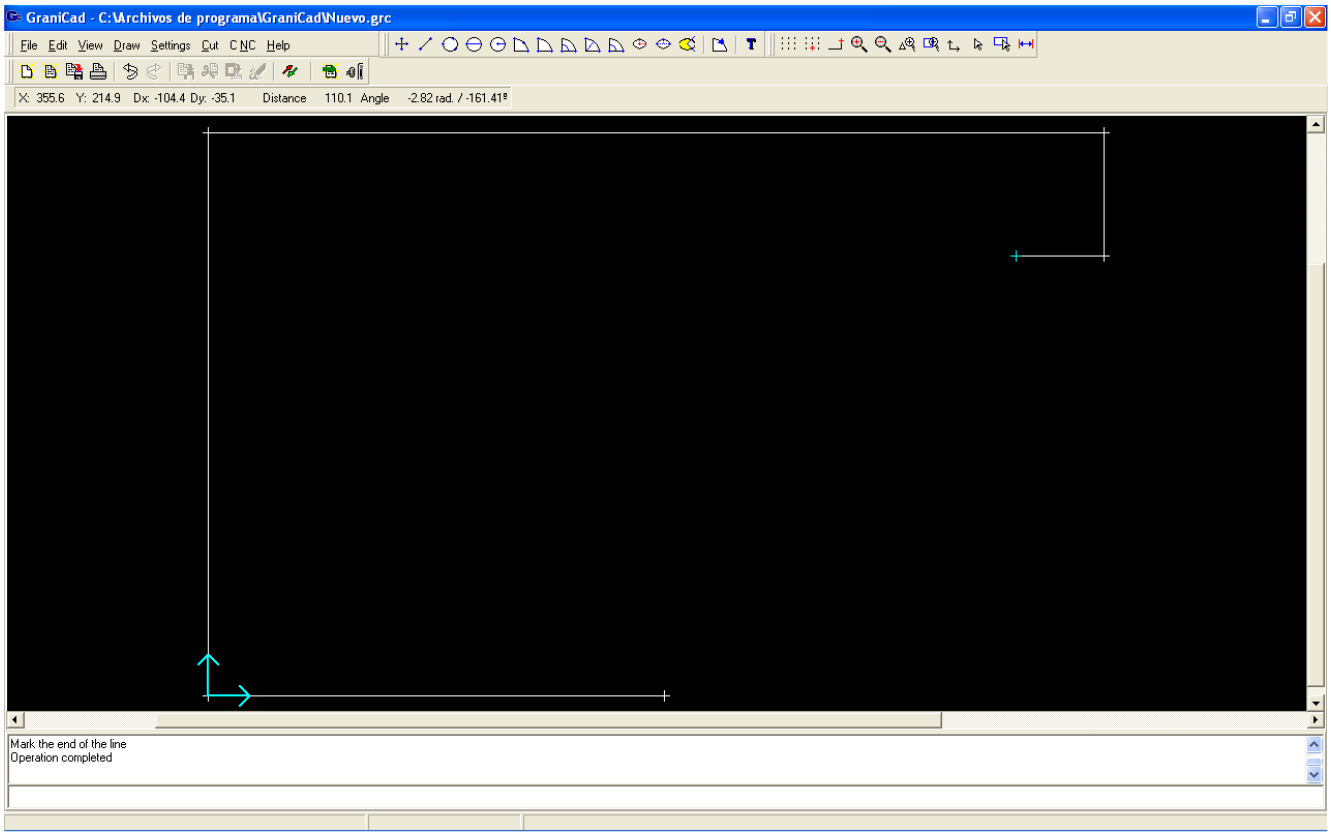

Figure 138

Continue drawing the bow that is located above. You note that the button arc is clockwise direction, and if not select it.

Figure 139

or write in the text-line ARCIFC or ARCOIFC. You will be asked to indicate the first point of the bow, and with the mouse clicking on the first point since the reference object allows us to indicate with precision. You may also be indicated by typing text as the starting point 460,250, which is corresponding to that point in absolute coordinates.

He asks the endpoint of the arc, and you indicate with absolute coordinates 360,150.

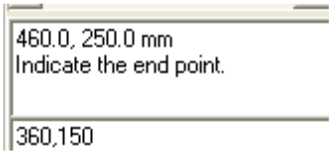

Figure 140

You will be asked to determine the centre of the arc, and you indicate with absolute coordinates **360,250**.

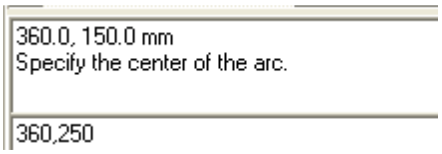

Figure 141

Press ENTER to validate the command and what you draw on the screen.

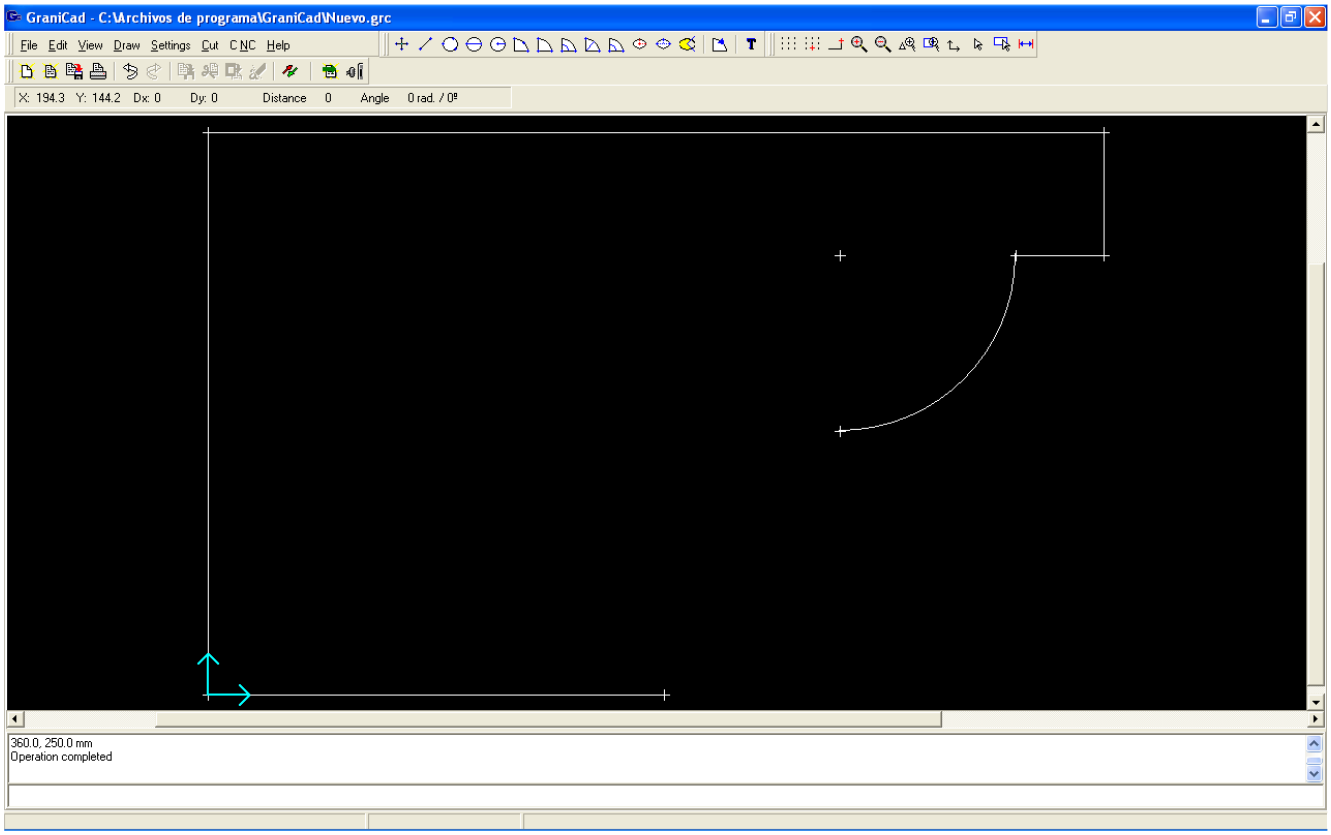

Figure 142

The next step will be drawing the horizontal 50 mm line.

Click with the mouse on the command line:

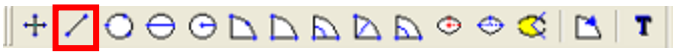

Figure 143

you will be asked to indicate the first point of the line, and with the mouse clicking on the first point since the reference object allows us to indicate with precision. You may also be indicated by typing text as the starting point 360,150, which corresponds to that point in absolute coordinates. It asks for the end-point line, and he indicated by coordinates on @ 0,-50 or R0, -50, or by absolute coordinates 360,100. Press ENTER twice to validate the command and end the command line. It will be shown on the screen.

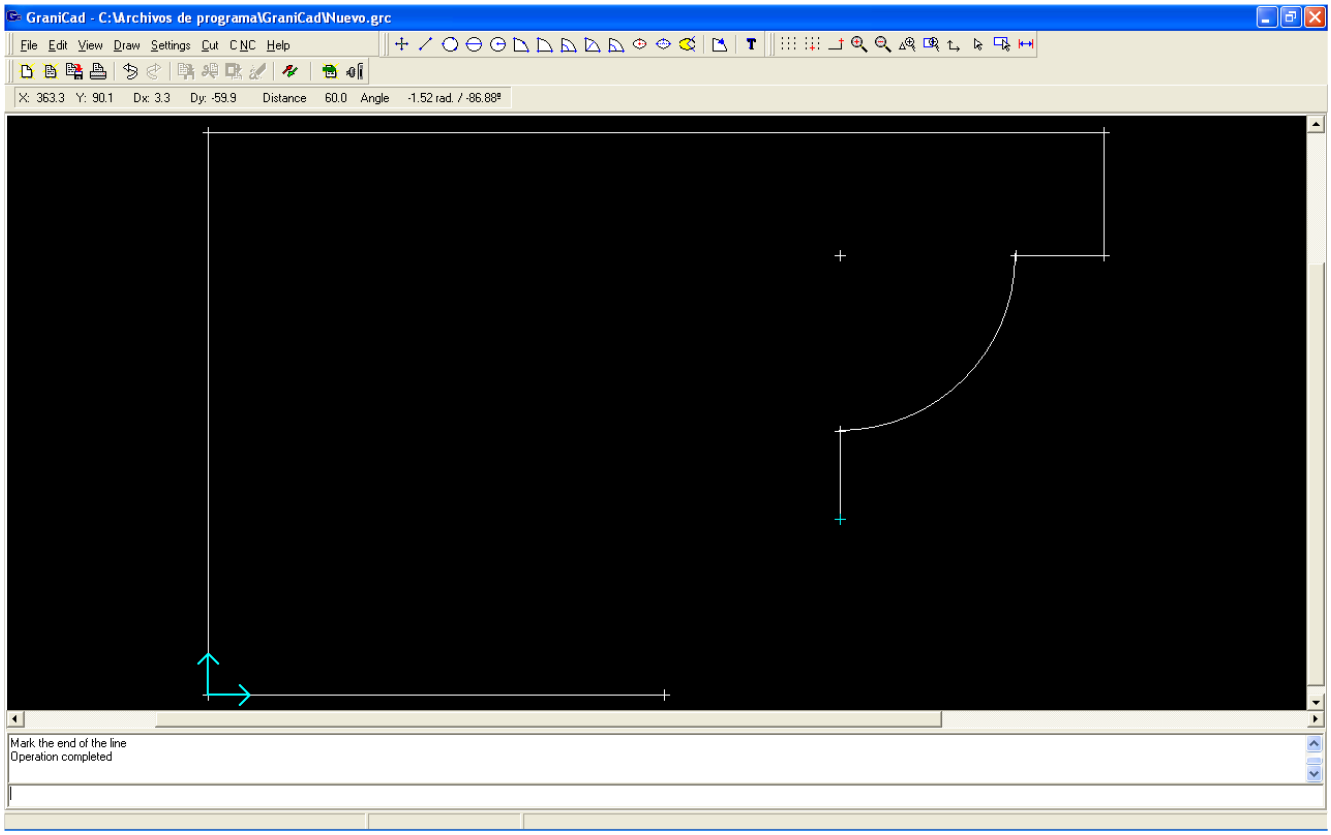

Figure 144

Finally, you draw the final bow.

You note that the button is arc anticlockwise direction, and if not select it.

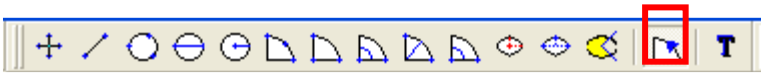

Figure 145

Click on the icon of ARCO IFC (since you know the coordinates of the starting point, end and centre):

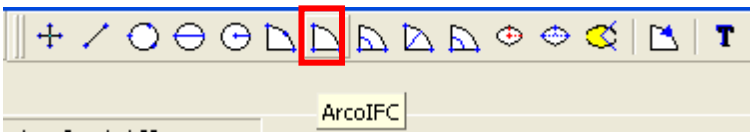

Figure 146

or write in the text-line ARCIFC or ARCOIFC. You will be asked to indicate the first point of the bow, and with the mouse clicking on the first point since the reference object allows us to

indicate with precision. You may also indicate it by typing text as the starting point Section 360,100, which is corresponding to that point in absolute coordinates.

It asks for the endpoint of the arc, and you indicate with the mouse as the reference object allowing us to do it accurately or by absolute coordinates 260, 0. Finally, you determine the centre, which you indicate with absolute coordinates as 360, 0.

Press ENTER to validate the command and what you draw on the screen.

You have the complete picture of the piece.

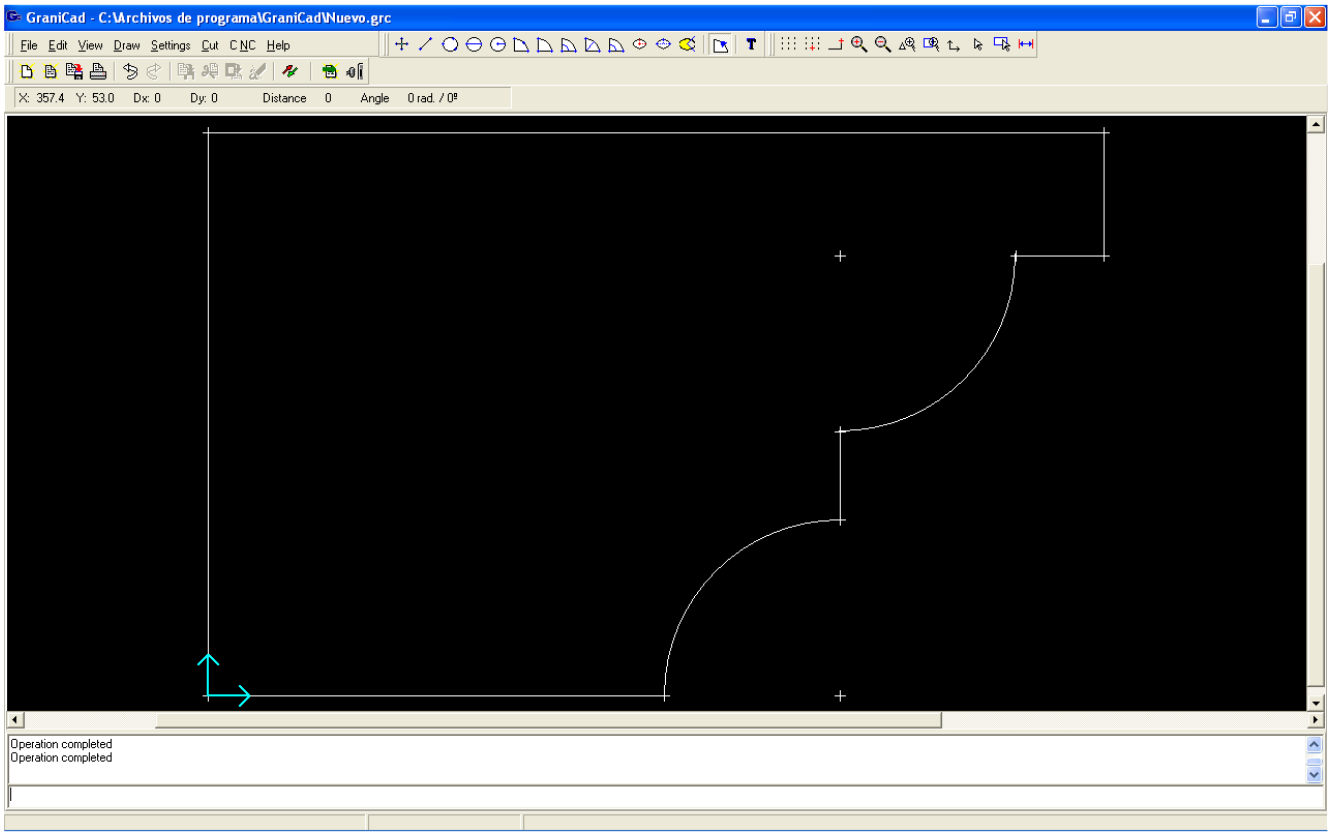

Figure 147

5.2. Drawing the Block EntryWay

You have to draw a driveway because the stone blocks are not usually pre-cut and do not serve us any side initially.

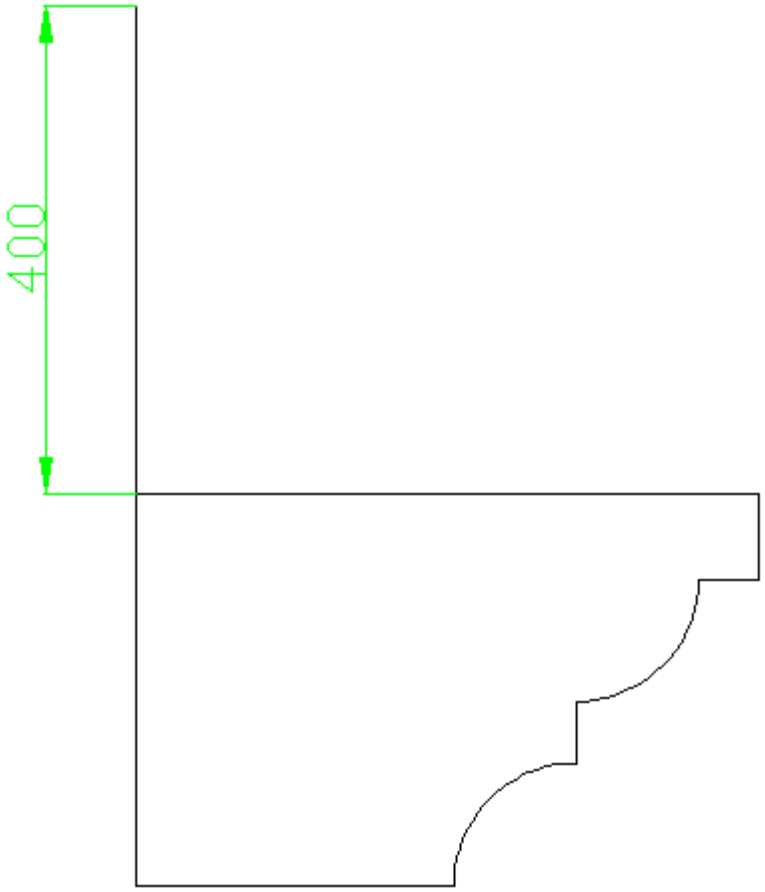

Figure 148

You are going to draw an entry line of 400 mm long.

Click with the mouse on the command line:

Figure 149

write in the text-line LIN or LINEA. You will be asked to indicate the first point of the line, and with the mouse clicking on the first point, since the reference object allows us to approximate accurately indicate the mouse to the upper left corner of our design.

He asks the end-point line, and he indicated by relative coordinates **@ 0,400** or **R0, 400**. Press ENTER twice to validate the command and end the command line. It will be shown on the screen.

5.3. Locate the Drawing in the Origen

You have to move all the designs, so the point that you take as reference (where you will do a PRESELECT of zeros with our CNC) to displace the point 0, 0 in absolute coordinates of GraniCAD.

You will take as a reference point the one indicated in the figure below:

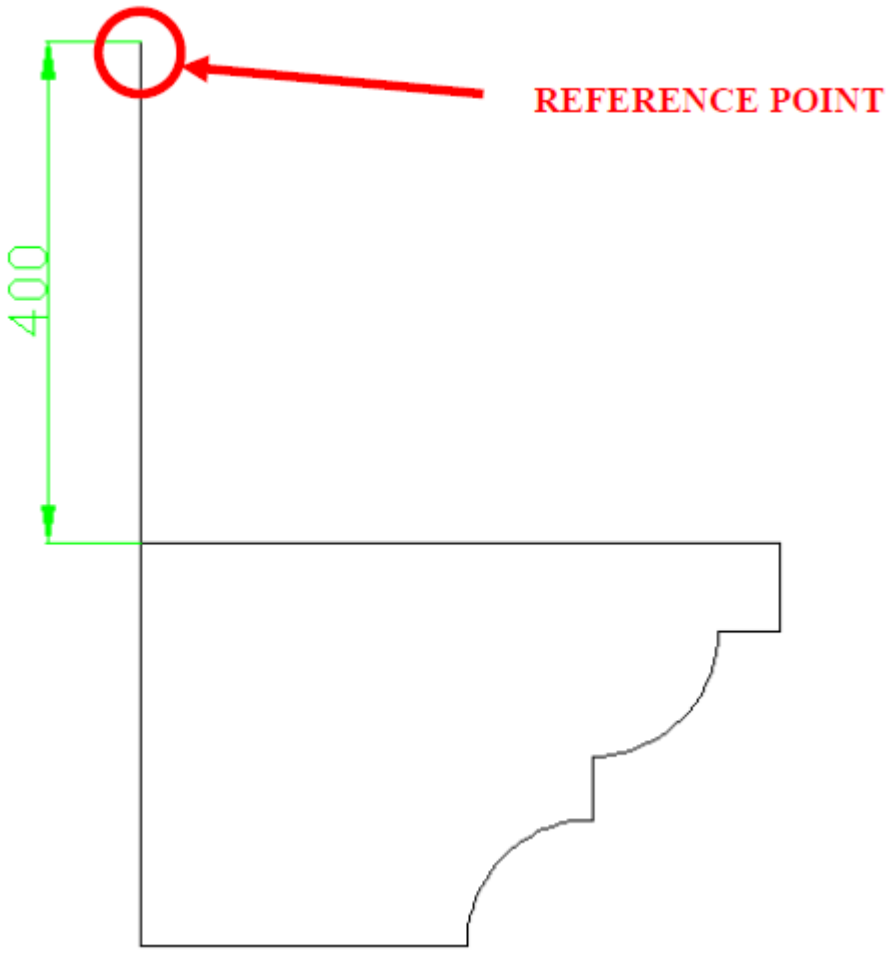

Figure 150

The steps in GraniCAD are:

- Select the entire drawing. This can be done from the menu EDIT-SELECT ALL

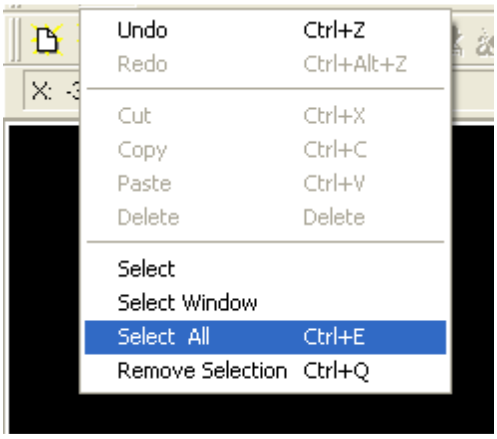

Figure 151

Or using the keystroke shortcut **CTRL+E**.

- Select in the menu MOVE - DRAW

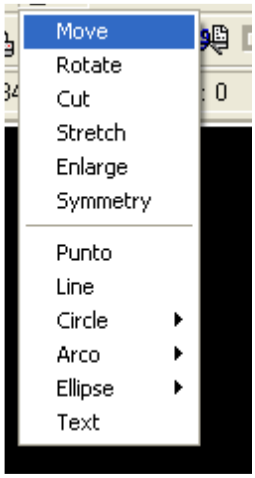

Figure 152

It asks us to indicate the origin point by using the mouse point as our reference point.

It asks to indicate the destination and type **0, 0**. Press ENTER.

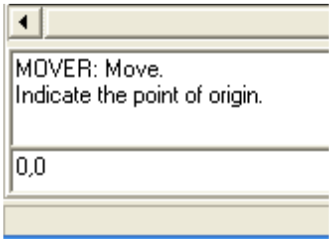

Figure 153

The program has moved the entire figure so that now our reference point is located at the origin point of the program (0.0).

5.4. Generate the Cutting Way

Press the button to generate the cutting way:

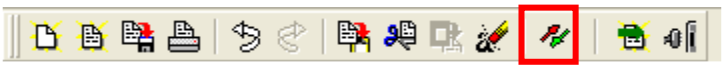

Figure 154

or access it through CUT-MAKE OR CHANGE THE TRACE.

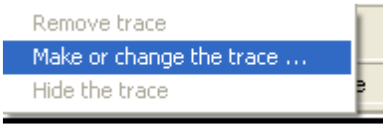

Figure 155

You will see the next screen:

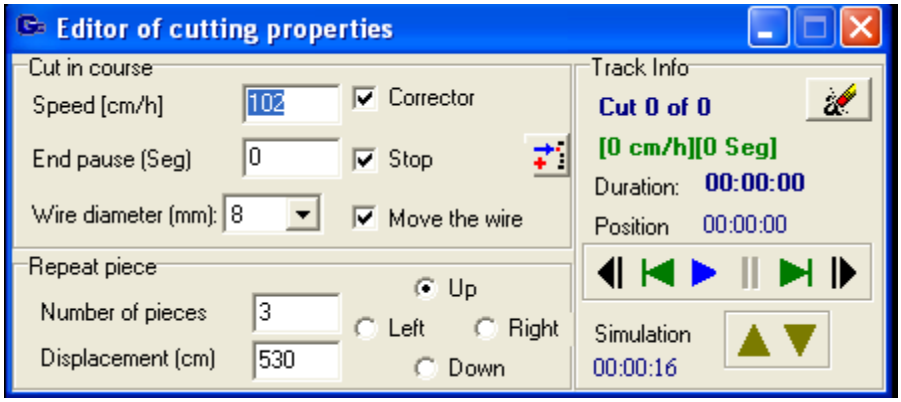

Figure 156

Activate the box on the CORRECTOR, because in this piece you want to offset the wire radius and thus give us the exact measurements which you are adding.

It asks us to indicate the starting point: select with the mouse our point of reference.

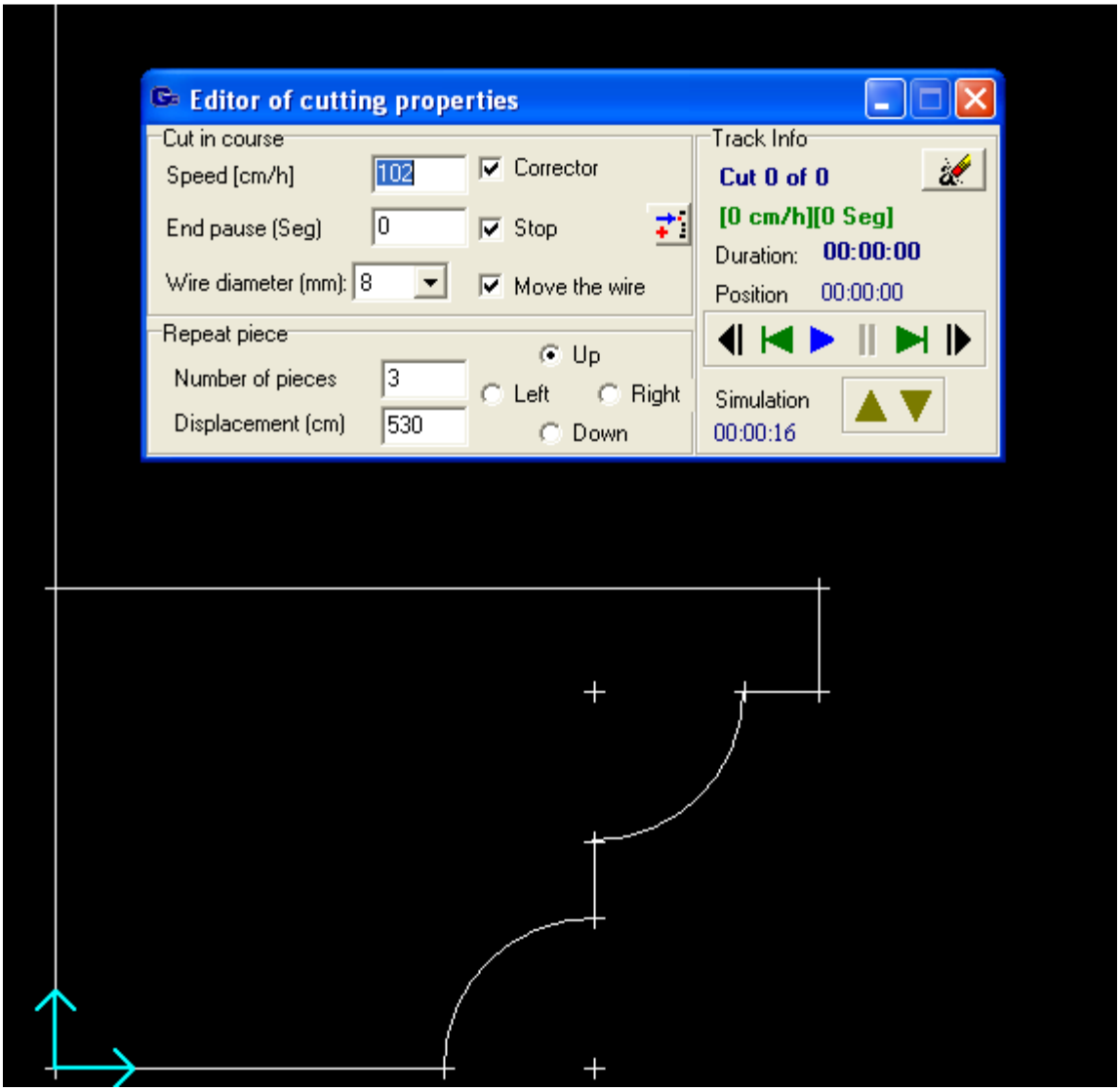

Figure 157

Then you select the sections according to the route you want to describe the wire and indicate the side (after you select a path you click the mouse on the side of the line that interests us). Once you have done this, you get something like follows:

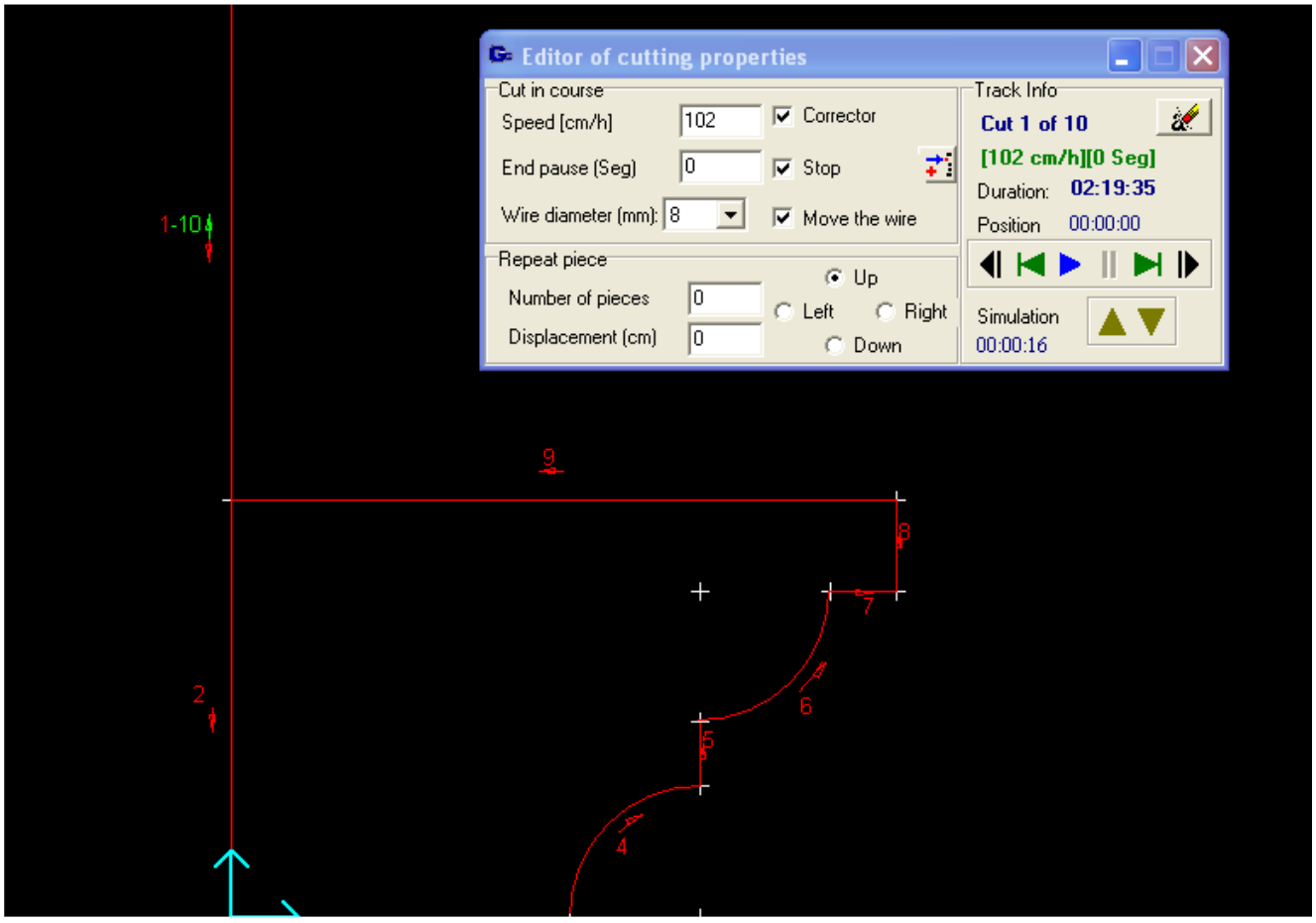

Figure 158

Then edit each section of the way.

You started on section 1, just click on it:

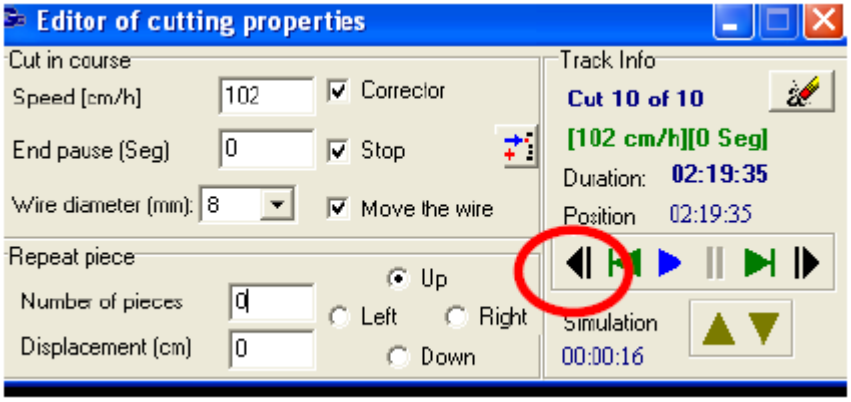

Figure 159

5.4.1. Trace 1

The stretch of approximation. Configure the editor as shown in the figure below:

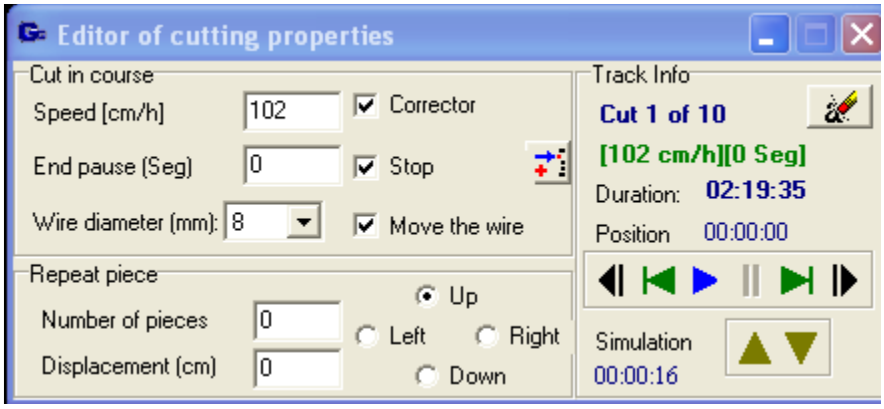

Figure 160

(You have set a wire diameter of 8 mm which is supposedly going to make this piece as an example. The wire diameter is always the same for all sections) (The advanced speed rise to 102 cm / h will be the same for all sections of our example, although in practice it can be changed for each part).

(You don't make pause at the end of this section, because there is no change of direction).

(No disable the MOVE WIRE, although it could be off to get some time, this option is not recommended because it can cause some damage in the corners of the parts when starting another section).

You upload the parameters you set for this section by clicking the appropriate button below:

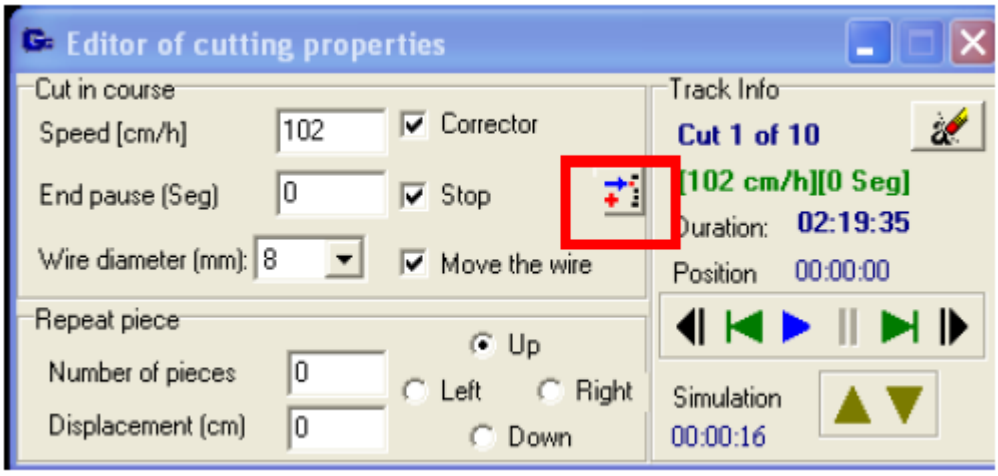

Figure 161

Then you jump into the next trace by clicking on this button.

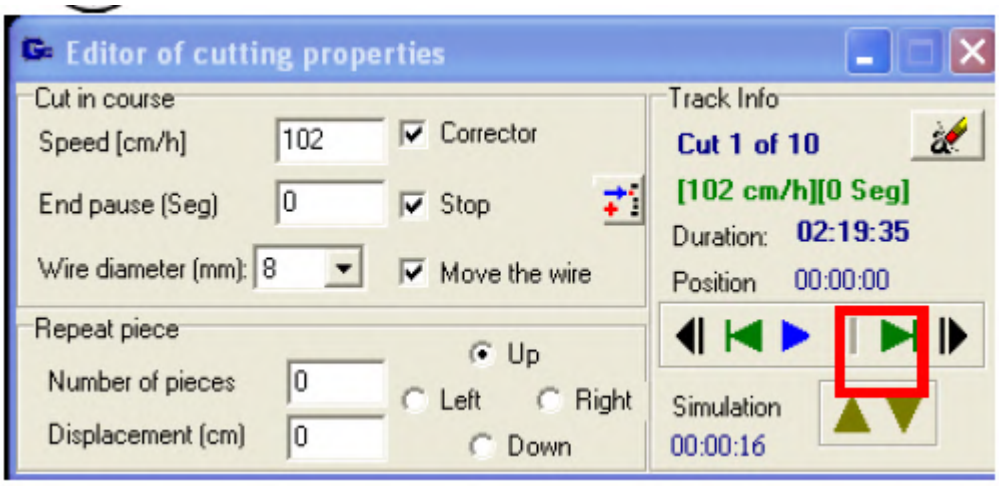

Figure 162

5.4.2. Section 2

Configure the editor as shown in the figure below:

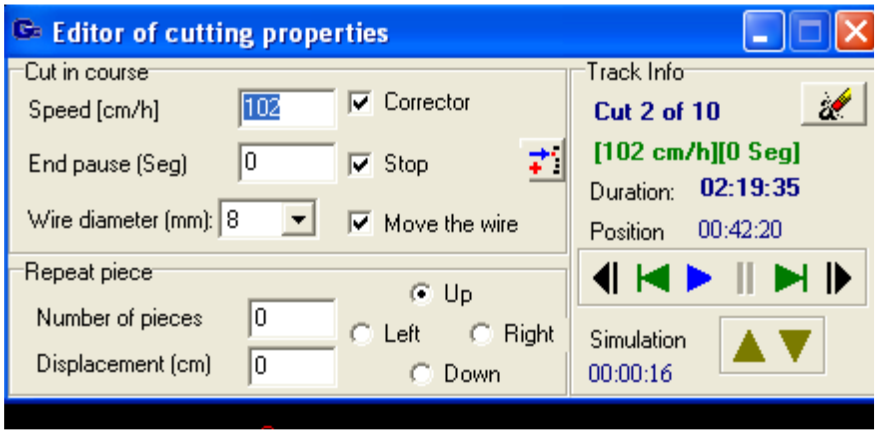

Figure 163

The change made from the previous screen is that you have added a pause of 300 sec since there is a change of direction and you must wait to clear the shaft formed by the wire.

You upload the properties of the section by clicking on the button shown.

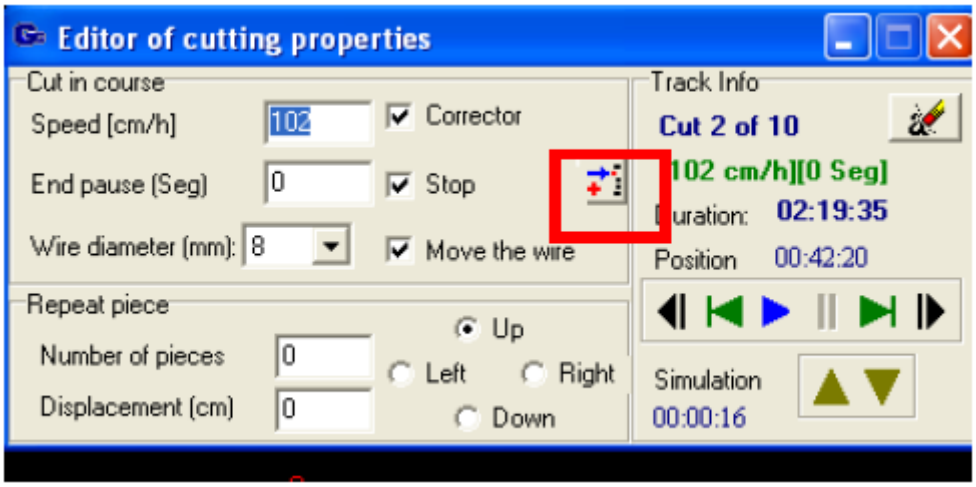

Figure 164

Move to the next trace by clicking on this button.

Figure 165

5.4.3. Traces from 3 to 9

All other sections will apply the same parameters as for section 2.

You have generated a full cutting path and you can close the editor.

5.5. Generate the CNC File

Press the button:

Figure 166

or access through CNC-GENERATE CNC FILE…

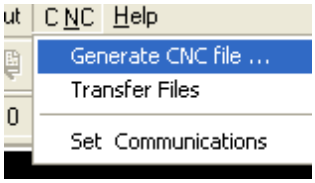

Figure 167

It displays the CNC file created from the cutting trace you have defined in the previous steps:

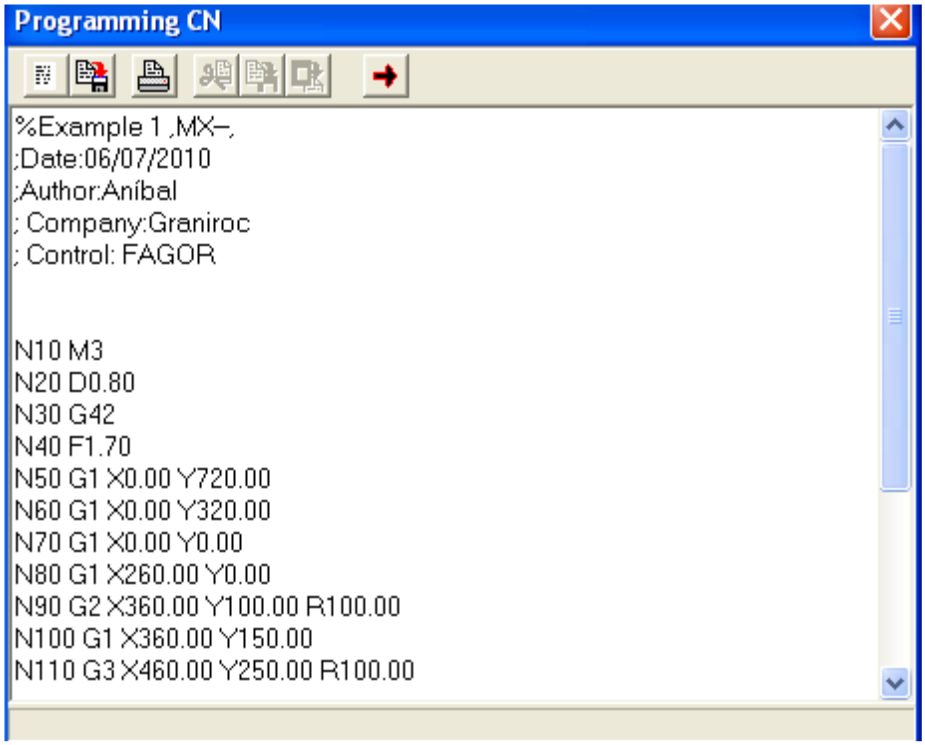

Figure 168

The next step is to save the generated file. To do this you click on the button remarked:

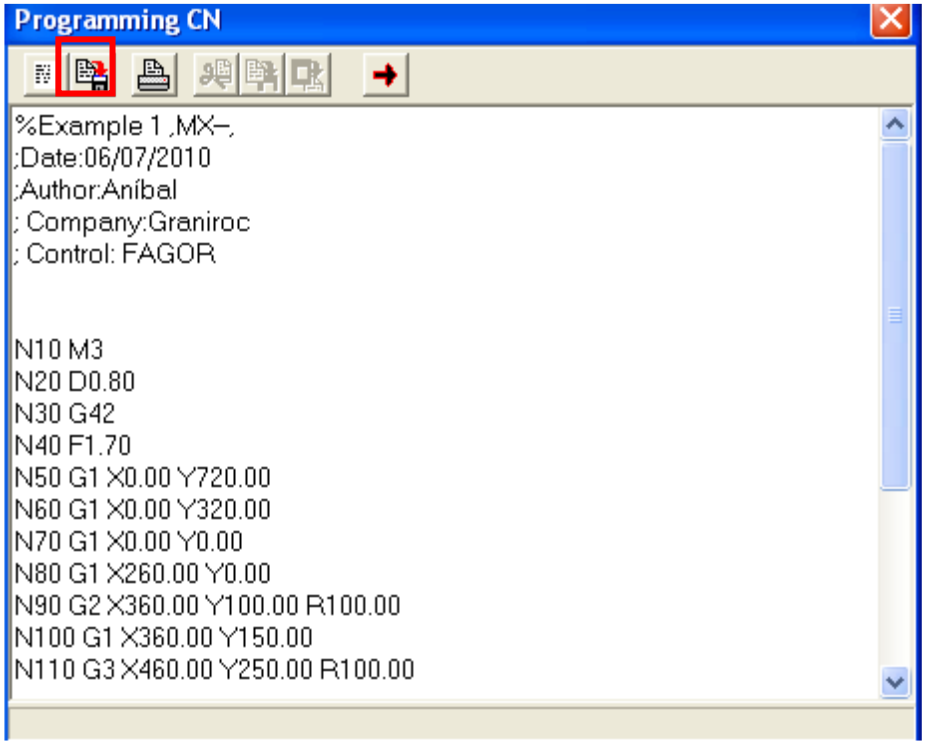

Figure 169

And it opens a window with a dialogue box to specify where you want to save the .pim file you have generated:

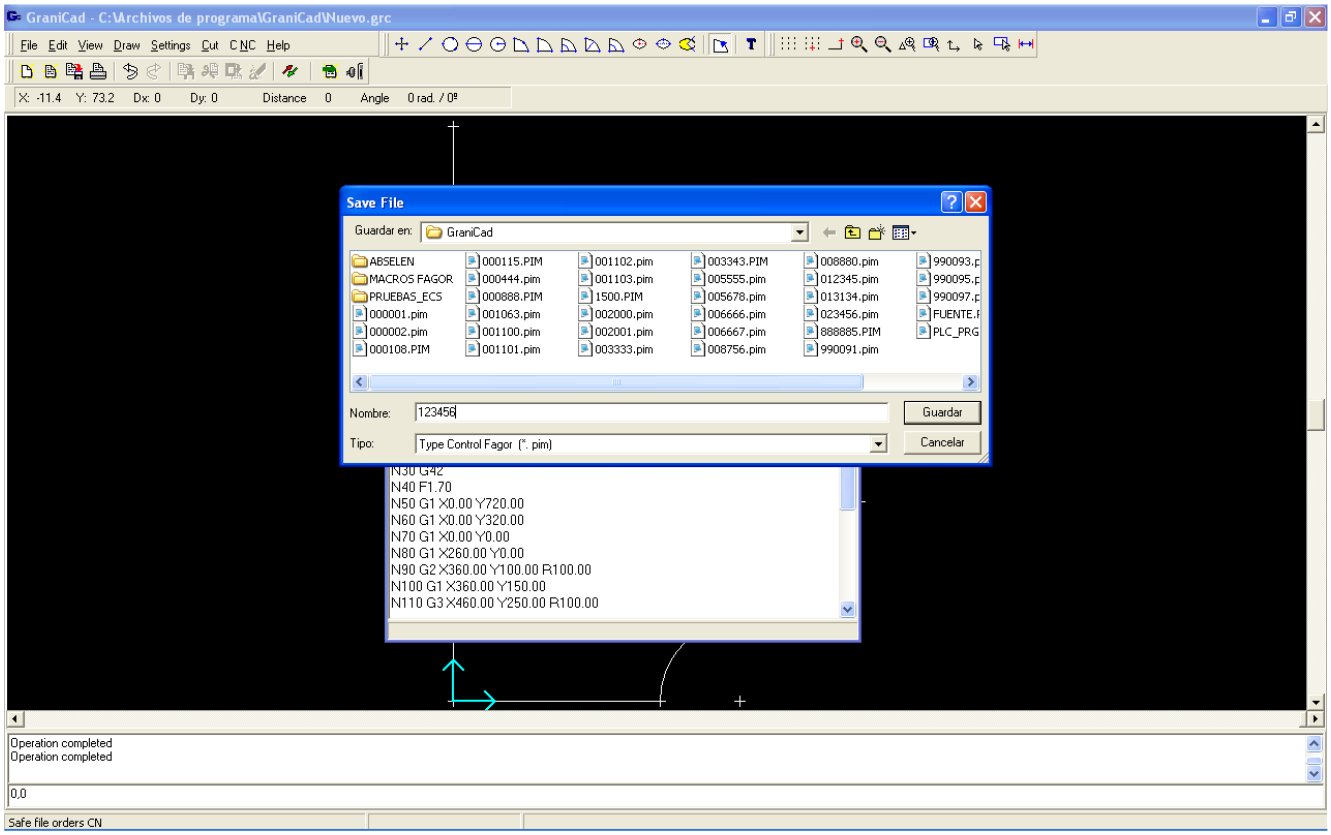

Figure 170

You save it in the desired direction.

EVERYTHING IS READY TO TRANSFER THE PROGRAMME TO OUR CNC AND STAR WITH THE PROCESS OF CUTTING.

Now you will discuss the design of the previous piece, but considering that you do several repetitions. The steps are the same, except those relating to the generation of the cutting path are the ones that are going to be studied here.

5.6. Generate the Cut Profile

Press this button to generate the cutting profile:

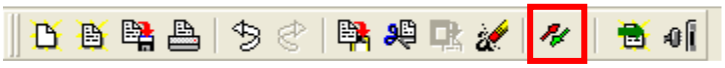

Figure 171

or accessing it through CUT- MAKE OR CHANGE THE TRACE.

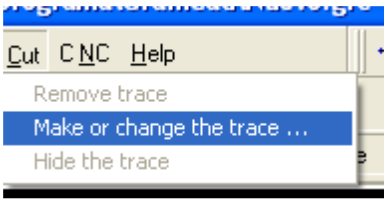

Figure 172

The editor of cutting properties gets opened.

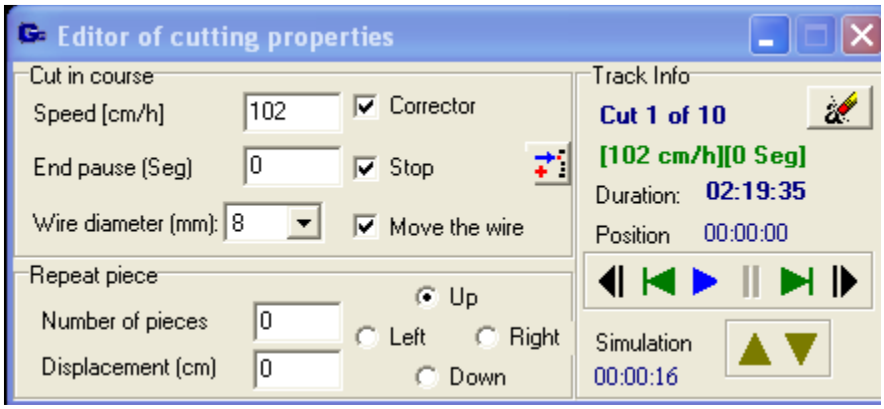

Figure 173

Activate the box corrector, because in this piece you want to offset the wire radius and thus give us the exact measurements which you are adding.

Indicate the number of repetitions you want to do, the movement and the meaning. In our case, you believe that you will perform three pieces with a displacement of 530mm between parts and to the left.

The editor shall be as follows:

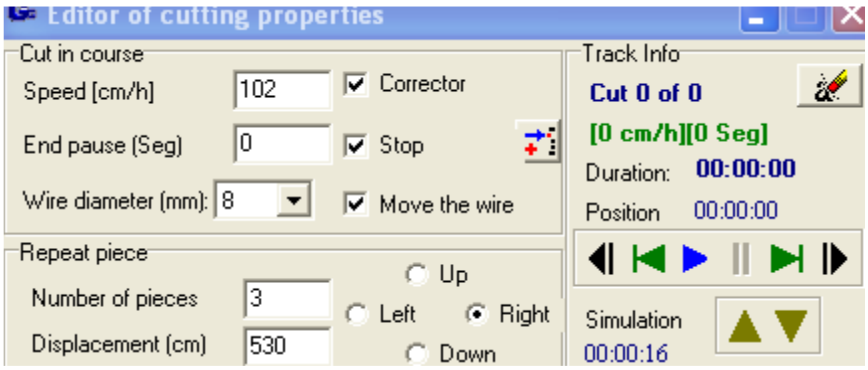

Figure 174

It asks us to indicate the starting point: select with the mouse our reference point.

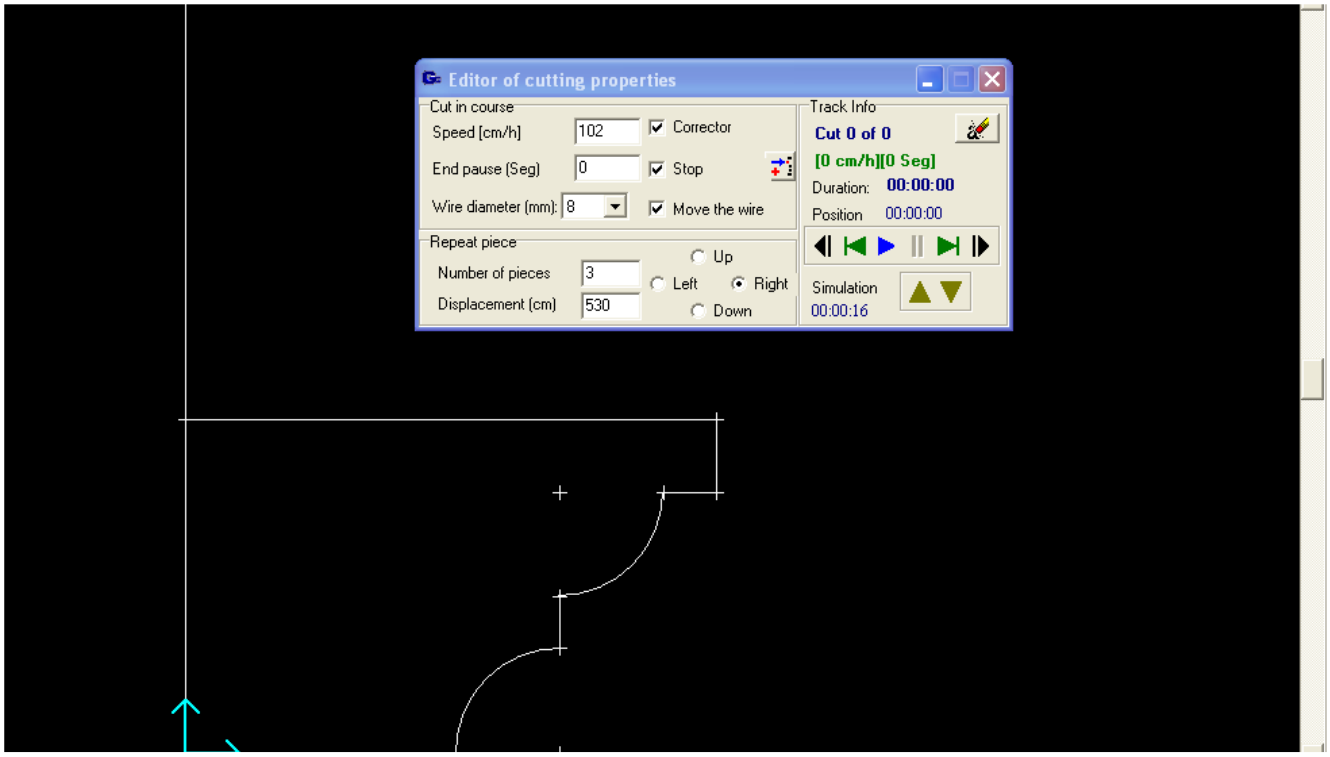

Figure 175

Then you select the sections according to the trace you want to describe the wire and indicate the side (after you select a path you click the mouse on the side of the line that interests us). Once you have done this it appears something as follows:

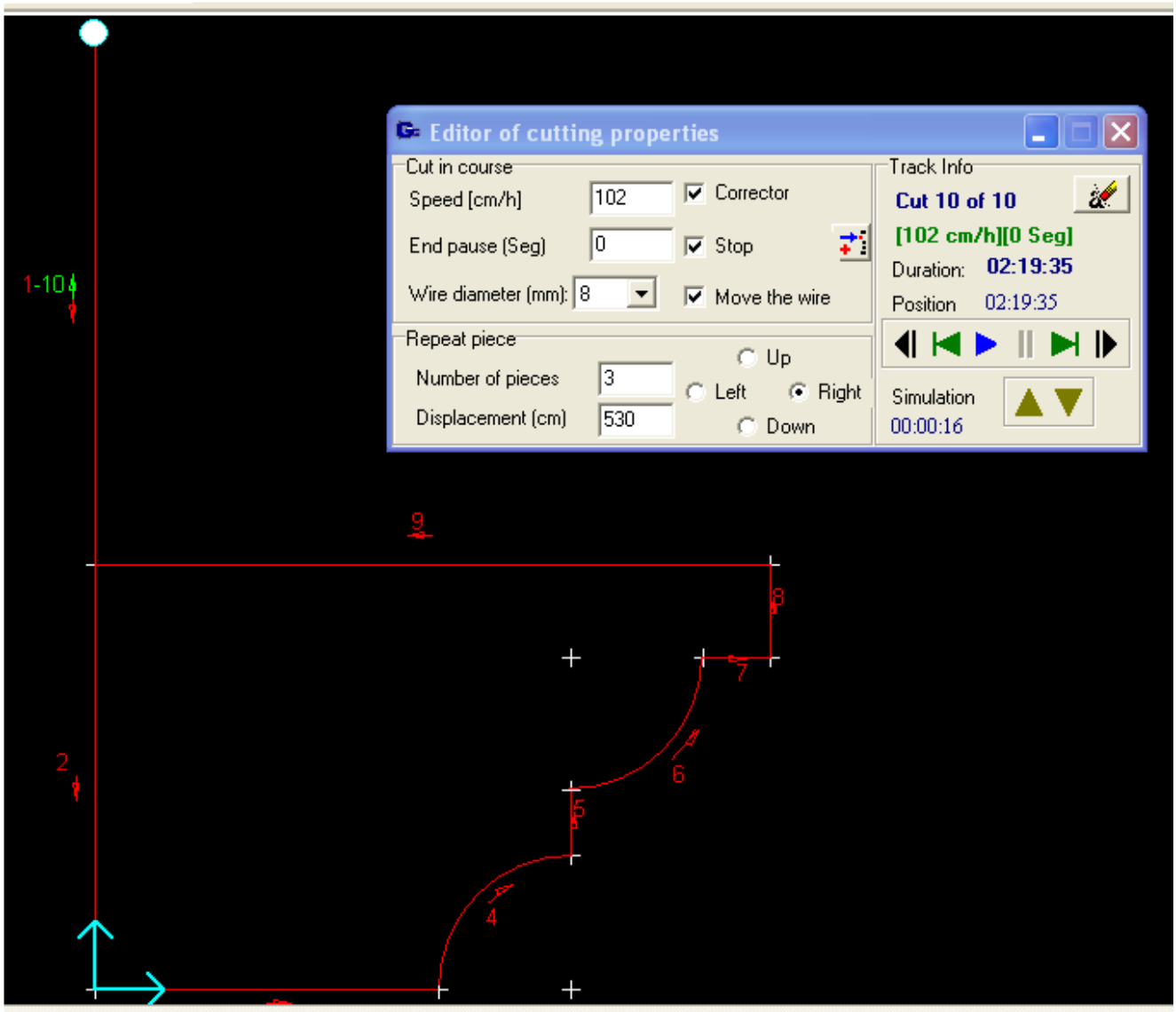

Figure 176

Then you will edit each section of the trace.

To start with section 1, just click the button:

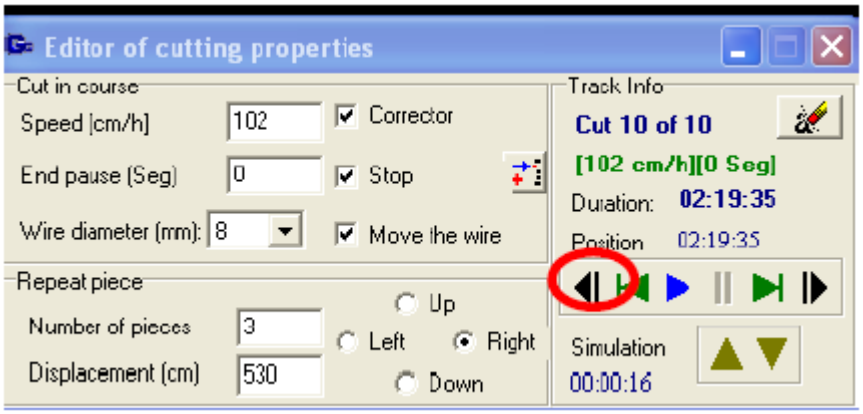

Figure 177

5.6.1. Trace 1

The stretch of approximation. Configure the editor as shown in the figure below:

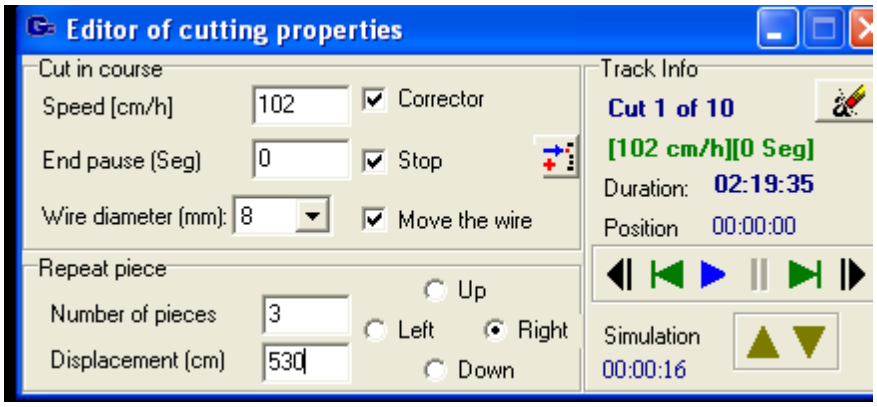

Figure 178

(You have set a wire diameter of 8 mm which is supposedly going to make this piece as an example. The wire diameter is always the same for all sections).

(For the speed of advance look at the 102 cm / h and leave the case for all sections of our example, although in practice can be changed for each leg).

(Do not pause at the end of this section, since there is no change of address).

(No mark the MOVE WIRE, although it could be off to buy some time, not recommended because when starting on the next leg on the corner of our room, you can leave it checked).

You pass the parameters you set for this section by clicking the appropriate button below:

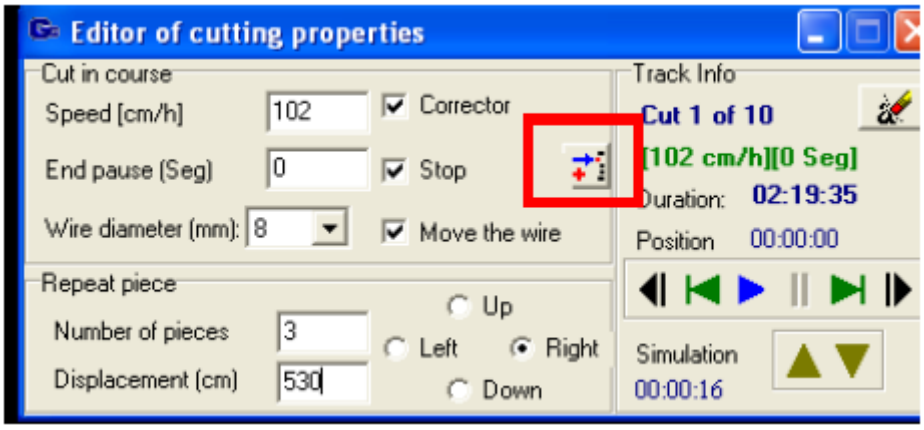

Figure 179

You go to the next section by clicking the appropriate button.

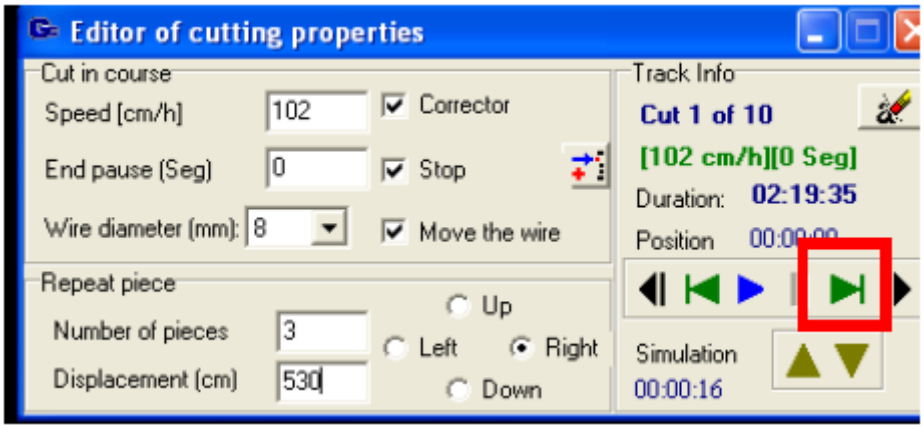

Figure 180

5.6.2. Trace 2

Configure the editor as shown in the figure below:

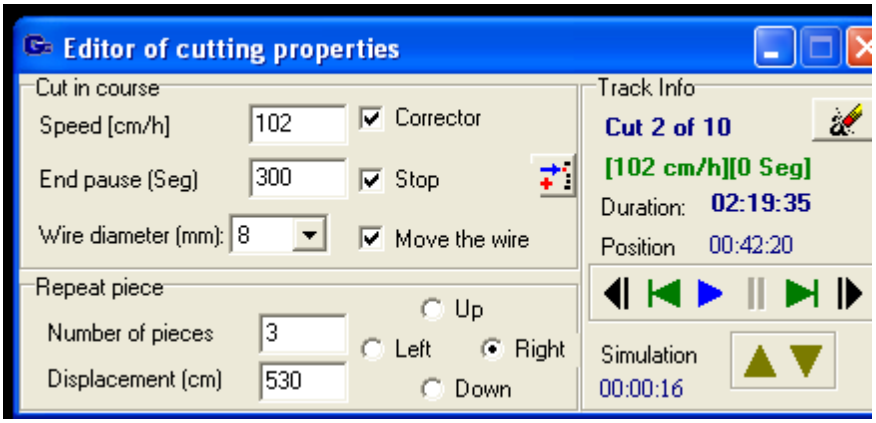

Figure 181

The only thing that changed from the previous instalment is that we have added a pause of 300 sec since there is a change of direction and we must wait to clear the rope formed by the thread.

We turn to the section properties by clicking on the button shown:

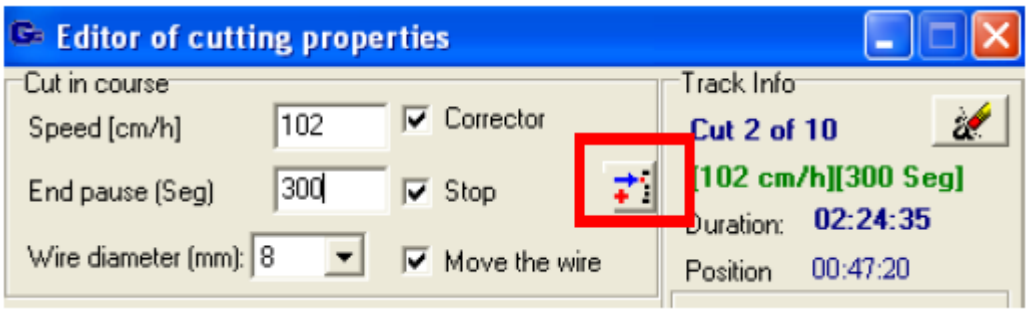

Figure 182

For the next step press the following button:

Figure 183

5.6.3. Trace 3 to 9

For other sections, except section 10, apply the same parameters as for section 2.

5.6.4. Trace 10

The output section of the stone is to be placed at the beginning of the next repetition. In this section, you can stop the thread to not cut up as it goes through a stretch already cut and waiting to give him time to save time in performing the pieces. The editor for this section is as follows:

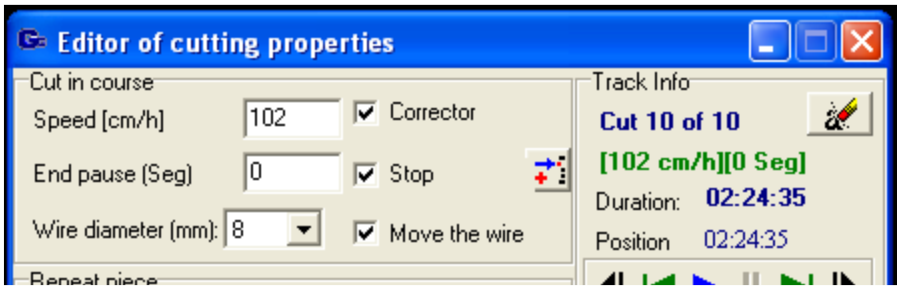

Figure 184

You have generated a full cutting path and you can close the editor of the cutting path.

For more information contact us at

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or

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