

**MDM Metrosoft S.a.s.**  
di DENI Ing. M. & C.

USERS GUIDE

# Metrospline

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# PROFILE GENERATIONS

## INTRODUCTION

The METROSPLINE program is used to the bidimensional measurement of mechanical workpieces with NOT defined geometry, as workpieces with profiles not constituted by elementary geometric elements like a: Circles, Arcs of circle and segments

From the measurement, it is possible to calculate the "MATHEMATICAL MODEL" for the probed workpiece, that is the whole points that constitutes the profile and that gives exact indications about the shape and size.

METROSPLINE purposes are:

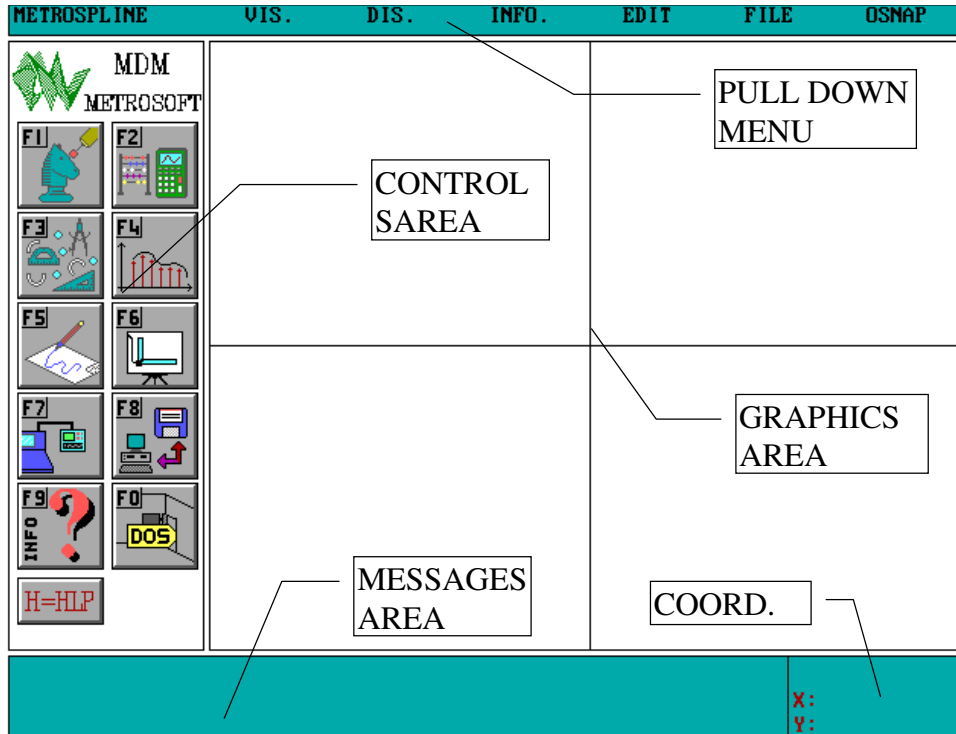
- Location of an unknown profile.
- The comparison of a probed profile with the correspondent sample profile, that could be obtained from a CAD file in the DXF format.
- To extract the program for the C.N.C. which machines the workpiece.

METROSPLINE contains some CAD functions, with which it is possible the EDITING (Creation) of designs, for example the design of geometric entities as circles, arcs of circle and straight lines. The created design could be saved and converted in the CAD format or in the CNC (ISO) format.

The menu is displayed with IMAGES (ICONS), that give an easy and immediate understanding of the function that they represent, with an HELP which shortly describes the functions contained in the menu.

To select the items of menu can be used the MOUSE or the keyboard, pressing the key correspondent at the wanted command; METROSPLINE is used by the people not expert in the use of computer.

## VIDEO PRESENTATION



On the illustration above, we see how METROSPLINE is displayed on video. On the screen there are different areas.

Every area has a specific meaning and function :

- **CONTROL AREA** : here are displayed the MENUES and SUBMENUES.
- **MESSAGES AREA**: interactive area, destined to the communication between the user and the computer, where are displayed the user's messages and the requests of input data.
- **GRAPHICS AREA**: here are displayed the profiles.
- **COORD. AREA**: in this Area are displayed the coordinates corrispondent to the position of the mouse arrow on the profile.
- **PULL DOWN MENU AREA** : destined to profiles editing; the menu is available only after main menu activation.

## COMMAND SELECTION AND MOUSE USE

- **CONTROL AREA**

There are displayed the different **MENUES** and **SUBMENUES** with a number **N** of options that changes according to the specific **MENU**.

Every option is characterized by :

- by the drawing of the key
- by the icon corresponding to the function
- by the label of the key that selects it

The active keys are the keys function from **F1** to **F10** (this last labeled "**F0**"), on the drawing of the key, the key "**H**" which activates the generic **HELP** and the **ESC** key, activated only in the submenus with the function of cancel the last command and to go back to the preceeding phase.

It is possible also to activate a specific **HELP** regarding the options to be known, pushing in the same time the **CONTROL** key (**CTRL**) and the corresponding function key of the command.

- **PULL DOWN MENU THROUGH KEYBOARD**

To select one of the pull down menu it is sufficient press the **ALT** key + the letter highlighted of the menu item that you want to use. It will be displayed a window containing the items of the corresponding submenu. To select an item move cursor with the keys over the wanted item, which will appear in reverse and press **ENTER**. Using the **RIGHT** and **LEFT** arrows are available the adjacent menus.

- **COMMANDS THROUGH MOUSE**

Two **POINTERS** are associated to the mouse; they characterize the position on the screen .

and one of the two pointers is activated according to the area video which is pointing.

The first is displayed as an arrow and is activated in case of commands selection or of **PULL DOWN MENU** enable.

The second is displayed as a cross and is activated when must be selected a zone of the graphic area.

For the selection of a command through the mouse it is necessary to move the arrow over the drawing of the desired command and press the **LEFT** button; for activating the **HELP** of that command press the **RIGHT** button.

## MAIN MENU



### 1. SPLINE MEASUREMENT.

This key loads MIX program, which enables all the options regarding the measurement of the spline:

- Probes Calibration
- Reference System
- Points Measurement (See in MIX USER'S GUIDE the option POINT MEASUREMENT). After the measurement, the coordinates of the sensor's center in successive positions will be memorized in "PROBED POINTS FILE"..



### 2. PROFILE ELABORATION

Through a complex calculation, as we will see later, the ELABORATION of these points (sensor's center) will give points of the compensated profile; these points will be memorized in the "Compensated File".

First of all will be required :

"PROBED POINTS FILE" NAME

To select this file is sufficient to choose it in the list of file displayed (see chapter "File selection").

"COMPENSATED FILE" NAME

Is the name that we want assign to the file where will be stored the points of the profile at the end of the elaboration. For this file will be proposed of default the name of the "Probed points file" previously selected. If wanted, a different name can be typed by keyboard.

PROFILE PROJECTION PLANE

May be the plane of the workpiece reference system on which is projected the profile that we want calculate; in this phase will be displayed the projection of profile on each of the three coordinate planes, so that the user can make the correct choice.

The profile could be projected over a plane that is not parallel to one of the three coordinated planes; in this case as plane of visualization will be chosen that one.

APPROXIMATION TOLERANCE

Is another parameter of the profile calculation algorithm, that denotes the maximum distance admitted between a probed point and the point of the calculated profile nearer to it.

Usually to this parameter is assigned, through keyboard, a value included between 0,001 mm. and 0,100 mm.( 0,10 mm of default), according to the desired fidelity. Tolerance value is selected considering that the profile is calculated through

"following approximations" and, consequently, that it influences the time of calculation (time increases if tolerance value decreases, and vice versa).

#### CORDAL ERROR TOLERANCE

Is a parameter that, at the end of elaboration permits the elimination of all the points that haven't importance for the profile identification; the parameter must be selected with a number in micrometers ( 5 micrometers of default).

After this selection METROSPLINE, will realize a complex algorithm including :

- calculus of the 'sensor's center profile' approximating (see Tolerance parameter) the points (sensor' s center) previously probed.
- individuation of points on the the 'senso r's center profile', their compensation of the sensor' s radius with calculation of the corrispondent compensated points.
- calculus of a profile containing points previously compensated and individuation of a number of significative points on this profile.
- successive elimination among preceding points of all the points that don't give an adjunctive information for the profile identification (see Cordal error tolerance parameter). The result of this calculation will be a serie of points that will be memorized in a File named 'Compensated File'.

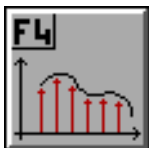
Obviously these points will represent the probed points as better as lower are the values selected for TOLERANCE and for CORDAL ERROR TOLERANCE.

In any moment it is possible to abort the operation of elaboration through the ESC key, that terminates the command and returns to the main menu.



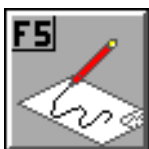
#### 3. GEOMETRIC TRANSFORMATIONS.

It could be necessary to make geometric operations on the compensated profile. At page 7 will be examined single transformations.



#### 4. DIGITIZINGS

The possibility of digitizing a compensated profile is a very powerful performance of METROSPLINE, because it allows quantitative evaluation of the profile considered alone, or referred to a teoric profile. At page 8 will be examined single possibilities.



#### 5. PROFILE HARDCOPY.

It allows the copy of a profile on printing or plotter.



## 6. CONVERSION TO & FROM CAD

It allows the conversion of points or arcs of circle into DXF or IGES format and the conversion of a DXF profile into the compatible METROSPLINE format.



## 7. CONVERSION TO C.N.C.

It allows the conversion of a profile from the METROSPLINE format to ISO format (language of programming for the numerical controls).

In this phase is open a request window for the parameters that they need to the conversion:

- type of file to convert, this could be a file with single points or a profile constituted by arcs of circle and straight line (see points 3.4.). In case the file of arcs has not been created, the file of points will be automatically converted and the user will be only notified of this choice.
- type arcs format for the control: as coordinates of the center (I, J), or as radius (R).
- unit adopted for measurement (it could be millimeters or cents of millimeter, according to the type of control to use).
- wanted maximum number after comma.
- numbers of first program block.
- step between the blocks, that is the STEP that you want have between two blocks consecutive.
- label for Axis coordinates; it is given the possibility to change the letter that identifies the single Axis.

Now it is created a file with extension ".CNC," with the PART-PROGRAM for the Numerical Control containing the only geometry, tightly connected to the profile. The technological part, machine setup, tool selection, etc., must be introduced later.



## 8. OPERATIONS ON FILE

- Enables a submenu for the Files management : Load, Save and Delete .



## 9. INFO

Opens a window in the center of the screen, where appear informations on the File loaded, i.e. the name, the number of points of the profile and the plane of elaboration of the profile.



#### 10. EXIT TO DOS

Ends the session and returns to the operative system.

### 3 - GEOMETRIC TRANSFORMATIONS



#### 3.1. POINTS ROTATION

It allows the rotation of all the points of the original profile around the origin of the reference system. Will be required the rotation angle (from 0 to 360 G.). The rotation is intended only in sense counterclockwise; it is not therefore admitted an angle with negative sign. Therefore, if the profile must be rotated of -10, the angle to insert will be:  $360 - 10 = 350$ .



#### 3.2. POINTS TRANSLATION

It allows to shift the points of the original profile in the direction of ONE or BOTH the Axis of the selected plane. It is required the offset on every Axis, which could have positive or negative sign according to the direction of translation.



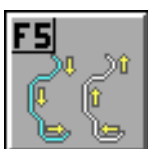
#### 3.3. MIRROR FILE

It creates a profile specular in respect of the original. The specularity can be calculated in respect of a single axis or both them. It is required the axis of specularity, pressing ESC in this phase the command is canceled and return to the menu of the geometric transformations.



#### 3.4. TRANSFORMATION TO ARCS

It transforms the original profile composed by points in a profile composed by arcs of circles and straight lines. Transformed profile will approximate the points of the original inside a tolerance selected from keyboard, with or without common tangency. Selecting the option "tangent arcs", the profile should be obtained through the elaboration, without following approximation (in this case it is necessary that is still existing the "Probed points file").



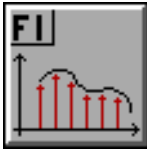
#### 3.5. FILE INVERSION

It changes the path sense of the original profile (the direction from the first point to the last) ; this function is very useful in

phase of comparison of profiles, because two profiles may be compared if they have the same path sense.

For items F1, F2, F3 will be required the name for the transformed profile; if a profile with that name already exist, it can be overwritten. Also the original profile can be replaced in case it is not necessary to hold it in archive. At the end of the operations it is possible to hold in memory or the transformed or the original profile.

## DIGITIZINGS



### 4.1. ABSCISSA DIGITIZING



### 4.2. ORDINATE DIGITIZING:

The profile is digitized in CARTESIAN coordinates, over one of the selected Axis. There will be a master coordinate for which will be selected the limit values and the increasing step from keyboard and a slave coordinate calculated in correspondence of the master coordinates.



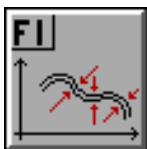
### 4.3. POLAR DIGITIZING:

A point is expressed in polar coordinates:

- module = distance point-origin
- angle = angle of straight line point-origin with the axis of the reference system

Digitizing steps are calculated in function of the angle; for every angle will be displayed and printed the corresponding point.

Choosing anyone of the items above described, it will be enabled another SUBMENU to select the area of the profile to be digitized:



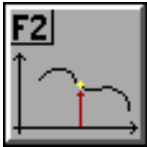
### 4.X.1. PROFILE COMPARISON

It is possible to compare a compensated profile (represented by a "COMPENSATED FILE") with a sample profile as :

- a compensated profile previously calculated from the measurement of a sample.
- a profile previously imported from a CAD system.

Input the compensated profile to be compared and after the sample profile; the two profiles will be displayed together with

two different colours, Red for the sample profile and Black for the other one. If the two arrows are discordant it will be necessary invert the sense of one of the two file ( ref. 3.5.).



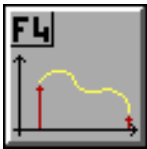
#### 4.X.2. SINGLE COORDINATE

It allows to print the values correspondent to a single point, in the case of Cartesian coordinates, or to a single angle in case of polar coordinates.



#### 4.X.3. REDUCED RANGE:

It performs the digitizing of a portion of profile, located through a pointer that will be displayed after the selection of this option. METROSPLINE asks the beginning and the end of the portion of profile to digitize, respectively with the question FIRST POINT and SECOND POINT.



#### 4.X.4. COMPLETE RANGE:

It performs the digitizing of the complete profile.

The selection of the point or of the profile portion to digitize is done through a pointing system with cross derived by MOUSE for the selection over the profile. In cases 4.1) & 4.2), the value or the range can be digitized through KEYBOARD having previously pressed TAB key. In this case will be displayed the limits within which must be the values to digitize In this phase :

- command is cancelled pressing ESC or the RIGHT key of the mouse
- the value is confirmed pressing ENTER or the LEFT key of the mouse.

The coordinates (polar or Cartesian coordinates according to the selection) of the mouse cursor can be read in the dedicated section, in low right zone of the screen.

Selecting the comparison or the digitizing in a ‘reduced range’, gets particular importance the sequence of the beginning & ending digitizing points.

In case of digitizing by Cartesian coordinates this order is irrelevant.

In case of polar coordinates the initial and final angle must be selected in ‘counterclockwise’ sense, independently by the profile route sense, otherwise METROSPLINE digitizes all the profile excluding the selected zone.

After selection of the range to digitize, METROSPLINE requires the digitizing step; this is expressed in millimeters, for the Cartesian coordinates, or in degrees, for the polar coordinates.

In case of profiles comparisons it will be requested the tolerance, that is expressed in millimeters. The selected value is used for the maximum lower deviation (- toll) and maximum superior deviation (+ toll.).

At the end of the comparison it is possible to select different types of data presentation i.e. PRINTING, PLOTTING, PLOTTING OVER FILE.

#### PRINTING:

In a schedule are printed the coordinates of the digitized point (on the sample profile) and the distance to the compensated profile of the measured workpiece, with an histogram which shows if the workpiece is in or out of tolerance.

#### PLOTTING:

Is plotted the sample profile, and vectors (in a scale selected by the user), that represent the deviation from the profile of the measured workpiece.

#### PLOTTING OVER FILE:

Plotting data are saved in a file specified by the user, instead to be sent to the serial port, (See paragraph 5.2 "PLOTTING PROFILE" more forward in this manual).

In the phase of digitizing, a line of GREEN colour denotes the point digitized in this moment; its coordinates are displayed in the coordinates area.

In the phase of profiles comparison could be displayed the message: "Comparison profiles not possible". The error could happen for two reasons:

- planes on which the two profiles have been elaborated are different; for instance two profiles elaborated one in the X-Y plane and the other in the X-Z plane could not be compared.
- the limits of the two profiles are different; for instance if the first profile has a range on the X axis included between X 10 and X 100 when the second profile has a range on the same axis included between X 110 and X 200.

## 5. HARD COPY PROFILE



### 5.1. PROFILE PRINTING

With this option it is possible to get an hard-copy of the profile displayed on video. Also in this case, before starting to the data transmission ascertain that the printer is ready to receive it. It is possible abort the printing pressing the ESC key..



### 5.2. PROFILE PLOTTING:

First of all will be requested the scale factor. Higher is the scale factor, larger is the design. On video is visualized the profile with the dimensions that it will appear on the drawing sheet.

At this moment it will be possible :

- to confirm the selected scale pressing ENTER key.
- to change it pressing "S" key.
- to cancel the command pressing the ESC key.

Before starting to plot the profile, verify that the plotter is setted in ON-LINE mode, ready to receive signals from the serial port of the machine. If everything is OK and if plotter is connected to the computer, profile can be plotted in real time.

It is also possible to memorize the profile in a File, of which needs to specify the name. The File can be transferred, when desired, to the plotter using the DOS command "TYPE" (see TYPE on the DOS User's Guide for more information). This possibility is particularly useful because it does not require that plotter is always connected to the computer.

## 6. FILE CONVERSION



### 6.1. CONVERSION TO DXF:

A profile may be converted into the DXF or IGES format. If it has been previously calculated the correspondent profile composed with arcs of circle & straight lines, it will be displayed a menu for the selection of the file type to convert :

- point to point .
- circle arcs & straight lines.

Profile converted may be transferred to a CAD System. If the CAD S. is resident in the CATRIM P. C. at the end of the conversion METROSPLINE, if desired, transfers the profile with automatic procedure to CAD and runs CAD program.



### 6.2. CONVERSION FROM DXF :

For this conversion is required the name of the DXF file to convert and the plane containing the converted profile, due to the fact that the comparison between two profiles is possible only if these lie on the same plane. ( See 4.X.1: "PROFILES COMPARISON"). If CAD is AUTOCAD it is important that:

- the data of the profile stay on the LAYER 0.
- the quotation lines must stay on a different LAYER.

## 8. OPERATIONS ON FILE

### 8.1. LOAD PROFILE



METROSPLINE requires that the profile to be examined is in memory; otherwise it is necessary, first of all, to load it with this option that displays an ARCHIVE Window containing the list of the profiles; the name of the profile can be inserted from keyboard or selected directly from the list with the MOUSE or the cursor keys , PGUp and PGdn.



### 8.2. SAVE PROFILE

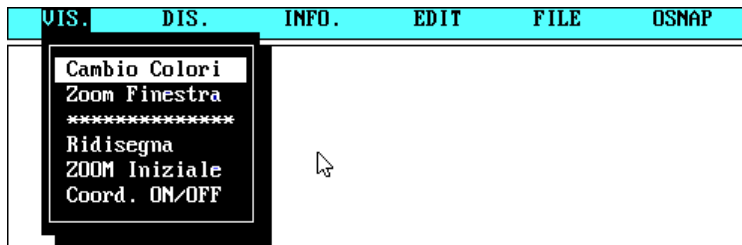
With this option you can save a modified profile



### 8.3. DELETE PROFILE

It is possible delete a profile with a procedure similar to that one of chapter 8.1; deleting command requires confirmation.

## PROFILES EDITING



With the pulldown menu (Fig. above) it is possible enter into a series of specific functions that allow the drawing of geometric elements as circles, arcs of circles, straight lines, and their comparison with the profile's points. The created design could be saved on disk and later recalled.

- **Display**: Contains the commands for the video management:

#### ⇒ CHANGE COLOUR

Changes the colour both of the elements that are drawn, and the points of the profile; selecting this item it will be opened a Window in the design area containing, in two lines, the 15 colours that you can use; the upper line shows the colour of the design, the lower the colour of the profile. A turned down arrow denotes the actual selected colour; with the cursor keys "<" and ">" the arrow is moved on the wanted colour and it is confirmed with the ENTER key. The

selection through mouse may be done by moving the mouse arrow over the wanted colour and pressing the LEFT key.

⇒ ZOOM WINDOW

Allows to visualize a part of enlarged profile, by selecting two points that define the diagonal of the window to enlarge. After first point selection, the area that will be "zoomed" is contained in a white square that is displayed larger or smaller depending on the movement of the MOUSE.

⇒ REDRAW

Clears the design Area and redraws the geometric entities.

⇒ ZOOM PREVIOUS

Redraws the design with the original scale.

⇒ COORDINATES ON / OFF

It enables or disables coordinates visualization of mouse pointing system.

- **Draw:** Allows the drawing of the simple geometric entities i.e. straight lines, circles, arcs of circle.

⇒ LINE:

Draw a straight line giving two point ( the beginning and the ending Points ).

⇒ CIRCLE 3 PT

Draws a circle passing through 3 Points.

⇒ CIRCLE C, P

Draws a circle giving his Center and a Point over the circumference.

⇒ ARC 3 PT

Draws an arc of circle passing through 3 Points.

⇒ ARC C, B, E

Draws an arc of circle of which are known the Center, the Beginning point and the Ending point . In this case is important to know that the route sense of the arc, from the initial point to the final point, is the counterclockwise sense.

For the selection of a point of the preceding geometric entities are available these possibilities :

◇ DIRECTLY OVER THE DESIGN:

Moving with the mouse or the cursor keys the two white lines (that act as cross pointer) on the wanted point, whose coordinates are displayed in the low right corner of the screen, and pressing the left key of the mouse..

◇ INPUTING COORDINATES THROUGH KEYBOARD:

The coordinates through keyboard could be inserted :

- as Cartesian coordinates, in the XY format.
- as polar coordinates, in the L<A format, where L represents the distance between the last probed point and the new point that wants to be selected, while A represents the angle that constitutes a hypothetical straight line passing through the two points with the abscissa axis. Negative angles cannot be inserted.

The Cartesian coordinates X,Y and the polar coordinates L, A are computed in a reference System connected with the workpiece previously selected by the operator.

When is required the selection of a point, with mouse or keyboard, it is possible to abort the immission pressing the ESC key or the right mouse button. It is possible go back with the immission of a point at a time up to the complete cancellation of the command.

- **Information:**

Provides information on a geometric drawn element, giving the possibility to print it on the certificate with a label, the nominal value and the design tolerance.

⇒ **LIST:**

According to the geometric selected element is visualized the list of the characteristics values of that element. For example for an arc will have visualized: center coordinates, radius, initial angle, final angle.

⇒ **DISTANCE 2 POINTS**

Provides the distance between 2 points.

⇒ **DISTANCE POINT - STRAIGHT LINE**

Provides the distance of a point from a straight line.

⇒ **DISTANCE POINT - CIRCLE**

Provides the distance of a point from a circumference of the selected circle or arc.

⇒ **ANGLE**

Provides the angle in degrees through two straight lines.

⇒ **COMPARISON**

Provides the distance of a point of a Compensated profile from a geometric drawn element ( Straight line, Circle).

- **EDIT:** Gives the possibility to modify one or more drawn elements.

⇒ **BREAK**

Separates an element in more parts: once selected the element is required a first point and a second point where perform the breaking. Let us consider a straight line (s. l.); if the two points are both inside the limits of the s. l. , this is divided in two separate s. l., and the central part is deleted. If one of the two points coincides with the initial or final point, the s. l. is shortened. If both the points coincide, respectively, with the initial point and the final point, the s. l. is deleted.

⇒ **ERASE:**

Deletes a geometrical element or a point from the profile.

⇒ **EXTENDS:**

Once selected two NOT parallel segments these are extended in their point of intersection.

- **FILE:** Manages the files containing the geometric elements of the drawing.

⇒LOAD DRAWING

Load a saved design .

⇒SAVE DRAWING

Save a design on the disk; the file name is the same of the profile resident in memory.

- **OSNAP**

In this menu is definite a set of functions abled only in the phase of acquisition of a point; the cross pointer becomes a square inside which must be the point or the element to be selected.

⇒NEAR TO :

Is selected a point, over a geometric selected element, that is the nearest one to the center of the cross pointer.

⇒PERPEND. TO :

This command allows to draw a perpendicular line to another. Once given the initial point is selected the wanted line over that METROSPLINE draws the perpendicular line.

⇒MID. PT. OF :

The point is included between the extreme of the element that it is selected. The option is not valid for a CIRCLE because there aren' t the extremities of it..

⇒END OF :

The point becomes one of the two extremities of the selected element. Also this option is not valid for the CIRCLE.

⇒CENTER OF:

The point becomes the center of an arc or a circle. In this case to obtain the required point it is necessary to select the relative arc or circle.

⇒TANGENT TO:

This option calculates a straight line passing for a point and tangent to a circumference. It requires to select the first point of the straight line and the circle or arc with which the tangency is wanted.

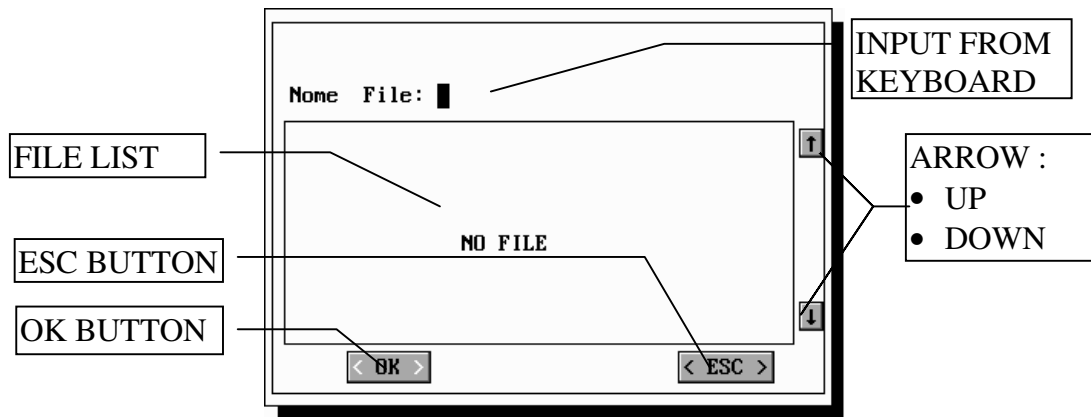
⇒INTERS. OF:

This option calculates the point of intersection between two elements of the design.

When it will be displayed the message "CHOOSE The ELEMENT," pointer of OSNAP must be shifted on the wanted geometric element. In case of "POINT INTERSECTION Of," both the elements (of which the intersection is desired) must be included in the square of selection. If must be selected a point of a profile, it will be used "NEAR" : it will be calculated the point of the profile nearest to the point cross center.

Using the commands of OSNAP may occur errors if are not present the conditions for the execution of the command (i.e. when is requested the intersection of two parallel straight lines); in this case will be displayed the error message. With the ENTER key program will continue.

## WINDOWS & FILE SELECTION



When a File must be selected, in the design Area will be opened a window containing the list of archived Files (See Fig. above). In the window are present different areas to which it is possible enter, pressing the TAB key marked by two arrows one above the other and turned in opposite directions.

1. **INPUT FROM KEYBOARD:** When on this field a black square (CURSOR) appears, through keyboard can be written the name of the file to select, with a maximum 8 alphanumeric characters; if the name is wrong, it is possible delete the last character pressing the BACK-SPACE key, that is situated on the up right corner of the keyboard with a left turned arrow.
2. **LIST:** Contains the files list of the archive; the name of the selected file appears in revers, written in blue on black background. A File can be selected **through keyboard**, moving the black selection line with the cursor keys, and pressing enter to confirm or **through mouse** bringing the mouse arrow over the file to select and pressing two time quickly the left key.
3. **OK BUTTON:** When this has selected the symbols "<" and ">" is WHITE, in contrary case is BLACK, this button is always activated, "clicking" with the mouse on this button is confirmed the file currently selected.
4. **ESC BUTTON:** It could be selected in different manners: With the pressure of the ESC key on the keyboard (left up corner), "clicking" with the mouse on the this button, or pressing TAB until the symbols "<" and ">" becomes WHITE and subsequently, pressing ENTER to confirm the selection. This selection involves the cancellation of the command.

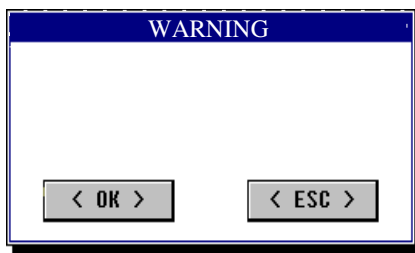
5. UP ARROW BUTTON:

6. DOWN ARROW BUTTON:

Options 5. and 6. correspond, respectively, to the PGUP and PGDN keyboard keys and are used to scroll up or down the file list of a page.

A page is composed by 24 names; if the files in archive are more, the visualization of the files list is displayed in more pages. To use this buttons press the keys PGUP or PGDN on the keyboard or "click" with the mouse on the symbols on the screen.

## WARNING & ERROR WINDOWS



Like for the files selection, also for the error messages, in the designs area it will be displayed a WARNING Window where the error is visualized . There are two types of WARNING Windows :

1. Visualizes an ERROR UNRECOVERABLE . To exit from the error condition it is requested the the ESC key or from keyboard or from mouse by "clicking" over the button on the video,.
2. The operator must make a choice. In the Window will appear two buttons, the "< OK>" and the "< ESC>"button.