MicroSurvey® CAD2002
Program Guide

By

MicroSurvey Software, Inc.

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MicroSurvey CAD2002

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Welcome

MicroSurvey Software, Inc. is pleased to present our flagship product - ***MicroSurvey CAD2002***. This version adds features that our customers will love. The Active Coordinate Editor, Active Traverse Editor, Active Batch COGO, and AutoCAD 2002 compatibility will help everyone.

Our sales have grown more than 2000% since introducing MSCAD. This enables MicroSurvey to move forward *faster to produce you a better product*. Our success is due to your success with our software.

We consider all the new features to be a Breakthrough. One of our beta testers reported a 50% timesaving since our MSCAD Pro 3.1 to produce a standard lot survey. This is what we are aiming for: a better, more enjoyable product to use that saves you time!

Included with this program is almost 6 Hours of training movies. Movies make the printed documentation obsolete. That is why we reduced our manuals to a "Getting Started" book that contains the tutorials in a lay-flat book, and a "Program Guide". The Program Guide is more like a map into the on-line help system than a full reference manual. Since the help system is electronic, we can treat it as a living document that can be updated with each service pack or new CD.

We plan on introducing more exciting products for you in the next year. Watch our web site for news and updates on this and other MicroSurvey products.

Yours truly,

Darcy Detlor,
President,
MicroSurvey Software, Inc.

Read the Getting Started First

This manual is a road map to the MicroSurvey program. Before diving into this program, you should read the Getting Started manual and do the tutorials. If you have finished the tutorials that are relevant to the work you do, then you can use this manual to read about some of the other capabilities of the program. Make sure you keep the on-line help system warmed up and ready to go. Many of the features are explained better in the help system, due to the ability to show colors and movies.

CAD Concepts

MicroSurvey is based upon a CAD system. As such, it is important to get a good understanding of the objects in a CAD drawing. Survey drawings consist mainly of simple items: Text, Lines, Arcs, Polylines (multi-segment lines), Parts (symbols), and dots.

Each of these objects is created manually by entering a CAD command, or the program can create hundreds or thousands of entities automatically for you. An example might be reading an ASCII file of coordinates. This will cause the program to create 3 text items and a dot for each coordinate. The text items are the description, elevation, and point number. The dot identifies the location. Each piece of text will be set to a specific color, style, and layer based upon default settings in the program. We can start by talking about these object properties…

Object Properties

These characteristics are said to be properties of an object:

- Objects may be placed on a different layer;
- Objects may have a different color; and
- Objects may be drawn with a different linetype.

These characteristics are associated with a drawing object automatically when it is created. This association is made through the parameters called BYLAYER, which determines the layer when it is defined. Alternatively, the user may choose (in the manner just described) a special color or linetype for an object.

Setting Current Object Properties

To call on these functions, select the buttons that are found on the control panel on the left-hand bar of the desktop. See the Program Interface section of the Getting Started manual for details on the screen layout.

Set Current Layer - Command SETLAYER

Set Object Color - Command SETCOLOR

Set Linetype - Command SETLINETYPE

Changing Object Properties

It may be desirable in certain circumstances to change these parameters once into the drawing or layer. In order to do this, the proper command is PROPEDIT.

The command PROPEDIT allows the user to make these changes:

- Objects may be placed on a different layer;
- Objects may be given a different color;
- Objects may be given a different linetype.
- Objects may be given a different linetype scale factor

Entering Data

Data Input

During the drawing process it is often necessary to modify the value of data that has been input during the various steps of design. This occurs when the program needs input values for solutions. In those cases a prompt will appear in the Command Window asking for the entry of a value.

Examples of these required values are:

- Coordinates for defining points of the drawing (starting and ending points, center points, points of contact, inserting points, base and target points);
- Distances;
- Angles;
- Width values (for example, line width); or
- Number.

Number

- Input by keyboard
- Pointing or Identifying

Coordinates

The different coordinate types can always be entered as absolute or as relative coordinate values.

- Absolute Coordinates
- Relative Coordinates

Deleting and Undeleting

Delete

During the creation of drawings, or, later during the editing, changing or detailing of drawings it may be necessary to delete certain parts of the drawing.

The program offers the command DELETE to accomplish this operation. You can also type ERASE to do the same thing.

The command DELETE may be used for these drawing entities:

- Individual drawing entities, like lines, circles, rectangles, or for groups of these entities;
- Complex objects;

Complex Objects

Complex objects are:

- Symbols and attributes
- Polylines
- Crosshatching
- Dimensioning

To delete pieces of complex objects, the respective object has to be converted to single entities again by the command EXPLODE Complex Objects.

Undelete: Restore Deleted Items

Entities which have been deleted by mistake can be restored by the command UNDELETE. This will cause all entities previously deleted to be restored.

DELETE can be canceled by the function Undo or the command U.

Call the commands

- By selecting the option in the pull-down menu *EDIT*
- By keyboarding the command in the command line area
- By clicking the icon from a toolbar

Transparent Commands

A series of commands and functions are at your disposal even when another command has been called and is being executed. These commands are called transparent commands.

They include commands and functions for:

- Changing the configuration of the drawing area (for example,. ZOOM, PAN) or the window functions (for example,. WCASCADE, WTILE);
- Controlling precision (grid, Snap, Ortho, Object snap functions);
- Switching between Command Line area and Text Window;
- Loading or closing palettes;
- Calling the info functions; and
- Calling the Help function.

After the execution of the transparent command the program continues to execute the original command.

If transparent commands are called using the Command Line area, an apostrophe must be placed before the command. For instance: 'ZOOMOUT.

Otherwise transparent commands can be called as usual by clicking the proper option in the pull-down menus or by choosing the proper symbol.

Transparent commands can **not** be executed if:

- The program is performing input of a sequence of characters; or
- A dialog or selecting window of another command is opened.

Precision Aids

The commands and functions that serve as precision aids are:

Grid Display

Snap Grid

Orthogonal Mode

Object Snap

Control of Precision Parameters

 The precision aids can be controlled by the common dialog box, which is activated by clicking the left button of the shown symbol in the vertical control panel or by entering the command PRECPAR.

Individual Precision Aids can be switched on and off and their parameters can be defined within this dialog. There are input fields and control elements for the individual settings. The user may enter numeric values using these fields.

With the buttons in the control panel on the left side of the screen the functions Orthogonal Mode, Snap Mode, and Grid Display can be turned on and off.

Coordinate Filter...

MicroSurvey offers the possibility to use **Coordinate Filters** when point values are requested. Coordinate Filters allow the user to relate the point value inputs to the X-, Y and Z-coordinates of existing entities.

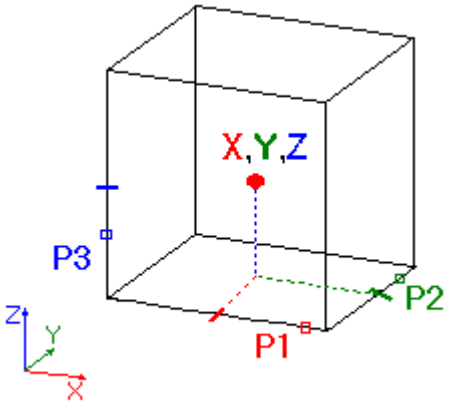
Some possible inputs that might be requested for point values are:

x, value	x, y
value, y	x, y, z
value, value, z	value, y, z
x, value, z	x, y, value

If the user answers the question with the input: **x,10** the user will be requested to enter the x-value separately. It is now possible to catch a second object or the geometrically significant point of an object by using an Object Snap function. The X-value of this point will be interpreted as input value for the original point definition (x,10).

> LINE
From point: **P1**
To point: **x,10**
X-coordinate: **end**
from: **P2**
To point: **15,y**
Y-coordinate: **cen**
from: **P3**

Example: Identifying the center of a cube



> POINT
Point: *X,Y,Z*
X-Coordinate: *MID*
of object: *P1*
Y-Coordinate: *MID*
of object: *P2*
Z-Coordinate: *MID*
of object: *P3*

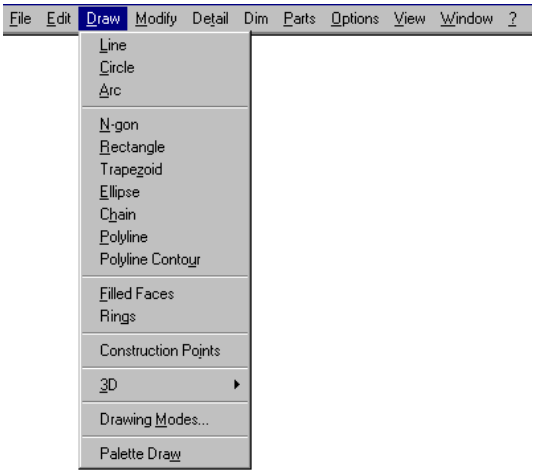
Drawing Measurement

The distance between two integer coordinates is called a drawing measurement. A line from point 0,1 to point 1,1 has a length of 1 drawing measurement. When drawing inside a coordinate system you can create objects at a 1:1 ratio, avoiding calculation- and other measurement errors.

The drawing as a whole or any excerpt from it can be printed or plotted in any scale. You save the trouble of creating complete drawings and detail drawings in different measurements.

Calling Commands

Calling Commands and Options: Pull-Down Menus



Commands (and their options) can be selected using pull-down menus. The menus and commands may be requested by key sequence (for example,. ALT + F, N for new file) or by clicking the proper option field within its menu.

For some commands, submenus are available. These submenus offer further commands or variations of the command.

Calling Commands and Options: Toolbars

A series of frequently used commands can be called using symbol fields.

Permanently displayed symbols in the Function Bar or the vertical symbol bar. These symbol fields represent mainly commands from the pull-down menus ***File*** and ***Display***.

Commands and functions are called by clicking on the symbol.

Calling Commands and Options: Input by Keyboard

Input by keyboard has the advantage of precision. Using this method, the user can construct drawing entities with exact dimensions or with exact placement in the coordinate system.

Values for distances, straight lines and other linear quantities are specified in drawing units, angle values in degrees.

Coordinates are given, in sequence, as the values for the X-, Y- and Z-axes with a comma between.

Please note that a point is used as a **decimal sign**. A comma serves as dash between the values for the single axes.

When specifying an input value for an angle the sign < (smaller than) comes before the value.

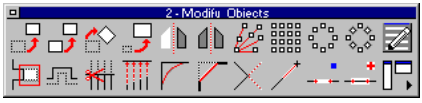
An input sequence on the keyboard might look like this:

LINE
From point: 2.5,0
To point: 5.75,0
To point: 0,3.25
<ENTER>

Moving an object might would look like this:

Select objects: Selecting
Basis point: 5,2
Target point: @3,0

Calling Commands and Options: Palettes



A series of frequently used commands can be called using symbol fields. Symbols in palettes, arranged in any way, which can contain any command for which the palette is designed.

Commands and functions are called from a palette by clicking on the symbol.

 **Loading palettes**

Palettes are files with the file extension <.mnp>. The command PALETTE is used to load a palette.

Object Snap

Object Snap

Many functions and commands require the definition of an exact position in the drawing. The determination of exact points can be done either by coordinate input or by indicating points. The input of coordinates has the advantage of precise placement, whereas indicating points often has an advantage in speed and ease of handling.

A very efficient method to take advantage of both techniques is to indicate points with precise references to a given geometry with the help of the Object Snap functions.

Object Snapping allows geometrically significant points of existing drawing entities to be found. These points are then used as inputs for a new drawing step.

The Object Snap functions can be activated permanently or be called for a single operation.

Controlling the Object Snap Functions

Object Snap: Permanent Activation

Using the SETTINGS dialog box, individual modes of the Object Snap functionality can be permanently activated.

Implicit Object Snap: Activating from Case to Case

For the single use activation of a certain mode of the Object Snap function you can choose between several options:

Snap Palette

The user may use the Snap Palette to call individual object snap modes for singular use in combination with a designing or editing command. This standard palette can be opened when needed and be placed anywhere on the drawing area. The user may now call the object selecting modes by cursor selection. The desired object snap modes can also be included in user-defined palettes with the help of the menu and dialog editor.

Cursor Menu

Another very effective method of calling an individual object snap mode is by activating the Cursor Menu using the middle mouse button.

Keyboard Input

Keyboard input of a command abbreviation is another method of activating the object snap functions. The object snap functions and their abbreviations are as follows:

Object Snap Mode	Object Snap Input Abbreviation
End point	end
Mid point	mid
Center point	cen
Quadrant	qua
Intersection	int
Implied Intersection	imp
Perpendicular to	per
Tangent to	tan
Next point on entity	nxt
Insertion point	ins
Point entity	nod

Object Snap: Mode of Operation

For precise drafting, many commands require that you select an exact position in the drawing. You can pick the precise location by typing the coordinates, or by picking a point in conjunction with an object snap. Object snap allows you to pick geometrically significant points on existing entities.

Object snap mode can be activated permanently or called for a single pick. The object snap functions can not be called separately, but may be used only with a command which requests the input of a coordinate in the drawing. (ie. an x,y,z point)

The operating principal when using object snap functions is always the same:

1. Start a command that requires a point, such as "From point:".
2. When the point is requested, reply with an object snap mode. The cursor then takes the form of a small box. the command prompts for the point with "of".

3. Select the object with the cursor.

Which geometrically significant point is "picked" during the activity depends on

1. The type of the drawing entity;
2. The respective object snap mode; and
3. The specific point of the object which is chosen.

Object snap modes occasionally offer no practical use. For instance, the user cannot define the ending point of a circle, while a line segment has no center point.

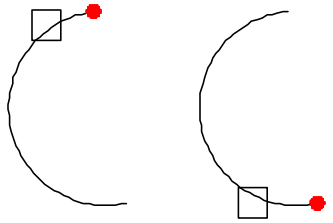
On the other hand, it is possible that a drawing entity contains several points to which the criteria of the object snap function correspond. A line entity or an arc, for instance, always has two end points.

Note:

The object snap function *Endpoint* does not differentiate in regard to the starting or ending point of an object.

An object snap function identifies, in those a cases, the nearest of the two or more possible points. It determines, when selecting an object by positioning the cursor, which of the possible points will be snapped. The user should always place the cursor as near the desired point on the object as possible.

The illustration below shows the connection between the point selected and the point snapped:



Object Snap Modes

There is a difference between object selection and object snap modes. Object selection treats a group of entities, such as a rectangle, chain, and polyline, as a single object. In contrast, object snap modes treat these as individual entities. For example, you can snap to each segment and vertex of a polyline. This is also true for polygons and 2D and 3D faces.; the edges are interpreted as by object snap as

individual lines In this way, the object snap modes can be used for the edges of planes as well as for segments of chains and polylines.
For simplicity, when this text refers, for example, to the "end point of a line" the term also refers to polyline segments and the edge of a plane.

END Endpoint

The Object Snap function End snaps to the end point of a line or of an arc.
Planes or the edges of solids are also interpreted as lines, and end points of planes and solids can be trapped by the mode end point.

MID Point

Snaps to the center of a line or an arc.
Planes, or the edges of solids, are also interpreted as lines, and may be snapped by the mode mid-point.

CEN - Center Point

Snaps to the center of a circle or an arc.

QUA - Quadrant

Snaps to the nearest quadrant of a circle or an arc.
Quadrants refer to the points of a circle or an arc, situated on the circumference at 0, 90, 180 and 270 degrees. These are the intersecting points of the axes of a coordinate system upon whose origin lies the center of the circle or the arc with the circumference of this circle or arc. Only the next visible quadrant of an arc can be snapped.

NOTE: For further operations, the command ROTATE will not rotate the quadrants with the object.

INT - Intersection Point

Snaps to the intersection points of lines, arcs and circles or any other combination of them. This includes dashed and centerlines where the intersection point may not have line segments actually touching.

IMP - Implied
The Int object snap mode only returns an intersection point of entities that have a real intersection. In contrast, the Imp mode snaps -- in addition -- to an implied (intersection point. This is the point where the extension of two entities would intersect.

PER - Perpendicular to...

This mode requires that a starting point for a function has already been selected. Activating the Object Snap mode will relate this point to the snapped point.
For instance, when a perpendicular is desired, the ending point for the perpendicular is picked, and the Object Snap mode Perpendicular is chosen. The result is a perpendicular line from the ending point to the chosen entity.

TAN - Tangent

Similarly, this mode requires that a starting point for a function has already been selected. Activating the OBJECT SNAP mode will relate this point to the snapped point.
For instance, when a tangent to a circle is desired, the ending point for the tangent line is picked, and the Object Snap mode Tangent is chosen. The result is a tangent line from the ending point to the chosen entity.

NXT - Next Point on Entity

This mode snaps to the next situated point of an object, when at least one point of this object lies within the borders of the selecting cursor window. This ensures that entities which are designed or modified by use of the OBJECT SNAP function share actually a point with the selected object.

If the user selects a point on the space without using this OBJECT SNAP function it is possible that the selected point is not on the desired object. The two objects would not have a common point, or a closure.

INS - Insertion Point

Snaps to the insertion point of a part insertion, an attribute or a text object.

NOD-Point

Snaps to a point that was created by the drawing command POINT.

Object Selection

Object Selection - Summary

Many commands, especially the commands for modifying and detailing of objects, presuppose that the objects to be worked on are **selected** (and highlighted). This can be done by the Object Selection functions.

Object Selection functions can not be called by a separate command, but will always be activated by the program when the current operation requires selection of entities. The request for selection occurs in the command line area with this prompt:

Simultaneously, the program displays functions that may be useful in the current operation in the option bar.

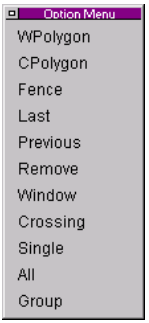
All objects selected in subsequent operations will be added to the selection set. The selection set includes the sum of all entities selected for an operation. Using the delete / add option ('+' or '-'), it is possible to change the selecting mode between ***Adding Objects*** and ***Removing Objects*** from a current selection set. The user may add other entities for selecting or erase them as desired.

Selected entities will be highlighted.

After marking all desired entities the user may finish the selection by pressing ENTER: The program will continue with the execution of the command.

Object Selection

Some commands allow you to “pick objects on the screen. For example – building a house by picking lines and arcs from an existing drawing. When you are in the object selection mode, you will see this small menu appear:



When selecting objects, you create a “selection set. Using the Add/Remove option, it is possible to change the selecting mode between **Adding Objects** and **Removing Objects** from the current selection.

Selected entities will be highlighted. After marking all desired entities you finish the selection by pressing ENTER: The program will continue with the execution of the command.

Object Selection Modes

In order to select objects the user can employ several modes of OBJECT SELECT. Two standard modes are at your disposal immediately. Other modes are available in the option bar.

The modes can be called by:

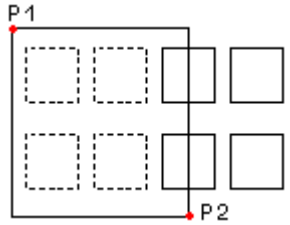
- Selecting the proper switch;
- Inputting of the name of the object-selecting mode;
- Inputting the abbreviation for the name of the object-selecting mode.

The abbreviations to choose an object selection mode are as follows:

Window	w
Crossing	c
WPolygon	wp
CPolygon	cp
Fence	f
Last	l
Previous	p
Add	+
Remove	-
Single	si
All	*

Object Selection Modes

Window

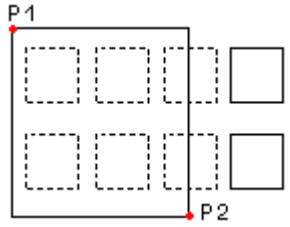


A (rectangular) selecting window is defined by two diagonal opposite corner points. Position the cursor on the first corner of the selecting window and select that point.

The user must take care that no object is inside the cursor window (box) during the fixing of the first corner point. Otherwise the option "Crossing" will be active instead of the option selecting window and the object that is situated in the cursor box will be highlighted.
Determine the second corner point of the selecting window. This will select and highlight all of the entities that are entirely within the window frame described.

The command line will again display the prompt to select objects. Continue the object selection operation with the same or another selection mode, or terminate it by pressing ENTER.

Crossing

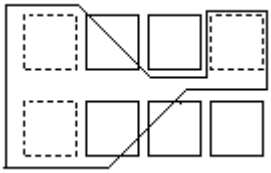


When selecting an object by **Crossing** only entities whose contours touch, or cross, the window frame will be chosen.

Position the cursor so that its window crosses the contour of the desired drawing entity and select to highlight the object.

An object will not be highlighted if the cursor is only situated on the area that is included by the contours of the entity, without touching the contour itself. In this case the program will activate the selection mode Window and interpret the input as the first corner point of the window.

Wpolygon



The mode **Window Polygon** functions in principle by the same proceeding as the option Window. This means, that all drawing entities are highlighted whose contours are **entirely inside** of the selection window.

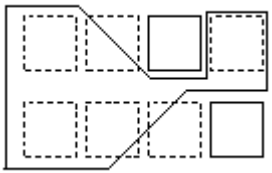
The difference between the two modes is in the form of the selecting window. The mode Window Polygon allows the user to design an irregular window with as many corner points as are desired. This allows more flexibility in selecting then selecting using Window.

To use **WPolygon**, first define in succession the corner points for the selecting window.

The corner points for the selecting window can be defined in a clockwise or counter-clockwise series. The program allows the polygon lines to cross or intersect themselves.

ENTER will conclude the definition of the selecting window. All objects that lie totally inside of the window will be highlighted.

Cpolygon



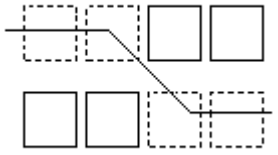
By the selecting mode **Crossing Polygon** you can create an irregularly formed window with as many corner points as you like. All objects that lie **inside** of the selecting window or are **crossed** by the window frame will be highlighted.

Define successively (clockwise or counterclockwise) the corner points for the selecting window. The preview mode will offer a preview of the emerging polygon.

The program also allows the polygon lines to cross or intersect themselves.

ENTER will conclude the definition of the selecting window. All objects that lie totally inside of the window or are crossed by it will be highlighted.

Fence



The Object Selection option **Fence** represents an effective variant of a crossing selection mode. The object to be selected will be not be closed or crossed by a window, but only "cut through" by a line, or fence.
The line of the fence can be defined by as many points as the user desires, and the fence may even intersect or cross itself.

ENTER will conclude the definition of the fence line. All objects that are touched or crossed by the line will be highlighted and added to the selection set.

Last

The Object Selection option **Last** marks the last drawn entity and adds it to the selection set.

It does not have a memory function, that is, it is not possible to add to the selection set the entity created during subsequent last operations by repeated calling of the option.

Note: If the last drawn entity lies on a frozen layer you will receive the message in the command line area that no entity was selected.

All

The Object Selection option **All**, which is called by the input of ALL or the abbreviation A marks all entities included in the drawing with the exception of those located on frozen layers and adds them to the selection set.

Previous

The Object Selection option **Previous** (P) restores the previous selection set. All entities selected by the previous selecting operation are highlighted.

Note: This mode has the same restrictions as the modes Last and All, in that entities which are located on a frozen layer will not be highlighted, entities on a locked layer will be highlighted but cannot be handled.

Remove Mode / Add Mode

The option bar contains a toggle button to switch between *adding objects* to the current selection set and *removing objects* from the selection set.

The mode *Add Objects* is the default active mode, so the user can select further objects and add them to the selection set at any time.

By selecting the button **Remove** in the option bar the user changes the

selection mode to *Remove Objects*. Objects already selected and highlighted will be removed from the selection set by this change.

Single Selection Mode

The option **Single,** which is called by the abbreviation **Sl,** de-activates all Object Select modes for the next and subsequent operations, with the exception of the standard modes WINDOW and CROSSING.
Note: This option is not offered in the option bar, but may to be called by keyboard input.

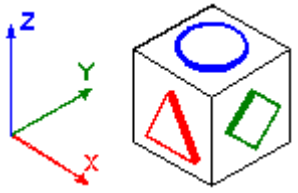
Coordinate Systems

Coordinate Systems

The coordinate system of this program allows you to determine every single point of a drawing surface or a drawing object unambiguously through coordinates. These coordinates originate in the coordinate system. The program uses a Cartesian coordinate system consisting of three coordinate axis, of which every pair is arranged orthogonally and which cross at the origin. All axis have the same measurements.

To determine the coordinates of a point, the distances of this point to the x- and y-axis, in a 3-D system also from the z-axis, are measured.

The program differentiates the coordinate system between a world coordinate system (WCS) and the user coordinate system (UCS).



World Coordinate System

The world coordinate system is a permanently installed Cartesian coordinate system that is used as a standard for all drawings. The x-axis measures the horizontal distance, the y-axis the vertical distance from the origin. The z-axis is orthogonal to the x- and y-plane and measures the distance of a point to this plane.

The user cannot change the world coordinate system.

Coordinate Input

During the drawing process it is often necessary to enter coordinates for defining points of the drawing (starting and ending points, center points, points of contact, inserting points, base and target points)

Coordinates are entered in the following formats:

Cartesian Coordinates

Cartesian Coordinates define a point by its X-, Y-, and Z-values. These values are entered separated by a comma. A point is used as a decimal sign.

If no Z-value is entered it will be interpreted as Z=0.

Polar Coordinates

Polar Coordinates define a point by its distance from the coordinate origin and the angle in the XY-plane (from the X-axis). The two values are separated by the sign < (smaller than).

For example, the value input *10<60* defines a point with the distance of 10 drawing units from the origin of the coordinate system and an angle of 60 degree from the X-axis.

Spherical Coordinates

Spherical Coordinates define a point by its distance from the coordinate origin, the angle in the XY-plane, and the angle to the XY-plane.
For instance, the value input *10<60<45* describes a point with the distance of 10 drawing units from the origin of the coordinate system, an angle of 60 degree from the X-axis in the XY-plane, and an angle of 45 degrees from the XY-plane in direction to the Z-axis.

Cylindrical Coordinates

Cylindrical Coordinates define a point by its distance from the coordinate origin, the angle in the XY-plane and its Z-value. Distance and angle are separated by < , angle and Z-value by comma.
For example, the input *10<60,5* defines a point with the distance of 10 drawing units from the origin of the coordinate system, an angle of 60 degree from the X-axis in the XY-plane, and a Z-value of 5 drawing units.

This table gives an overview of the input formats of the different coordinate types, both for absolute as for relative coordinates.

The different coordinate types can always be entered as absolute or as relative coordinate values.

Absolute Coordinates...

Absolute Coordinates use as their reference (starting) point the origin of the current coordinate system, or, in the case of angle values, the angle to the XY-plane of the coordinate system.

Type of coordinate		Input Example
Cartesian	X,Y,Z	3,8,6
Polar	distance<angle	7.5<45
Spherical (3D)	distance<angle,Z-coord	7.5<45,12.2
Cylindrical(3D)	distance<angle1<angle2	7.5<45<33

Relative Coordinates...

Relative Coordinates use as their starting point the last defined point. Relative coordinates are selected by adding a sign **@** before the value.

Type of coordinate	Input	Example
Cartesian	@deltaX,deltaY,deltaZ	@3,8,6
Polar	rel. distance<angle	@7<45
Spherical (3D)	rel. distance<angle,Z-coord	@7.5<45,12.2
Cylindrical (3D)	rel.distance<angle1<angle2	@7.5<45<33

Coordinate Symbol

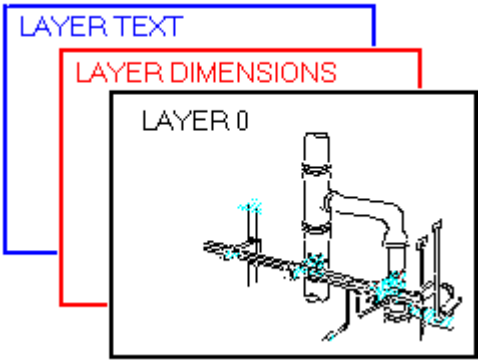
Normally, the program puts a coordinate symbol in the lower left corner of the Drawing Area. This symbol indicates the position of the axes of the coordinate system and provides orientation of the alignment of the current coordinate system.

The user can classify the individual axes of the coordinate symbol by different colors, which increases the clarity of this symbol.

In addition, the coordinate symbol can be faded in and out by the transparent command TICON (Toggle Icon).

Layer Concepts

Layer - Basic Information



The concept of working with layers is analogous to manual drawing using layers of see-through foil. Parts or portions (which may be functional portions, such as construction lines) of one complete drawing are drawn on different foils, which, imposed on top of one another, form the entire drawing.

Layers are useful in separating objects, such as parts, sub-assemblies, or structural components such as pipe sizes or building materials. They also serve to separate elements such as construction lines, center lines, outlines, and dimension lines, or things that are to be placed in a group, such as foliage in an architectural drawing, or parking areas.

All parameters of a drawing that are active for the entire drawing, like the coordinate system, the Zoom factor, the view arrangement, fonts, attributes, and so forth, are valid for all layers and are always the same.

The **advantages** of creating a drawing in this fashion, whether working manually or electronically, lie in a clearer construction of the drawing and better control than drawing on a single foil.

It is possible to **structure a drawing** through formal or content characteristics.

The standard layer is always called **Layer 0**.

One layer in each drawing will be defined as the **active** layer. All new objects will automatically be placed on this layer. As long as **Layer 0** is the only one existing, it will also be the active layer.

Each layer has a **color** and a **linetype** associated with it when it is created. Anything drawn on this layer will be drawn with this linetype and color.

Aside from the color and the linetype, there are other attributes of the layer that allow the user switch between states. The states are **visibility** (on/off), **freeze/thaw** and **lock/unlock**.

The layers with their status and their attributes are part of the drawing. They are saved together with the drawing and can be called when re-opening the drawing.

Layer Attributes

Layer Name

To identify individual layers and differentiate between them, each must have a distinctive name. The name can consist of up to 31 characters. They can be letters, numbers, or special characters like _ (Underline) and $ (Dollar). All characters are automatically capitalized.

Color

Every layer has a color that is user-defined. If a new color is not chosen, the layer will be colored black. The standard color of the layer can be marked in the color box as BYLAYER.

It is possible to choose the standard color of a layer from among 255 color shades. Every color has its own number.

Since it is possible to draw on the active layer only, objects still to be drawn will be in the standard color of the active layer, unless a new color was chosen with the option Object Color. If the standard color of the layer is changed using the dialog box for layer control then all drawing elements of this layer that are highlighted with the color BYLAYER will assume the new color.

Linetype

Similarly, there is a standard linetype that is assigned to the line. If you do not choose another linetype, the Linetype CONTINUOUS will automatically be assigned to it.
The chosen linetype is marked in the dialog box BYLAYER. All elements are drawn in the linetype of the layer unless another linetype is chosen for the element.

Visibility (ON/OFF)

One of the major advantages of working with layers is that it aids the user in visualizing the part, or drawing. The visual clarity of a drawing can be improved greatly through choosing layers properly and hiding the drawing elements. Concentration on meaningful drawing elements makes the drawing and construction much easier.

The visibility of a layer can either be turned on or off. Every layer can be turned on or off independent of any other layer(s). Changing the visibility is done through the Layer Manager.(command LAYER).

Locking / Unlocking Layers

It is possible to lock layers so that the objects or elements contained within the layer may not be changed.

Normally, new layers will be created with the status *unLock*. There are no limits inside the normal program functions when working with an unlocked layer.

When a layer's status is locked it is not possible to change, add, or delete objects from this layer. The status must be changed to *Unlock* if the user desires to work with the objects of these layers. This function makes it difficult to make an unauthorized drawing change.

It is possible to draw new elements onto locked layers but it is not possible to work with them further. Other options of layer control can also be affected when working with a locked layer. For example, the standard color of the layer may be changed. A locked layer may be designated active, visible or invisible, frozen or thawed.

Thaw / Freeze Layers

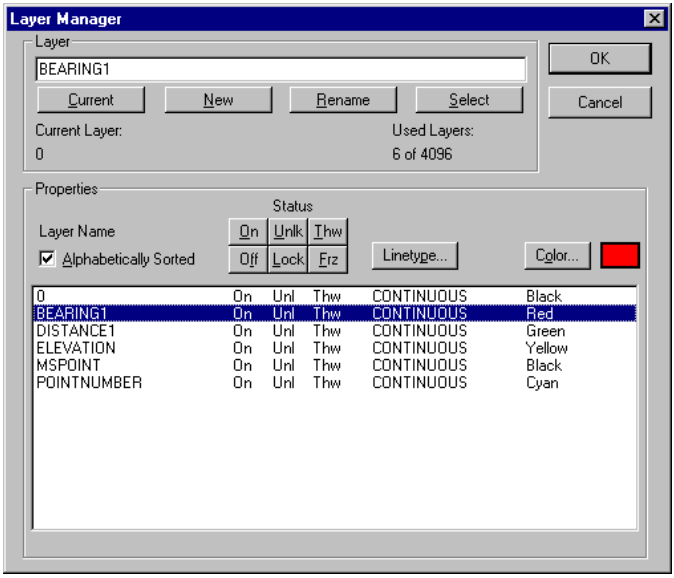
The options Thaw and Freeze are functions that control the visibility as well as the influence of the objects within the layer.

A frozen layer is both invisible and locked and thus impossible to change. A frozen layer will not be included in drawing regeneration. When working with very complex drawings, freezing some unneeded layers can increase the speed of drawing considerably.
The option Thaw undoes the freezing of a layer. The layer will become visible and changes may be made once more. The current layer of a drawing cannot be frozen.

Layer Manager

Command: LAYER

Creating new layers, choosing an active layer and controlling all previously described layer characteristics is accomplished through a dialog box. This dialog box is called *Layer Manager*.



Current Layer and Used Layers

In the upper rows of the *Layer Manager* Dialog Box is a field for the name of the active (current) layer and a field for the number of defined layers in this drawing.

Layer

Later rows contain combined view and entry fields for layers that still have to be worked on, and a selection box that allows the user to choose between creation of new layers, renaming layers and choosing an active layer.

Properties

There are fields for the layer state and for choosing linetypes and layer color.
Directly beneath that, in the dialog box, all the existing layers of the drawing are shown. Any of these may be selected and activated with the cursor. The layer color and linetype are also shown there.

The meanings of the abbreviations used are shown here.

Abbreviation	Meaning
On	Visibility on - The layer becomes visible
Off	Visibility off - The layer becomes invisible
Unlk	Status free, drawing elements can be edited
Lock	Status closed, drawing elements can not be edited
Thw	Layer thawed, visible and can be edited
Frz	Layer frozen, invisible and can not be edited, will not be regenerated

Viewing, Zooming, Panning

Viewports

The program is capable of opening as many as four drawings at the same time. Each of these drawings can be viewed in its own window. Further, it is possible to open as many as four viewing windows (viewports) for each drawing so that the drawing may be seen from different angles or so that portions of the drawing can be examined.

Opening Viewport Windows

In order to open a viewport, a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, the user should remember that all commands are executed in the active drawing. Open the menu *Window* and choose the option *Open New Viewport…* (command WOPEN). Before the program opens another window you will be asked to determine the type of view in this new window. Choose a standard view from the menu, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows.

Changing Viewport Windows

There are two ways to change the active window:

- Clicking on any part of the chosen window with the cursor will activate it.
- Open the menu *Window*, and find where all open drawing windows are listed. The active window will be marked by a check in front of its name. When a new window is opened, it its marked by a number in front of the specific entry.

Viewpoint / Angle of View

The direction from which an object is viewed influences the view of the object, and may be changed in many ways inside one drawing window. Each additional view will be portrayed in another viewing window. It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the pull-down menu VIEW provides the facility to make that definition.

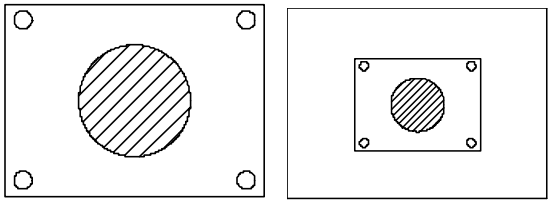
Saving and Retrieving Views

It is convenient when working with different drawing portions to save, and later, to retrieve changes in angles of view, different drawing portions and scales of view. This may be accomplished by using the command VIEW.

Zooming and Panning

The ‘ZOOM and ‘PAN commands are transparent ones which may be executed while other commands are being executed. It is possible, for instance, to execute the command ‘PANDOWN (move downward) while entering points for drawing an element.

Zooming



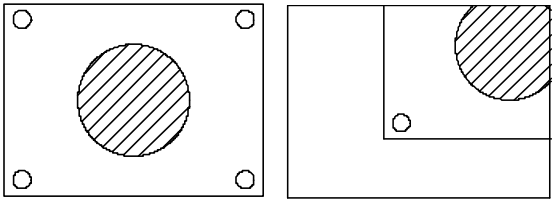
The ZOOM commands change the viewing scale of the drawing, or of the portion of the drawing chosen by the user. The absolute ratios of

scale of the entire drawing to the included objects chosen remains unchanged.

These variations on the ZOOM command are possible:

- Zoom Window ZOOMWIN
- Zoom Back ZOOMBACK
- Zoom Fit ZOOMFIT
- Zoom In ZOOMIN
- Zoom Out ZOOMOUT
- Zoom Factor ZOOMFAC

Panning



The command PAN moves the visible portion of the drawing to the right, to the left, upward or downwards. It is now easy to look at objects that were previously not visible because they were off the viewing surface. In contrast to the ZOOM command, the drawing scale does not change.

These variations on the PAN command are possible:

- Portion to left PANLEFT
- Portion to right PANRIGHT
- Portion up PANUP
- Portion down PANDOWN

Parameters for Zoom Commands

To activate the function *View Parameters* choose the item of that name from the menu *View*. Or, using the keyboard, type the command VIEWPAR.

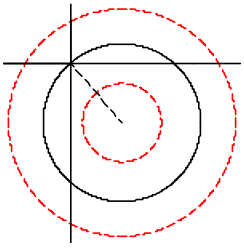
Zoom Factor

In the entry fields **Zoom In Factor** and **Zoom Out Factor** the user may choose the factor by which the drawing on the screen is to be enlarged or reduced. These factors affect only the visible portion of the drawing.

Pan Factor

In the entry field **Pan Factor** the user may choose the factor by which the visible portion of the drawing is moved using the command PAN. A pan factor of 0.5, for example, moves the drawing portion to about half of the visible drawing area.

Dynamic Preview and Rubberbanding



Dynamic Preview Mode and Rubberbanding are a visual construction aids that allows the user to watch the result of the pending operation before it is executed, and, if required, to make corrections. This feature also facilitates user understanding of commands and options and helps to learn the program.

Rubberband: A Marking Line

In any command that requires selection of two or more points, a marking line (rubberband line) follows the cursor movement. This line connects the last selected point with the current cursor position.

This gives the user a visual impression of the operation and allows a better estimate of distances and angles between the last fixed point and the following point.

The rubberband line is useful both in drawing and in modifying objects.

Used during the drawing process it illustrates:

- The ending point of the next segment of a line, chain or polyline;
- The radius or diameter of a circle;
- The distance between starting- and ending point of an arc; or

- The next corner point of a 2D or 3D plane.

When used during modification of objects it helps show the distance and angle from one base point to a second point, as well as the target point for moving, copying, scaling or other modifying steps.

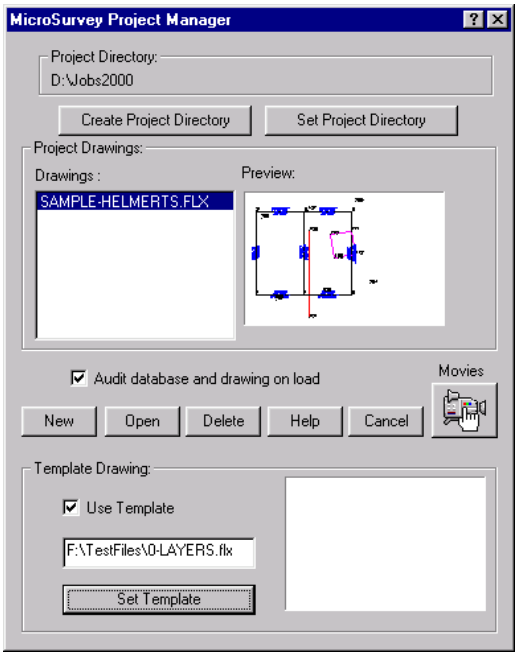
Object Preview Picture

The Object Preview Picture shows in advance the contours of an object to be designed or modified. It is possible to recognize the consequences of changing of parameters like the radius of a circle or like removing or changing the angle of a copy of the original object. This option allows the user to interactively influence these changes.

File Menu

Project Manager

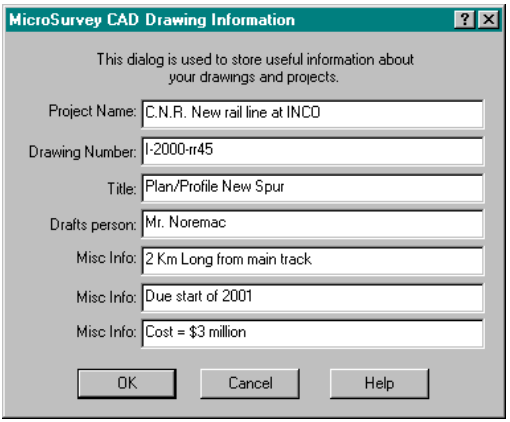
The Project Manager is a tool made to assist you in organizing your drawing and data into the correct folders or directories. You access the Project Manager at any time using the Open file button  on the top left toolbar.



This Project Manager does more than just open an existing drawing or create a new drawing; it allows you to group your drawings into projects to keep your hard drive from becoming cluttered. Other database and drawing checks are also done for you to help protect your job's integrity.

You can pick the dialog objects in the on-line help for more details on each item.

Drawing Information



Some information about a job really needs to be stored with the job so we have created the above dialog box to give you this option.

To save the information you must save the entire job at least once, and then enter this information and save the job once again. Doing this in the reverse will not save the information to disk and in the drawing.

Close a Drawing

Command: CLOSE

Closing a drawing indicates that the work on this drawing is finished. The program remains open. If a drawing has been changed before closing there will be a prompt asking whether the file should be saved. This is to prevent unintended loss of work.

If the user decides to save the drawing, the program will automatically switch to the standard routine to save the drawing under a new name.

Select the command CLOSE in the menu *File* or press the keys CTRL+F4.

Save Drawing File

 **Command: SAVE**

Save

The command SAVE causes the drawings in the active window to be saved. At this time the file will be saved under its present name. The file is saved in the same drawing format and to the same storage device and area as was specified at the time of opening. The file last saved in this area will be replaced.

The command to save can also be invoked from the menu *File* or by clicking on the symbol *Save* in the Function Bar.

Local variables will be saved along with the drawing and the enclosed drawing entities current at the time the automatic save occurs. Such settings as the current coordinate system, the dimensioning mode and other adjustments will remain at the user's disposal.

Auto-Save

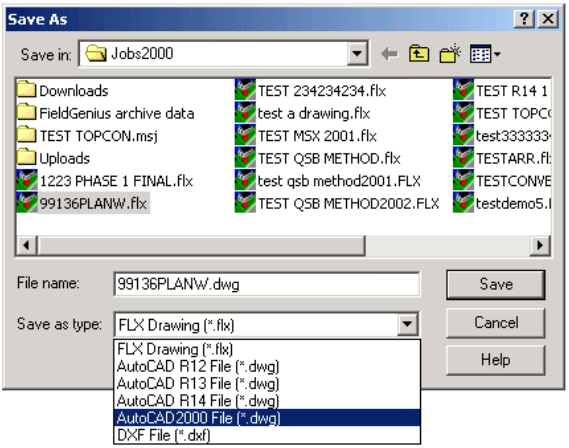
To avoid unforeseen loss of data we strongly recommend periodically saving drawings while working on them. For this purpose use the function ***Save Automatically*** which can be activated in the configuration dialog window. While an automatic save is occurring the current drawing will not be closed.

Save As...

Command: SAVEAS

The command SAVEAS also saves the drawing in the current window, similar to the SAVE command. However, it allows the user to save the drawing under a different name, in a different area, or of a different file type.

Use this command to save a file in another format such as an AutoCAD R2000. Use the "save as type" pull-down to select AutoCAD R2000. The file extension flx will automatically change to dwg when the selection is complete.



Save As MSCAD 2001 (Backwards Compatibility)

Command: SAVEAS2001

SAVEAS2001 is used to save an MicroSurvey CAD2002 file in MS-CAD 2001 format. The file created will load correctly into MS-CAD 2001.

The program will prompt you for the name of the file. It will default to the current file name. When you have entered the file, the program will save the drawing in the older format, plus it will strip out some of the extra features found in the 2002 version.

The program will immediately close your drawing so it does not do an Autosave into 2002 format.

We think it makes more sense to upgrade all of your machines to CAD2002!

Command: SAVEAS31

SAVEAS31 will perform the same function as SAVEAS2001, but you will lose TrueType font capabilities as well as many other features.

Save All

Command: SAVEALL

The SAVEALL command will not only save the drawing in the current window but also all other open drawings.

Delete Network Lock File

Command: DELFLK

The program contains a function for deleting the so-called lock-files, called from the menu *File,* or entered by typing the command DELFLK.

Lock-Files are security files that the program creates when a drawing file is opened. They carry the same name as the original file and can be identified by the extension **.flk**. Lock-Files will be deleted automatically, if the drawing files are closed in the normal manner.

This procedure avoids two or more users working on one drawing at the same time when in a networking environment**.**

If the program is unexpectedly interrupted and the files have not been closed before the interruption, the same situation will occur. If the user tries to re-open a file after an interruption he or she will get the message *file locked.*

In order to re-gain access to the drawing, the corresponding Lock-Files must first be deleted.

In order to delete these files, first enter the command. Next, the user must perform the standard steps for the selection of files (similar to the steps in opening files). The program will offer the extension **.flk** as the desired file type.

Delete the offending Lock-File, and the drawing file can again be opened.

Send Drawing(s)...

Commands: SENDFLX, SENDALL

The command SENDFLX allows you to send the current drawing via EMail.

The command SENDALL allows you to send all active drawings via EMail.

Print

 **Command: PRINT**

Start the output of a drawing on the printer/ plotter or plotter by selecting the option *Print ...* or by entering the command PRINT.

Select one of the options available in the options bar. These options allow the user to determine the size and position of the print zone or the area of the drawing to be printed.

View

The currently visible window of the drawing will be copied on the print.

Extents

This option determines the print zone so that all objects included in the drawing are covered.

Note that before using this option the drawing must be regenerated (Command REGEN).

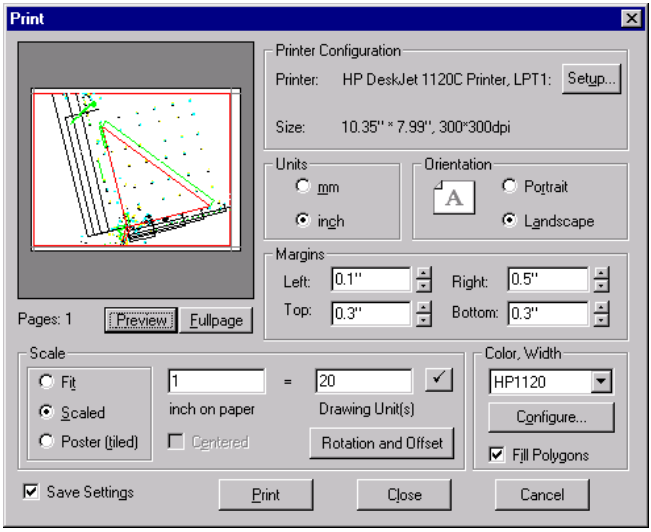
Window

Window enables the user to determine the print zone by selecting a window in the drawing. Indicate or enter the coordinates of two diagonal opposite corner points inside the drawing.

The command PRINT controls output of drawings to printers or plotters configured on your system. It uses the Windows print drivers and by this requires that the appropriate drivers have been set up under Windows.

The Print dialog box

After defining the size of the print, the dialog box *Print* will open, and additional print parameters required for the orientation of the print in relation to the edges of the paper format and to the picture size must be set.



The dialog box is divided into the areas Configuration, Layout, Color/ Width, Pages and Scaling. It also contains a preview window and selections for *OK, Cancel* and *Setup.*

The *preview window* will assist the user in controlling the effects of setting changes in Layout and Scaling.

Printer Configuration Area

The Printer/ Plotter configuration area contains the options for the current Printer/ Plotter and the desired paper size and print resolution (in dpi).

Setup Button

Selection of the *Setup* button allows the user to set parameters such as the paper feed (printer/ plotter drawer), paper size, format position, resolutions and the number of the copies to be run.

Note that in the SETUP Dialog Box only the settings can be considered and edited which the selected device supports.

Units and Orientation Section

The parameters of the Layout Area control the size and position of the print on the available format.

Margins Section

Controls the area that is not to be printed (in mm) at the sides of the page. Enter the desired values in the proper window or select with the Cursor at the right edge of the fields and set the values for margin width.

Scale Section

Selecting the option *Fit* will result in a format-filling picture of the object in the print zone.

This option is activated by default during any print or plot output. To de-activate, click the control field. An dot in the control field displays the active status of the option.

Note that when the option *Fit* is active, it is not possible to set the options for scaling.

Center (only available if you choose *Fit*)

Center has a very close relation to the option *Fit*. Because the relation between the pages of the print zone remains unchanged during *Fit*, a format-filling picture may only be reached either to the length *or* width of the picture, but not with both.

Center results in a centered placement of the print zone on the paper in relation to the side that could *not have been fitted* to the margins of the page.

Scaled

Scaled allows the user to define the exact scale of the drawing.

Note that *Scaled* is only available when the *Fit* option has been *de-activated*.

The definition of the picture scale is made in two steps:

1. Define the size of a drawing unit in mm or inches. Place the cursor in the input field for the drawing unit – mm on paper or inch on paper and enter the desired value. (Normally for Surveying or Mapping this is 1.)
2. In the input field Scale enter the scale for the print output or select a value by means of the arrow keys at the right side next to the input field. If you are working in Metric, then you would enter the scale factor in the Drawing Units(s) edit field. For example a scale of 1:250 would mean that 1 mm on paper = 250mm in the scaled distance. Therefore you would just enter 250.

3. If you are working in feet and inches, then 1 inch on paper would equal the plot scale in feet. So, if you are working at 1" = 40', then enter 40 in the Drawing Units(s) edit field.

The user can view the effects of the inputs in the preview window.

Poster (tiled)

Multiple pages will result when the user has selected a picture scale that results in a print output larger than the selected page size (minus the defined margins). The preview window displays how the drawing will be split over a number of pages.

Area Color / Width Section

By choosing the *Configure* button, the Color/Width Configuration Dialog will appear.

Configuration Colors / Line Widths

This section allows you to save configurations under different names. This is particularly useful if you have different clients who want different types of lines widths or colors. Some of the features you can modify in this section include:

- Use of Line Width settings
- Printing in Color
- Setting plotted colors for each screen color
- Setting individual line widths for each plotted color
- Setting the number of segments that will be plotted for arcs
- Saving the configuration under a specific name

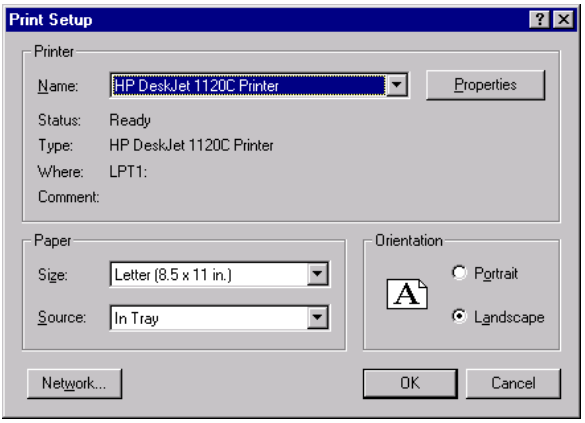
Fill Polygons

Fill Polygons will cause filled polygon areas to be displayed with fill in the print output. If the option not activated, only the contours of filled areas will be output.

The default position of this option is *Inactive*. To activate select the control field. A cross in the control field displays the *Active* status of the option.

Printer Setup

Command: PSETUP
The command PSETUP (Printer/ plotter Setup) or the selection of the option **Printer Setup...** from the *File* menu will start the selection and installation of the desired printer or plotter.



The procedure for this installation corresponds to the procedure for installation of a printer/plotter through the system control of Windows.

The user will find detailed instructions regarding the operating mode of the specific options for the printer installation and for installing and connecting a printer or plotter in the Windows Manual or in the output device manual. Only the most important steps for printer/plotter installation are reviewed in this text.
The command will open a dialog box where the user can make appropriate changes.

Dialog Control Elements

Printer

In the field Printer/ Plotter the user will specify the printer/ plotter or plotter to be used in the output of the drawing. Select either one of the standard devices defined in the Dialog Box control, or select another printer or plotter which is available under Windows. Output devices that are available are displayed in a list in the selection area. To see the complete list, select the arrow at the right edge of the window. Select the desired printer from the list of the installed output devices.

Orientation

Select the format **Portrait** or **Landscape** for the desired alignment of the paper.

Paper

The size of the paper as well as the feed from a particular printer or plotter is controlled from the field **Paper**. Select the proper size and feed.

Options

The **Options** switch (at the right edge of the Dialog Box) opens another Dialog Box that contains parameters that are mainly a function of the capability of the device to be connected.

Batch Processing

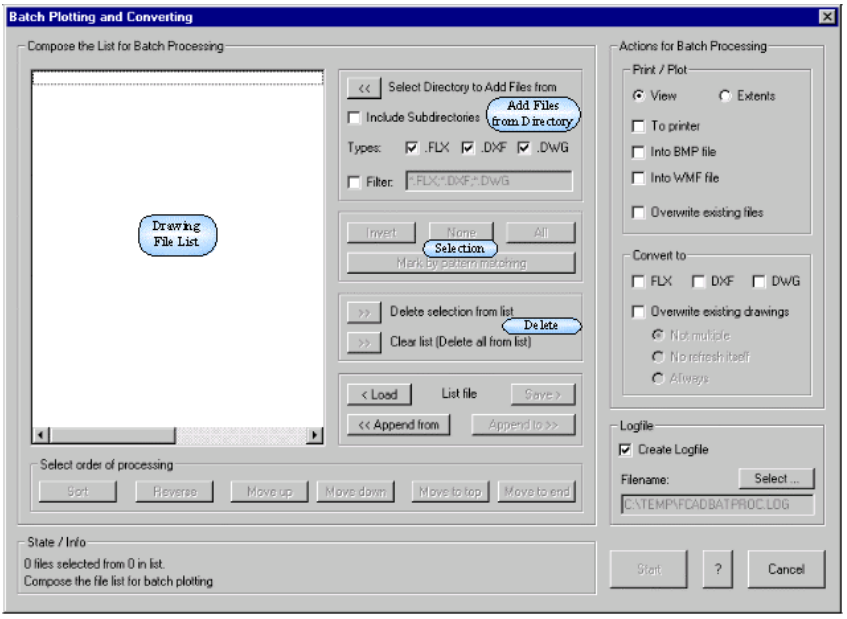
Command: BATPROC

Introduction

The command BATPROC makes it possible for you to arrange a list of drawing files and to do something with each drawing of this list. There are many built in capabilities to build the list and manipulate it. See the chapter Arranging the List for Batch Processing below. After arranging the list you can plot each file inside of the list not only to the printer but also to a Bitmap file (.BMP) file or into a Windows Metafile (.WMF). Moreover you can convert drawings from one format (of the set DXF, FLX or DWG) into each other of this set.

Preferences and Start of the Command

The first step consists of composing a list of drawing files. Therefore, the tool provides you with the ability to add files from a any directory. It's also possible to load or append files from a list. Of course, you can also remove any items from the list.



This dialog is divided in two main parts, on the left side you find the functions to compose and manipulate the list, while on the right part the actions for the batch process can be defined.

Adding Files from Directory

This field contains the ability to select a directory and add all files inside of the directory. Before pressing the button "Select Directory to Add Files from" you should setup the desired options. For that you can switch on the recursive search inside of subdirectories. Moreover you can select a filter for the files, you want to add to the list. Two different alternates can be selected here: The first and simple one by defining the desired set of extensions, while the second one provides the ability to apply a normal file filter that is conform to those of the system. Here you can also put in more then one file pattern separated by a space or a ';".

List Box

When the list of drawing files has been filled by adding files (see above) or by loading / appending an existing file list, you can select some files by marking them inside of the box. You can use the mouse in combination with the SHIFT- or CONTROL- Key to get a multiple selection. The selected files will be highlighted. Note that if you start the

processing, all files inside of the list but not only the highlighted files will be used.

Selections

After selecting or deselecting items of the list box you can change the selections in four ways:

- "All" will select all items inside of the list.
- "None" will deselect all selected items of the list. As a result, no item of the list will be selected.
- "Invert" will invert the selection. That means all selected items will be deselected, while all other will be selected.
- "Mark by Pattern Matching" will open a new dialog, there you can select or deselect items by applying one or more patterns to the list of drawing files. The following picture shows the dialog:

The meaning of the pattern matching is not the same as for DOS/ WIN - file names, but the same as the for wcmatch - function. See the "Customization Document" for more information about this.

Here is a short summary of the special characters:

*	=	any sequence of characters;
?	=	any single character;
@	=	any single letter;
#	=	any single number;
.	=	any non-alphabetical character;
,	=	separates patterns;
~	=	not the following match (works only at the first position)
[..]	=	any character inside
.-.	=	character in the range
[~]	=	any character, that is not inside of the brackets
‘	=	suppresses the special meaning of a character above

Delete List Items

Using these buttons, you can clear the list (fully clean up) or you can remove the selected items from the list. These functions do not delete any file from your file system.

List File

This field contains four buttons that serve to store or reload a list of drawing files.

After pressing the "Load" button, you can select a file, that should contain a list of drawing files. If you select one, the old list will be substituted by the list inside of the file.

"Append from" will do almost the same like "Load", but the files in the list file will be added to the current list.

Using the "Save" button you can store the current drawing file list to a list file.

"Append to" appends the current list to a file.

Note: You can simply generate a drawing file list using the old DOS-Command dir:

> dir *.flx /S /B > C:/TEMP/draw.lst

List Order

Five buttons will help you, to define the order of the processing:

1. "Sort" will sort the list in alphabetical order. If you apply this function, duplicate items will be removed too.

2. "Reverse" turns around the order of the items inside of the list box.

3. "Move up" will move all selected items by one item.

4. "Move down" moves all selected items down by one step.

5. "Move to top" moves all selected items up to the top of the list.

6. "Move to end" moves all selected items to the end of the list.

Note that all of these functions for changing the list order leave the selection unchanged.

Info/State Field

This field informs you about the process and the results of all list actions. At the first line you can see the current state – I.E. how many files are in the list and how many are selected. The second line shows you the progress of the current action, and if ready the result of this action.

Defining Batch Processing Actions

After composing the list, the actions for all items of the list have to be specified. The related fields are placed on the right side of the dialog box.

1) The Plotting/Printing Field

Field provides the following actions:

- Plotting to the printer.- Before you start the batch printing, it's necessary to select the printer and the options for printing. It's highly recommended that you open one drawing of the files that should be plotted and print this using the set options. These options apply to each drawing of the list.

- Plotting into an WMF-file (Windows Metafile)

- Plotting into a BMP-file (Bitmap)

With the radio buttons "View / Extends" you can decide if you want to plot the drawing like it's stored, or if it is zoomed to extents before plotting.

The switch Overwrite Existing files determines, if BMP and WMF files will be overwritten or not.

2) The Converting Field

This allows selection of the target format that can be FLEXIBLE DRAWING EXCHANGE (FLX), DXF or DWG. The AutoCAD formats will be written for release 12. It's possible to realize multiple conversions, i.e. each drawing can be converted to each other format at once. But, in this case it's highly recommended to switch off the overwrite option, because enabling overwriting of existing files can destroy the original drawings. If you switch on the overwriting option, the batch processing tool should be only applied to files, a backup exits

from.
The options for overwriting existing files have the following meaning:

The setting "Not multiple" prevents overwriting the file, if it have already written or if it is inside of the drawing file list, that will follow. As an example assume you have the composed a list, that contains both, a file conv.flx and a file conv.dwg. These files should be converted into both the DXF and the DWG format. If the unlimited overwriting would be set, the first file conv.flx would be open a saved as conv.dxf and as conv.dwg. So it would be overwrite the existing file conv.dwg too. Because this file is also in the list, it would be opened again... To prevent such cases it's highly recommended to use this option, if you activate overwriting of drawing files. The technical background of this option is, that for each file that would be overwrite another one is checked, if this file have already be written by the batch process or if it is part of the remaining list. In this case the file won't be overwritten.

The option "Not refresh itself" only prevents overwriting.
The option "Always" allows you to overwrite each file - itself too.

3) The Log File Field
This allows you to define, if a log-file should be written during the batch processing. Moreover, you can select a filename for it. If this already exists, it will be overwritten. It's useful to leave this option activated, in other case after finishing the batch processing there is no information about any error available.

Layout Mode-Paper Space

One main advantage of working with CAD is that any plan and construction can be created in real world units. Nearly any construction, however, needs to be printed on paper. Usually this means plotting it to a certain scale.

For that reason, at a certain state during construction, you will want:

- To add title blocks to your drawing,
- To display different views and regions,
- To add model details,
- To lay out part lists, legends, and annotations and instructions on the drawing sheet.
- To help solve these tasks in a flexible manner, the program provides two modes to handle a drawing: Model Mode and Layout Mode. In other words, there are two separate spaces to work on a drawing: Model Space and Paper Space (or Print Space).

The drawing objects in model space constitute the geometry of the model. The objects of paper space reproduce the model in different views and paper space assists when annotating the drawing.

Notes:

1. Layout mode is a complex environment and not always easy to work in. Users are therefore advised to become familiar with layout mode practices and concepts through studying the documentation, before using it on mission-critical project.
2. If you create smaller, purely 2-D drawings, it is often easier to inset (on separate layers), separately scaled title blocks, part lists, and details in model space, rather than doing this work in layout space.
3. When using layout mode you should always keep in mind that layout mode is primarily intended for the arrangement of complex drawings in preparation for plotting. The true benefits of this occur when you have a drawing model that must be spanned by multiple plots, or when multiple views of your drawing model must be presented on a single drawing sheet.

Techniques and Tips for Layout Mode

Create your drawing (in model space) in real world units (at scale 1:1). Utilize specific layers for dimensions, hatching, text, etc. When managing and using layers keep in mind, that you might want to illustrate different "logical views" to your model later when presenting or plotting your construction with the aid of layout mode.

At a certain state of the construction process, you will find the need to print out the drawing to present it within your company or to forward it to a manufacturer. In this situation layout mode can help in many cases, to achieve fine paper work:

1. Switch from model space to paper space using the command TLMODE. The title bar now says Layout: followed by the drawing name.
2. Set the limits to equal the paper size in those dimensions (units), which will be used when printing (for example A-size, B-size, C-size, ISO-A0, ISO-A1, ISO-A2, ... Letter Portrait, Letter Landscape).
3. Create a layer for the title block (if you have one to use), set it current, and insert your title block in paper space at coordinate origin (0,0). Or, draw a rectangle on a temporary layer with the paper size you will use when printing.
4. Specify a layer to place the viewports on. Make this layer current before you create new viewports with the VIEWPORT command and make sure to use this specific layer only for viewports. It is recommended to activate the snap mode with applicable units (inch, millimeter, centimeter, ...) when creating viewports.
5. The next step, once a viewport has been established, is to set up scaling between model and its representation on the drawing sheet. This is the step that sometimes causes a headache. The task, to set up the adequate scale for displaying the model, can be prepared already in model space in several ways, especially by:

    a. Positioning rectangles on a temporary layer in model space, which meet to viewport windows in paper space at a certain scale;
    b. Creating named views in model space with the command VIEW, which then can be retrieved within the viewports in paper space with the option View.

6. Once the scaled relation is set up, you should avoid uncontrolled zooms in the viewport, and confine yourself to Pan commands

and to the ZOOMFAC command (usually specifying whole numbers).

To modify the height or length of a viewport, we recommend to use the command STRETCH or SCALE. The display of the model in the specified viewport will not be scaled with this command. It maintains the scaled relation between model space and paper space.

Keep in mind the "golden rule":

Drawing entities you create in model space can only be edited in model space. Drawing objects you create or insert in paper space can only be modified in paper space.

Layout Viewport Control

Command: VIEWPORT

The command VIEWPORT command is used in layout mode to create rectangular windows that display the drawing model that resides in model space. In these viewports, you can control both the view direction and the display of a specific portion of the model. The VIEWPORT command options are: New, ON, OFF, Layer, and View. These functions are described in detail, below.

Creating Viewports

After switching to layout mode for the first time, usually you will create one or more viewports to show one or more views to your model.

Before creating a viewport, you must ensure that you are first in layout mode. At the top of the desktop window, you should check to see that “Layout” appears before your current drawing name, as in the above illustration. The command TLMODE toggles the display between layout and model mode and may need to be executed if you are still in model mode.

The option *New* of the command VIEWPORT predictably allows you to create a new viewport. New is also the default option of the VIEWPORT command. Next, you are asked to specify two opposing corners for the new viewport. If you have determined the size of the viewport that you need, you can either enter defining corner coordinates, or pick the corners of a rectangle drawn previously in layout mode, for this purpose. After specifying two corners of the viewport, the entire model is displayed in plan view and to its current extents, in the newly created viewport.

> VIEWPORT
First corner: **10,10**
Opposite corner: **@45,25**

Then the next step will be to determine the size of the content of viewport to match to a certain scale and to maintain the accuracy of this scale between model space and paper space within these viewports.

Detailed hints and tips to apply appropriate techniques to set and keep a scaled relation between paper space viewports and model space are described later in this section after all the options of the command VIEWPORT have been discussed.

Viewport Alteration using Modify Commands

A Viewport is a drawing entity represented by an rectangle and contains a certain view of the drawing. Theoretically, any number of viewports may be created in layout mode.

Several editing commands can be performed on viewport entities, because these are treated as drawing objects:

- You can move a viewport to another position.
- You can change the size of a viewport with the Stretch or the scale command. In these cases, the view inside the viewport is not scaled with the viewport, only the frame, itself. If you scale a viewport, however, the displayed view inside of the viewport is scaled with the same factor as the viewport frame.
- You can remove viewports from the drawing, using the Delete command and picking the frame.
- Using_Drawing_Windows may be copied.
- Note also that, the commands Rotate, Array, Flip, and Mirror modify the position of a viewport (respectively a copy of a viewport), but the edges of the rectangle remain parallel to the X- and Y-axis of layout mode. Also, these commands maintain the displayed view in the rotated or copied viewport.
- When editing viewports and when drawing in paper space you can snap to the end point, mid point, etc. of the viewports rectangle, and you can utilize the coordinate filter options.

Command Options:

On/Off - Turning viewports off and on

The Viewport command options **On** and **Off**, control the visibility of the contents displayed in existing viewport windows. To turn the visibility of

the viewport objects off, choose the option OFF and select the viewport frame or pick to an object inside the frame. To redisplay the representation of the model within a deactivated viewport, choose the option ON and pick an edge of the corresponding viewport frame.

The **visibility of the viewport frames** can also be controlled through layers. Like to any other drawing entity a viewport entity is located on a layer. It is recommended that you place viewport entities on one or two specific layers (for example LM_VIEWPORT_ON and LM_VIEWPORT_OFF). Now you can control the visibility of the frames of viewports by turning off or freezing a corresponding layer containing viewports.

Layer - Control of viewport contents

The VIEWPORT command option **Layer** permits manipulating the display in specific viewports by freezing and thawing drawing layers. This means that it is possible to have layers frozen and invisible in one viewport, yet unfrozen and viewable in another.

After choosing the option *Layer* the dialog box to control layer visibility in layout mode is displayed:

The list box on the left-hand side the dialog box contains all layers defined in the current drawing. The other list box is meant to determine those layers not to be displayed in the currently selected viewport ("Frozen Layers"). You can alter the visibility mode by marking layer names in one of the two lists and add or remove them from the list of frozen layers with clicks to the > button or < button. To freeze or thaw the status of entire layer groups, you can utilize the edit boxes by specifying strings with wild cards (for example "2D_*") and than click to the Filter button that will highlight all matching layers in the appropriate list.

View - Control in viewports

The VIEWPORT command option **View** toggles the layout window into a mode, where the selected viewport is displayed such that it covers the entire window. This mode serves to change the view to the model inside of the specified viewport.

> VIEWPORT
> First corner: **View**
> Select viewport: ***Specify viewport***
> Switch to viewport view ...
> Viewport control:

When calling the option *View* the prompt "Viewport control:" is displayed. The following view options are at your disposal: Zoom, Pan, 3DView, View.

Zoom

By intention the default prompt at the command line after choosing the Zoom function says "Zoom Scale factor: ". This will remind you that - once a scaled relation between model space and paper space has been established - you should manipulate the view within the viewport only via scaled zooming (usually by specifying whole numbers) to maintain the scale. In addition, however, the option bar contains the other zoom options (Window, Extents, Previous) as usual.

Pan

The option *Pan* solely moves the visible portion within a viewport to a direction specified by two points without changing the scale of the model representation in paper space.

In *Viewport control* mode any transparent command may be used. By this, you can also utilize the pan functions from the control panel (Pan Left, Pan Right, Pan Up, Pan Down); for example:

Viewport control: **'PANLEFT**

3DView

Naturally, the layout mode can be utilized also to illustrate the model from different viewpoints in space. The option *3DView* of viewport control allows you to set individual 3D views or predefined views like plan view, front view, left side view.

The option *3DView* will open a dialog box similar to the one invoked by the command 3DVIEW in model space.

Note: Using the option *3DView* will zoom the contents of the viewport to the drawing extents. To achieve a scaled relation between paper space and model space for isometric or other individual 3D views of the model, it is recommended that you use named views prepared in model space, as discussed next.

View

The option *View* allows you to display views in specified viewports, that have been saved as named view in model space. Layout mode efficiencies can be improved by saving named views in model mode designed for reuse in the viewports of the paper space. This is especially true if maintaining a scaled relationship between model and print space is required. See the section *Techniques and Tips for Layout Mode*, for further hints and recommendations.

The option *View* will open a dialog box similar to the one called by the command VIEW in model space.

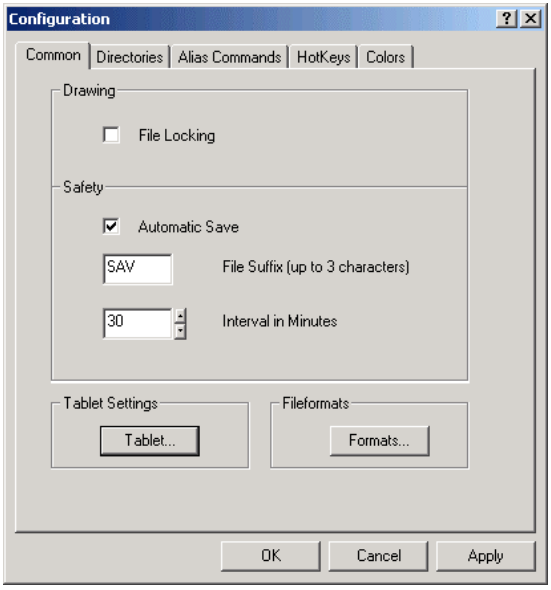
A particularly useful help feature is provided with the Show button in the Save/Retrieve named views dialog box, because this option allows you to preview a selected named view in the drawing before confirmation.

Note: When calling the command VIEW in layout mode outside the command VIEWPORT, you can create named views within paper space (for example to zoom fast to portions of the drawing sheet).

Utilizing transparent commands

You can utilize in *View* control mode all transparent commands. As already mentioned, you can call the functions 'PANLEFT, 'PANRIGHT, 'PANUP, 'PANDOWN from the control panel. Another example is the transparent command 'SETVIEWDIR to set an individual or standard isometric 3D view within a viewport. Or, the transparent command 'DIST that allows you to measure the distance between two points.

CAD Configuration Options



Command: CONFIG

By calling the CONFIG command, the Configuration dialog will appear.

Note: The changes will only become effective once the Apply button has been selected. No changes will become effective if you close the dialog window by pressing either the Cancel button or ESC key. See the on-line help for more information.

Palette Manager

Command: PALMAN
The user may control palette opening and display and the exchange of palettes on the program surface. Select *Palette Manager* from the menu *File*.

In the selection area of the Dialog Box, all of the palettes which are currently at the user's disposal are shown.

Select the desired palette. Underneath the selection area are three switches with the designations Palette 1, Palette 2, and so forth, to Palette 10. By selecting one of the ten switches the user can assign an identification number (for example, Palette 1) to the palette. If there is another palette with the same ID-number already opened, this palette will be closed and replaced by the new palette having that ID number.

When the palette is opened the selecting window will close. If no palette is to be opened or replaced the user may close the window by selecting *Cancel.*

If the user wants to include customized palettes in the Palette Manager she/ he should proceed as follows:

Edit the file <*Menu name*>*.pal,* which is situated in the \APPLIC directory. For instance, if the user menu is called *fcad.mnu*, the user must edit the file *fcad.pal* . The structure of the file is:

Palette description 1
palette1.mnp
Palette description 2
palette2.mnp
and so forth.

The palette description will be found in the list shown in the Palette Manager Dialog Box, and the line following will always name the palette file. The user can complete or delete the entries as he or she wishes.

Resource Manager Options

The Resource Manager options allow the user to use different customization commands.

Select *Resource Manager* from the menu *File*.

The cascading menu contains the following commands and functions:

Run Dialog and Menu Editor...	Calls the MicroSurvey Dialog and Menu Editor
Run Bitmap Editor...	Starts the Microsoft Paint Bitmap Editor to create, modify or view a bitmap file (.bmp)
Palette...	Loads a palette (toolbox) specified in the *Palette* File Dialog Box to the desktop (.mnp)
Menu...	Sets a pull-down menu specified in the *Menu* Dialog Box as current menu (.mnu)
Load Lisp File...	Loads a specified Lisp file (.lsp)
Run Macro...	Executes a specified Macro File (.mcr)
List File...	Displays the contents of resource text files (.lin, .pat, etc.)
Editor...	Calls the Windows Text Editor (Notepad.exe) to edit a resource file

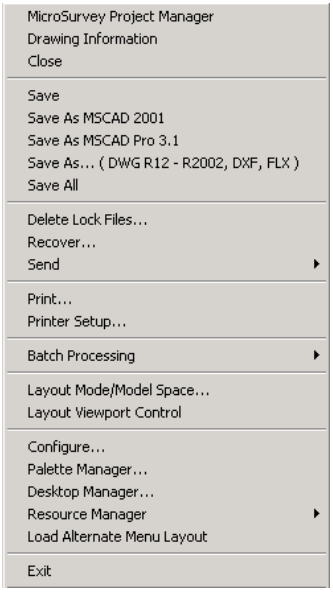
Load Alternate Menus

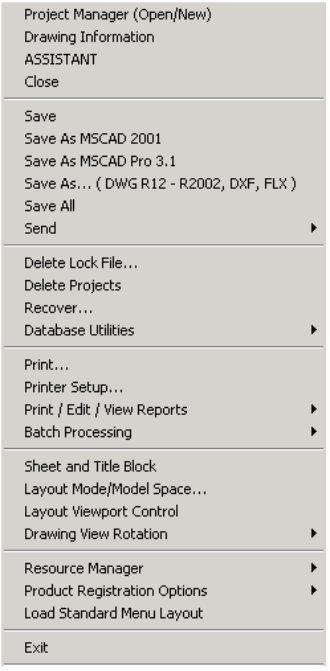
MicroSurvey contains an alternate menu set that some may find more convenient than the standard pull-down menus.

The standard default menus are called **METERS.MNU** and **FEET.MNU**, and will be loaded automatically when you set the units for that drawing. The alternate menus are called **METER-N.MNU** and **FEET-N.MNU**. To have these menus load automatically, instead of the regular menus mentioned above, you would have to rename the standard menus and then rename the alternate menus.

Example: **rename METER.MNU to METER-OLD.MNU** then **rename METER-N.MNU to METER.MNU**. Do the same with the imperial menu and then the program will automatically load the new menu designs.

We have a menu command that will allow you to switch between the two menu systems just by picking the command. Below are the menus and commands to pick (to switch both ways). **Try them out!**





You can also load a menu temporarily by typing MENU and then locating the correct menu to load. The menus are found in the \MSCAD2002\COMMON directory. If you do not rename your menus as shown above, and choose to simply try them out by loading manually – be aware that some commands will automatically reload the original menus. You can always type MENU again and reload the menu, or you can rename the menus permanently as alternatives.

The New overhauled menus are setup to optimize the grouping and access of the commands. It should be easier to find similar commands now that they are all grouped together. The only problem is that the Help file will not show you where the commands are in this new menu so it may take a short period of time to become familiar with the new

locations. Once you have become familiar I think you will find it hard to go back to the old menus.

Please let us know what you think of the new style menus

Exiting the Program

Command: EXIT
It is very important to end every working session correctly. The only correct means of terminating the session is by executing the command EXIT in the menu *File*.
The command EXIT is correctly performed in any of these fashions:

- Select the command Exit in the pull-down menu File and confirm.
- Press the keys Alt + F4

If there are open drawings that have not been saved, the program will ask you if you wish to save your drawing.

Edit Menu

Undo

 **Command: U**

Invoking the command U (for Undo) will cause the last command made to be withdrawn and all changes in the drawing caused by this command to be canceled.

The command can be repeated until all working steps stored in the Undo List have been canceled. The program will then send out the message "Undo List is empty".

Note: The command U allows only to withdraw drawing database modifications. It does not undo modifications of the view. Use ZOOMBACK to restore previous views.

Redo (Restore an Undo Operation)

 **Command: REDO**

The command REDO offers the ability to Redo an Undo. The state of the task is restored to the last state prior to issuing the command U (or UNDO).

This is only possible if no other command has been executed between the commands U (or UNDO) and REDO which have changed the drawing database.

REDO is used in this fashion:

- Call the Redo function by clicking the Redo symbol, or
- Enter REDO at the command line.

Note:

Only the last relevant working step can be restored.

Delete

 **Command: DELETE**

During the creation of drawings, or, later, the editing, changing or detailing of drawings it may be necessary to delete certain parts of the drawing.

Call the command

- By selecting the option **Delete** in *Edit* menu; or
- By clicking the Delete symbol from the *Edit* palette;
- By entering DELETE in the command line area.

After calling the command the request to select the objects to be deleted will appear. The program offers (in the option bar) a series of methods for object selection.

For a deleting operation as many objects can be selected the user desires. When the object selection is complete the selected entities will be deleted without another confirmation, by simply pressing ENTER.

Undelete: Restoring Deleted Items

 **Command: UNDELETE**

Entities which have been deleted by mistake can be restored by the command UNDELETE.

Call the command

- By clicking the UNDELETE symbol;
- By selecting the option **Undelete** in the menu *Edit*; or
- By keyboarding UNDELETE in the command line area.

This will cause previously deleted entities to be are restored.

Purge

Command: PURGE

The PURGE command cleans the drawing file of unused named objects. You can remove blocks, layers, linetypes, text styles, or dimension styles that you have defined or loaded before but are not utilized in the drawing.

The command line prompt of the command is as follows:

> PURGE
Purge un-referenced symbols (Blocks, Dimstyles, LAyer, LTypes, Textstyles or All):

The PURGE command can be utilized on drawing database tables and table entry names *selectively*:

- You can use the command for deletion of unused named objects within only one of the drawing database tables. To do so, first select the type of table to be purged. You will then be prompted with the names - one after the other - of unused named objects of the specified table.
- If you want to purge all drawing database tables, reply *All* to clean up the entire drawing in one operation. You may respond with a name pattern to limit the selection to certain names by using wildcard specifications (for example: TEMP*).

To purge a drawing file

To purge all un-referenced symbols in any drawing database table respond as follows:

> PURGE
Purge unreferenced symbols (Blocks, Dimstyles, LAyer, LTypes, Textstyles or All): **ALL**
Names to delete <* = All>: *****
Confirm each name? <Y>: **N**

Notes

- Purging a drawing can significantly reduce its file size. This operation is a good habit before archiving or backing up a drawing file.
- Some standard objects cannot be purged: These include the layer named 0, the linetype CONTINUOUS, and the text style STANDARD.
- The PURGE command does not handle named views and user coordinate systems, because these named objects are never referenced by another component of the drawing. You may delete unused views and UCS. The corresponding commands (VIEW and UCS) provide options to delete named items from these tables.
- With nested blocks in the drawing file, only one level of nesting is purged at a time.

Copy Selection to Clipboard

 **Command: COPYSELECT**

The command COPYSELECT copies selected objects from the drawing and places them onto a MicroSurvey specific clipboard. The selected objects remain in the drawing.

The command COPYSELECT is useful especially to copy objects from one drawing to other drawings. COPYSELECT places these objects onto the clipboard. Use the PASTE command to paste them from the clipboard into other drawings.

To copy objects onto the clipboard:

1. First specify the base point for later insertion of the objects.

2. Then select the objects to be copied to the clipboard.

Command line example:

```
> COPYSELECT
Insertion base point:
Select objects:
```

Note:
Do not confuse this command with the COPY command found in the *Modify* menu that makes copies of objects in the current drawing. The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another drawing. They will remain on the clipboard until you place new objects on it with the COPYSELECT command.

Cut Selection to Clipboard

 **Command: CUTSELECT**

With the command CUTSELECT selected objects are copied to a MicroSurvey specific clipboard (like with the command COPYSELECT).

However, unlike with COPYSELECT, the selected objects are deleted.

The command PASTE will insert the objects from the clipboard into any open drawing. PASTE will request the insertion point.

Paste Selection from Clipboard

 **Command: PASTE**

The command PASTE copies objects from the MicroSurvey specific clipboard to a user-defined location in the current drawing.

Use the command to copy graphic data from other drawings into your current drawing. You can copy objects from a drawing to the clipboard with the COPYSELECT command, then paste copies into other open drawings.

To paste the objects from the clipboard into the current drawing:

1. Choose the PASTE command.
2. Specify the insertion point.
3. The objects of the clipboard contents appear highlighted, attached to the cursor. You drag the highlighted objects to position them in the drawing.
4. Specify the scale factor for insertion
5. Specify the rotation angle

Example:

> PASTE
Insertion point:
Scale factor <1>:
Rotation angle <0>:

When inserting the objects from the clipboard you can also specify the scale factor and/or rotation angle before you indicate the insertion point by choosing the appropriate option from the context bar when prompted for the insertion point:

> PASTE
Insertion point: **Factor**
Scale factor <1>: **.5**
Insertion point: **Rotation**
Rotation angle <0>: **90**
Insertion point:

Note:

The objects selected with COPYSELECT remain on the clipboard, ready to be placed in another drawing, until you place new objects on it with the COPYSELECT command.

To copy objects within the current drawing it is recommended to use the COPY or ARRAY command found in the *Modify* menu.

Copy Region to Bitmap

Command: BMPOUT

Call the command either by keyboarding BMPOUT or by selecting **Copy Region to Bitmap...** from the menu *Edit*.

The following options allow the region to be copied into a bitmap:

Window
Define a window by setting two points.

View
The complete current view will be exported into a bitmap.

Extents
The region to be copied is defined by the extent of the drawing.

Next, specify the **Width** and **Height** of the bitmap in Pixel Units (10...2000 pixels).

Finally, you may choose whether the bitmap should be written into a **File** or copied to the Windows **Clipboard**.

Note

The bitmap remains in the clipboard until you clear it, or place new objects on it with the RENDER, BMPOUT, WMFOUT or other command, or Cut or Copy in other Windows applications.

Copy Region to Windows Metafile

Command: WMFOUT

Call the command either by keyboarding WMFOUT or by selecting **Copy Region to Windows Metafile...** from the menu *Edit*.

The following options allow the region to be copied into a WMF vector format file:

Window

Define a window by setting two points.

View

The complete current view will be exported into a Windows Metafile.

Extents

The region to be copied is defined by the extent of the drawing.

Finally, you may choose whether the picture should be written into a **File** or copied to the Windows **Clipboard**.

Note:
The WMF vectors remain in the clipboard until you clear it.

Change Object Display Order

Command: DRAWORDER

DRAWORDER changes the drawing and plotting order of any object in the MicroSurvey drawing database. In addition to moving objects to the ”front” or ”back” of the sort order, you can order objects relative to another object (that is, above or below a selected object). DRAWORDER provides the following options:

Above	Under	Front	Back

- **Above** - Moves the selected object above a specified reference object.
- **Under -** Moves the selected object under a specified reference object.
- **Front -** Moves the selected object to the top of the drawing order. The selected object becomes the last drawn entity in the drawing database.
- **Back -** Moves the selected object to the bottom of the drawing order. The selected object becomes the first entity in the drawing database.

Differences between MicroSurvey and AutoCAD

Different object sort methods are not supported in MicroSurvey. The object order is changed in the database directly, instead, using DRAWORDER. A SORTENTS system variable for controlling the object sort mode is therefore not required.

Modify Object Properties...

Command: PROPEDIT

The function **Modify Properties...** invokes the dialog boxes that allow you to set the properties of entities, and make changes in **Color**, **Layer**, **Linetype, Thickness, and Linetypefactor.**

The function is found in the menu *Edit* and may also be entered by keyboarding the command PROPEDIT.

As with the other commands in this set, first select one or more objects.

Then press RETURN, or use the right-hand mouse button.

The dialog box that appears displays the current properties Color, Layer, and Linetype of the entities selected. You may then edit these properties.

The Thickness allows you to set a vertical height for lines, arcs, circles and other entities. This is useful if you want to represent the walls of a building.

The Linetypefactor allows you to set different LineType Scales for individual lines.

Copy Object Properties

Command: COPYPROP

This command allows you to change the properties of selected objects to those attached to a specified reference entity.

Only the properties that are common to all objects (layer, linetype, color) are transferred to the selected objects.

To change the properties of a selection:

1. Select a single reference entity.
2. Select one or more objects to be changed.

Command line example:

- COPYPROP
  Select entity for reference properties:
  Select objects to modify. Select objects:

Modify Object Layer ...

Command: PROPLAYER

This command transfers selected objects from their current layer to a chosen layer. Call the command keyboarding PROPLAYER or by selecting **Modify Object Layer...** from the menu *Edit*.

To modify the layer property of drawing objects:

1. Select the drawing entities to be transferred to another layer.
2. In the dialog box choose a target layer and confirm the selection by OK.

Explode Complex Objects...

Commands: EXPLODE, XPLODE

Symbols and other complex objects, like polylines, crosshatches, and dimensioning, inserted in the drawing constitute one single drawing object.

In this mode, it is not possible to edit any single one of the entities that make up the block. If, for any reason, there is a requirement to edit one of the entities comprising a block, then the block must be exploded (broken into its individual entities).

The explosion of complex objects is executed by the command EXPLODE, or by selecting and activating the option **Explode Complex Objects...** in the *Edit* menu.

When the object selection is complete, the objects (Symbols, polylines, hatches, or dimensions) will be exploded.

Now you may choose any of the entities for individual editing.

Note:

It is important to realize that, if there is attributes attached a block to be exploded, these attributes will be lost. Instead, the attribute names will appear.

Entity Info...

Command: EINFO

The command EINFO lists information about the selected objects in the command line.

Drawing Xplorer...

Commands Command: XPLORER

The Drawing Navigator (Xplorer) allows you fast and simple access to important information on the currently open drawing(s). Furthermore,

you can use the Xplorer to alter various properties of the current drawings.

The Drawing Navigator is a tool to manage the named objects of the drawing database tables. You can view the tables and table entries of layers, linetypes, textstyles, views, user coordinate systems, and blocks.

Type in the command XPLORER to activate this tool or select *Drawing Explorer* from the *Edit* menu.

The *Xplorer* window displays the preview bitmap of the currently open drawings in the area underneath the menu bar. To switch to another drawing table, click the appropriate bitmap. A ToolTip over the corresponding bitmap displays the drawing name including drive and path.

The left-hand list box titled *Table* serves to select and access a specific drawing database table. The table entries with all its named objects and characteristics will then be displayed in the right-hand list box.

In detail, the drawing navigator permits the following:

- Delete unused, not referenced table entries (allowed only for the active drawing)
- Rename table entries (allowed only for the active drawing)
- Set the current layer, current linetype, current textstyle, current user coordinate system, or current named view (allowed only for the active drawing) by double-clicking to one of the table entries
- In case the *Block* table is the current one, you can insert a block by double-clicking to its name.

Clicking the *Tool* button will invoke the main corresponding function associated to the active table.

For example, if *Layer* is the Drawing Navigators current table, you will call the Layer Manager dialog window by clicking the *Tool* button. This way, you might create new layer entries.

Identify Coordinate

 **Command: 'ID**

The transparent command 'ID (identify coordinate) determines the coordinates of a point to be indicated displays that information.

The function **Identify Coordinate** may also be selected from the menu *Edit*.

Using the command

Indicate the point to be determined with the Cursor. If it is a geometrically significant point of a drawing object (for instance,. corner point, center of a circle, intersecting point of two lines etc.) use the Object Snap functions.

The coordinates will appear in the display.

➢ ID
Point: *P1*
X: 1.27 Y: 4.45 Z: 0.0

Distance / Angle

 **Command: 'DIST**

The transparent command **Distance / Angle** will determine the distance or angle between two points and display them in the text window:

- The absolute distance (range);
- The Distance on the X-, Y- and Z-axis;
- The Angle on the XY-plane; or
- The Angle from the XY-plane.

The points may be arbitrary points that do not have to be part of a drawing object.

The function Distance / Angle operates as follows:

- Determine the two points required by selecting them with the cursor, using the Object Snap functions or by input of the coordinates; then
- The distance and angle between the two points will be displayed.

The term *Distance* is understood to mean the absolute distance between points.

The terms *Delta X*, *Delta Y* and *Delta Z* designate the distance between the two points in the direction of the respective axis of the coordinate system. All length values are given in drawing units.

Angle in the XY-plane measures the angle to the second point starting from the X-axis. It always displays the lesser-valued angle, no matter what the rotation direction.

Angle from the XY-plane measures the angle between the first and the second point from the XY-plane to the Z-axis. It is assumed that the first point lies on the XY-plane.

NOTE: If you use this command and snap to 3D coordinates, the resulting distance will be a Slope distance - not a Horizontal distance. See the 2D Inverse command for information about a more powerful routine for Surveyors.

Area / Circumference

Command: AREA

This function defines the circumference and the area of one or more drawing objects and displays the values. For objects that do not have width, the length instead of the circumference will be displayed. The areas of single objects can be added or subtracted.

Using Area / Circumference

To determine the circumference and the area of one or more objects proceed as follows:

Define the drawing object to be examined. Determining the circumference (or the length) and the area of an object is done by defining points which describe the geometry of the object. The preferred method of selection is by using the cursor and Object Snap functions, but inputting the coordinates of the points is also okay.

Pressing RETURN at the request **Next Point:** will finish the selection.

Object: Regular Areas

Regular geometric areas (of circles, arcs, closed polylines) can be evaluated by choosing the **Object** option.

Add or Subtract Areas

Select the first partial area or determine the points for describing the area geometry. The determined values will be displayed in the text window.

Select the option *AddArea* to add a partial area, or *SubArea* to subtract a partial area, from the total area.

The values (circumference /length and area) for the total area will be displayed in the text window.

When partial areas are added or subtracted from an area, the total area after the addition or subtraction is regarded as the new parameter value unless the user explicitly selects a contrary option from the Options Bar.

The Options Panel will display only that option which is not active at the moment.

Draw Menu

See the on-line help for an explanation of the prompts for these commands.

Line

 **Command: LINE**

The **Line** command is used to draw lines or line segments or connect lines to existing lines or arcs. Inputs to the command are two points; the starting point and the ending point of the line.

Circle

 **Command: CIRCLE**

There are several ways, geometrically, to describe a circle. The program allows the user to choose from among these ways, and draw the circle in the most convenient manner.

Arc

 **Command: ARC**

There are several ways, geometrically, to describe an arc, just as with a circle. The program allows the user to choose from among these methods, and draw the arc in the most convenient manner.

Polygon (N-gon)

 **Command: NGON**

The command **N-gon** is used to draw regular polygons. It is possible to draw polygons with up to 1024 sides, a value which should lie above any practical need.

Trapezoid

 **Command: TRAPEZOID**

With the command TRAPEZOID it is possible to draw a single conical element (trapezoid) on the active layer.

Ellipse

 **Command: ELLIPSE**

The command ELLIPSE draws a full ellipse.

Chain

 **Command: CHAIN**

The Chain command is a combination of the commands Line and Polyline.

The Chain command makes it possible to draw continuous lines in a manner similar to the command LINE using the option *Continuous*. The procedure is the same as drawing continuous lines.

Polyline

 **Command: POLYLINE**

Polylines are drawing elements that consist of connected lines or connected arc segments. Different elements can be of different widths. The width of a polyline segment can be defined differently for the starting- and end-point.

A polyline is defined as **one** element and treated as such for further operations (like **Edit**, or **Undo**), in contrast to continuous lines drawn with the command **Line**.

Polylines will always be drawn with solid fill.

Polyline Contour

 **Command: PCONTOUR**

This command creates a closed polyline on top of (overlapping) objects, which make up a closed boundary. Such a polyline contour can serve for hatching regions of the drawing with the HATCH command or can be used to determine the area or circumference of a region with the AREA command (applying the option *Object*).

Filled Faces

 **Command: 2DFACE**

2D Faces make it possible to draw solid filled planes that have straight-line borders. This is done by creating and connecting any number of

three- or four- cornered planes. It may be done continuously or using segments. It is also possible to connect 2D planes to already existing 2D faces.

Rings

 **Command: RING**

Rings are a special case of a closed polyline. This command is used to draw solid rings or circles. Two concentric circles are drawn, the smaller one is the inner diameter, and the larger one the outer diameter of the ring. The space between the circles is filled.

Rings can be edited using any of the editing commands for polylines and will behave just like a polyline object.

The last values entered will be used as an option for the next entry when drawing rings. If the user wants to keep using the same values, just enter a new center point. This step can be repeated as often as needed, always creating identical rings.

Construction Points

 **Command: POINT**

This command inserts construction points into a drawing.

The form and size of the points is determined by specifying parameters using the menu option *Drawing Parameters...* (See: Command DRAWMODE).

Measure and Divide

The commands MEASURE and DIVIDE order points or multiple blocks (Symbols) along a 2D-Object. This object can be an arc, a line, a circle or a 2D-Polyline. Before a Symbol (block) may be used with these commands**, it must already exist in the drawing database**. This may be accomplished, either through insertion or Symbol definition. When the command is given, the block can be inserted aligned, or not aligned to the object. Both commands work in a similar fashion. While the command MEASURE asks for a distance, DIVIDE requests the number of points and the distance between points or Symbols dependent upon the length of the linear entity selected.

Drawing Modes...

 **Command: DRAWMODE**

The Drawing Modes dialog box allows you to specify default settings for subsequent drawing commands. Parameters may be changed, activated, or inactivated.

3D

3D Polyline

 **Command: POLY3D**

3D Polylines are drawing elements created from straight-line segments. It is possible to define 3D coordinates for each individual 3D Polyline control point (start and end points of the individual segments).

The user may include curved segments in 2D polylines, but this is not possible with 3D polylines. It should also be noted that each individual element is assigned a standard width of zero.

As opposed to a line characteristic created using the line drawing command, the program treats 3D polygon characteristics as a single item, this also applies when the objects are being processed with the edit or undo functions.

3D Polyline may be accessed by selecting the option from the menu item **3D**, in the **Draw** menu, or by typing in the command POLY3D.

3D Faces

 **Command: 3DFACE**

The **3D Faces** drawing function creates three or four sided surfaces in space, or, in other words, in an X, Y, Z coordinate system. The difference between 2D and 3D surfaces is that the 3D surfaces are not shown as solid areas, but are drawn as wire frame models.

The 3D Faces definition points are fixed in either a clockwise or counter-clockwise direction so that one edge of a 3D surface is used as the base line for the next 3D surface.

The user may also draw invisible lines, a function which may increase the clarity of complicated 3D structures.

3DMESH

Command: 3DMESH

The 3DMESH command creates polygon meshes. Usually it is used to generate irregular surfaces, vertex by vertex. For example, it is used in surveying and GIS applications to create topological surfaces.

A 3D mesh is a wireframe rectilinear blanket composed of *M* column lines by *N* row lines passing through a matrix of *M* x *N* three-dimensional points in space. *M* and *N* are indices specifying the number of rows and columns that make up the mesh, and determine the number of vertices required in the mesh.

PFACE

Command: PFACE

The PFACE command creates three-dimensional polygon meshes of arbitrary topology and surface properties.

REVSURF

Command: REVSURF

The REVSURF command creates a surface by revolving a path curve (the profile) around an axis of revolution.

The curve sweeps around the selected axis to define the surface in the *N* direction of the mesh. The *M* direction of the mesh is determined by the axis of revolution. The result is determined by the SURFTAB1 and SURFTAB2 settings.

TABSURF

Command: TABSURF

The TABSURF command creates a tabular surface using a path curve and a direction vector.

You can extrude lines, arcs, circles, 2D or 3D polylines. The entity provided as path curve defines the surface of the mesh. The direction vector (extrusion path) must be determined by picking a line or an open polyline.

The system variable SURFTAB1 defines the number of *M*-direction surfaces on the extruded entity. When extruding a line, arc, or circle, the number of *M*-direction surfaces is evenly divided determined by the value of the SURFTAB1 system variable. When extruding a polyline,

extrusion lines are drawn at the ends of the line segments, and the arc segments are divided evenly into the number of surfaces determined by the value of the SURFTAB1 system variable.

If the polyline has been curved (fitted) or splined, the number of M-direction surfaces is evenly divided using the SURFTAB1 variable. Like with the command RULESURF, you only need to set SURFTAB1 system variable.

The resulting surface is a polygon mesh (stored in the drawing database as a complex polyline entity).

RULESURF

Command: RULESURF

The RULESURF command allows you to create a ruled surface between two boundary curves (two profiles).

Valid entities that define the edges of the surface can be points, lines, arcs, circles, 2D polylines or 3D polylines.

The system variable SURFTAB1 determines the number of equal intervals between the resulting vertices on the defining profile entities. The number of intervals is the same for each curve.

The resulting surface is a polygon mesh (stored in the drawing database as an complex polyline entity).

3D Objects

Below are the commands to create 3D objects. They are contained in a LISP program and are automatically loaded when you pick on one of the commands on the menu.

Follow the prompts on screen, to see how they work.

3DBOX - Drawing of a 3D box.

3DWEDGE - Drawing of a 3D wedge.

3DCONE - Drawing of a cone/cylinder/frustum/prism/pyramid etc.

3DSPHER - Drawing of a sphere

3DELLIPS - Drawing of an ellipsoid.

3DTORUS - Drawing of a torus.

Modify Menu

Move

 **Command: MOVE**

The command is used to move objects within the coordinate system.

The first prompt will be to select the object to be moved. Multiple objects may be selected, and they will be moved together in exactly the same relationship as they are currently placed. Select the object or objects, and confirm the selection by pressing RETURN.

Copy

 **Command: COPY**

Duplicating drawing objects

This function allows the user to copy drawing elements and put them at a specified place.

The copies created by this command are exact copies of the selected entities. All entity attributes such as layers, line type, color, and all other attributes are copied as well as the outline of the object(s).

Rotate

 **Command: ROTATE**

Rotate will serve to rotate an entity or entities through an angle, about an axis.

As with the other modification commands, Rotate requires the selection of the object/ objects to be rotated. There are several methods suggested by the Option Bar of selecting the object(s). Choose a method, and make the selection.

Scale

 **Command: SCALE**

The **Scale** command allows the user to change the scale of the objects, that is, to enlarge or reduce them while keeping all proportions the same.

Flip

 **Command: FLIP**

The command **Flip** contrasts to the **Mirror** command in that the original object does not remain in its original position; it is simply flipped about an axis specified by the user.

The selected objects have, after flipping, the same relative position to the symmetry axis as they had prior to flipping, but are now flipped in a new direction.

The reflection axis can be defined freely, or parallel to the X-axis or the Y-axis

Mirror

 **Command: MIRROR**

Mirroring drawing objects produces identical copies of selected entities, which are placed in a mirrored position about an axis specified by the user.

The selected objects have, after mirroring, the same relative position to the symmetry axis as they had prior to mirroring.

The mirror axis can be defined freely, or parallel to the X-axis or the Y-axis

In order to use the command the user must select an object to be mirrored. Select the object or objects, remembering that the selection of multiple objects will result in the treatment of the objects as one.

The next prompt will ask for the definition of the mirroring axis.

3D Editing

Below are the commands to edit 3D objects. They are contained in a LISP program and are automatically loaded when you pick on one of the commands on the menu.

Follow the prompts on screen, to see how they work.

3DROT - Rotating objects in three dimensions.

3DMIR - Mirroring objects in three dimensions.

Stretch

 **Command: STRETCH**

The **Stretch** command facilitates a change of position of selected entities, but allows them to hold their relation to other entities of an object. Using this command, entities forming the borders of the object may be stretched.

In drawing practice, stretching is useful for the enlargement or reduction of objects in only one direction (in contrast to scaling) or for moving single entities inside of one contour.

Like all the commands and functions for changing or editing, the prior selection of entities is prerequisite.

Careful planning must be done in the selection of entities for stretching. During stretching, the method of selection has an influence on the question of which entities are moved and which entities are stretched.

Array

Command: ARRAY

The command **Array** represents a variant of **Copy**, except that Array will create multiple copies, and they can be arranged at random, or in rows, or in vertical order, or on a circle, or on an arc.

The entities created by the command are exact copies of the selected entities in that all entity attributes such as layers, line types and color are copied.

Offset

 **Command: OFFSET**

Offset creates congruent pictures of lines, circles, arcs and 2D-polylines. This command enables a copy of a selected entity to be created and placed a distance from the original entity. The original entity(s) stays in position.

The operating mode of the command depends on the type of drawing entity:

For lines and polylines identical copies are created and moved in the direction of the X or Y-axis.

For circles and arcs copies are also created, but, in addition, they will be scaled by a selected factor. As a basepoint for scaling the center of

the original circle or arc is used. In this way, concentric circles or arcs are created.

Distance Offset

Command: DOFFSET

The DOFFET command is similar to the command OFFSET, but not identical. Like OFFSET, the command DOFFSET creates lines and curves parallel to an existing entity. DOFFSET has been added for convenience of those users who are familiar with other CAD programs.

Trim

 **Command: TRIM**

Trim is selected from the menu *Modify*, or keyboarded by typing TRIM. The Trim command is used to trim objects with a cutting edge.

The first prompt for this command will be a request to identify the entities to make up the cutting edge. These entities will be the ones that will result in the trimmed intersection of lines. Pick these entities with the cursor. Press Enter to continue.

The next prompt will ask to select the entity to trim. That selection should be the portion of the cutting edge entities that the user wishes to dispose of. Again, select this entity or entities with the cursor. As the entities are selected, they will be trimmed, and the trimmed portion will disappear from the drawing.

Expand

 **Command: EXPAND**

The command **Expand** allows the user to edit objects, which should have common boundary edges. Parts of objects can be expanded to the intended boundary edge. This command makes possible the precise closure of polygons, and the extension of entities to the proper boundary edge.

The command EXPAND is selected from the menu *Modify*, or keyboarded by typing the command EXPAND.

During the execution of the command the drawing entities to be expanded are extended. They do not change direction. A line will remain a line, an arc an arc.

Select the entities which are to serve as boundary edges for the entities to be expanded. These entities will remain unchanged. There are object selection functions in the Option Bar.

Then select the entities to be expanded. In doing this, make sure that the selecting window of the cursor is on the part to be expanded.

The expansion of selected entities is always executed from the endpoint which lies nearest to the cursor when it selected the entity.

If more then one boundary edge has been selected, the entity is expanded to the first boundary edge first. Picking a second time will result in the expansion being continued to the second boundary edge.

Fillet

 **Command: FILLET**

Often two solid edges do not intersect in acute or right angles. With the Fillet command solid edges can be filleted quickly and precisely.

The command **Fillet** may be chosen from the menu *Modify*, or may be keyboarded by typing FILLET.

The first prompt will request the selection the two lines whose intersection is to be filleted. These can also be lines of an object such as a rectangle.

Chamfer

 **Command: CHAMFER**

The command **Chamfer** is found in the menu *Modify*, or may be keyboarded by typing CHAMFER.

The first prompt will ask for the entities to be chamfered. Select those entities by picking them with the cursor.

Intersect

 **Command: INTERSECT**

Intersect is a special case of trimming that causes intersecting lines to be trimmed following the intersection.

Lengthen

 **Command: LENGTHEN**

Lengthen provides the ability to change the length of existing lines or arcs.

Unlike the command EXPAND, **Lengthen** extends an entity to a specific value or in a particular ratio (percentage value) of the original size.

Entities will be only lengthened in their existing direction. All entities will always be lengthened on the end, which lies closest to the point where the cursor was positioned during the selection.

Delete Partial

 **Command: DELPARTIAL**

Delete Partial will cause a part of a line, circle, or arc to be erased.

These entities can also be divided by this command. The result of such a division will be two independent entities which can be edited, or have other operation performed on it, separately.

Deleting Part of a Line

Deleting part of a line is the standard option for this command. To perform this deletion, determine the end points of the part of the line to be deleted. The line will be divided at this point, and the portion of the line between the two points will be deleted.

If the entity is a **circle,** that part is separated and deleted which lies clockwise between the first and the second dividing point. A straight segment of a **polyline** with a width greater than zero will be separated at a right angle to the line.

Separating Drawing Entities

To separate or break an entity you select the option **1Point** out of the option bar.

Determine the point on the entity where the separation is to occur. It may be wise to use the Object Snap functions to increase the precision of the input. The edited object will be cut into two parts that can be edited, or otherwise operated upon, separately.

A circle cannot be divided or broken by the input of a point. In this case the program will send the respective error message.

Break

Command: BREAK

The BREAK command is similar to the command DELPARTIAL, but not identical. Like DELPARTIAL, the command BREAK breaks objects at a point or points you select. BREAK has been added for convenience of those users who are familiar with other CAD programs.

The main difference is, that the command BREAK assumes that you might want to use the pick point when selecting the entity as the first point of the part to delete:

Rejoin broken Lines or Arcs

 **Command: REJOIN**

Rejoin will unite separated lines segments that remain aligned in one direction, or of arc segments that share the same center and the same radius.

This function will also reunite separated line or arc segments that have been divided by the command DELPARTIAL (Delete partial).

Change

Command: CHANGE

The command **Change** serves to edit of entities including lines, circles, arcs, text, attributes and parts with regard to their geometrical character or their attributes.

The command is called by typing the command CHANGE or by selecting **Change** from the *Modify* menu.

Parameters for Modify Commands...

Command: EDITPAR

The function *Modify parameters* allows the user to define a series of parameters, which are used as preset values for some editing functions. These parameters are mirroring of text entities, trimming and chamfering, and the radii of fillets.

Modify parameters is called by selecting that option from the pull-down menu *Modify* or typing in EDITPAR. A dialog box will appear.

Mirror / Flip

Mirror / Flip determines the position and orientation of text objects in the drawing.

Activate MIRROR / FLIP by checking the box MIRROR TEXT in that position from the dialog box *Edit Parameters*.

Activating the option will cause the text objects to be mirrored, with the original text object retained.

With the option de-activated, the text objects will be copied, but not mirrored, about the specified axis.

Offset Distance

Offset determines the parameter value for the distance in which an object will be offset when the command OFFSET is used. This value defines the distance of each point of a copy from the original drawing entity. There is more detail on the command OFFSET in this chapter. Enter the desired value in the input area.

Note that, during a working session the value used last will always be displayed as the parameter value for the next call to the command.

Fillet / Chamfer

This area decides whether the trim function is used by default when these commands are called, or whether a separate TRIM will be required.

Trim Mode

Trim Mode, when activated, will cause the erasure of those parts of objects which lie between, inside, or outside of intersecting points. For more detail see the TRIM command.

Fillet Radius

This field allows the user to specify a fillet radius as a default value. Enter the desired value in the input area. Please note thereby that during a working session the value last used will always be displayed as the parameter value for the next call to that command.

Chamfer Distances

These values establish default values for the chamfer distance from the point of intersection. Each value may be specified independently of the other.

Edit Polyline

Modify Polylines

 **Command: POLYEDIT**

The command may be entered from the pull-down menu Modify, or may be keyboarded by typing POLYEDIT.

Polylines may be modified in the following fashion:

Editing of a Polyline as a whole;
Filleting of a polyline;
Chamfering of a polyline; and
Editing individual polyline segments by editing their control points.

Polyline to B-Spline

The Spline command (Modify > Edit Polyline > Polyline to B-Spline) fits a B-spline curve to a polyline. The curve is not forced to pass exactly through the polyline's vertices (as does the Pcurve command, described below) but produces a smoother curve fit.

2-D Polyline to Curved Polyline

The PCurve command (Modify > Edit Polyline > 2D-Polyline to Curved Polyline) fits a curve to the chosen polyline. The curve is forced to pass exactly through the polyline vertexes. The degree of fit can be made smoother with the number of steps and tension options. The tension is similar to pulling on the ends of a string constrained to pass through the polyline vertices.

Vertex Editing

 **Command: VTXEDIT**

Editing of single segments of a polyline is done through the command **Vertex Editing**. This command allows the user to change the vertices, the start and end points, of the polyline. The vertices of a polyline segment can be inserted, deleted, and moved. Also, the width of the polyline segments can be changed.

Detail Menu

Text...

Command: TEXT

The TEXT functions are used for inserting text into drawings. They provide labeling, create legends, descriptive notes, and other text elements in a drawing. Most of these commands and options are found under the menu *Detail*.

Draw Text on Screen

This is a very simple text command, allowing you to enter text on screen so you can judge when to end lines of text. When in the command, any character you type will appear on screen as well as in the command prompt area. There is no character justification allowed. This command uses the current text style. To change this style you may use the FONT command and choose the style you wish to make current.

```
Specify start point or select option (Justify,
Style):

Insertion angle <E>:
```

Text: ***This is some sample text***

Text:

QTEXT command (Command Line Version)

Command: QTEXT

The QTEXT function is used to create text entities in the current drawing.

The command QTEXT works like the command TEXT but with one difference: QTEXT is performed at the command line.

Reference Text...

Command: RTEXT

With the Reference Text utility new text is entered in a dialog box with a matching style and attributes of one previously entered. Simply pick a text entity and overwrite the string in the dialog box. Then specify the location of the new text entity in the drawing. The new text will have the same font, height, width factor, rotation angle and layer as displayed in the dialog.

Modify Text...

 **Command: TEXTEDIT**

You can modify an existing text line in the drawing by selecting **Modify Text…** from the menu *Detail*.

Correct Text

Command: TCORRECT

The function **Correct Text...** allows the user to edit text entities and attribute definitions. It is possible to correct texts or text attributes as well as to change the font or the point size.

Call the function Correct Text by keyboarding TCORRECT or select Correct Text... from the pull-down menu *Detail*.

One Line Text Editor

This is a very simple text editor, allowing you to change any characters or add as many as you need to edit the line of text. You may only edit one line at a time.

HOT TIP – if no command is active, you may simply pick a piece of text on screen and this editor will automatically be activated. This is a new and quick way of editing text any time you are not running any other command. This command is not normally used to edit point numbers, descriptions or elevations.

Font... (Text Style Definition)

 **Command: FONT**

A font file defines the shape of the text characters that make up a character set. In MicroSurvey, you can use MicroSurvey's own compiled shape (FSH) fonts, AutoCAD compiled shape (SHX) fonts and True Type fonts.

Text Styles maintain the font and how it is configured - oblique angle, height, etc.

In the example below, the Text Style is named MYSTYLE and when used would create text 10 units high with an oblique angle of 15 degrees. The shape of the text will be based upon the Arial (True Type) font with a bold italic typeface.

The command FONT allows you to load fonts from the font files and customize them to suit the requirements of the drawing.

All installed True Type fonts provided with the Windows operating system are listed by font family name.

Compiled SHX and FSH fonts are listed by the name of the file in which the fonts are stored.

So you can assign a font to a text style by selecting either a True Type typeface name and its font style (Standard, Bold or Cursive for example) or SHX or FSH files.

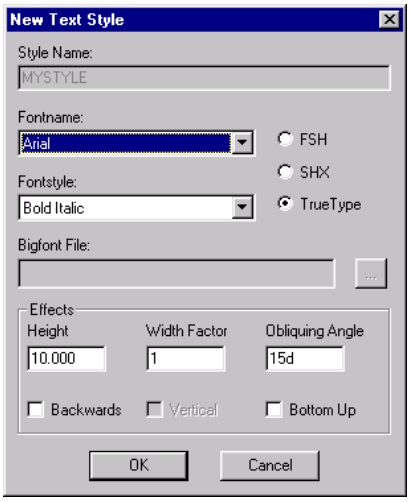
Selecting an existing font and making changes in it or adding a new font are done in the same way and are, therefore, discussed as one operation.

Text Styles can be created or modified from the Text Style dialog. Type the command FONT at the keyboard or select Fonts from the Text menu to open the *Text Styles* dialog.

New Text Style

To add a new Text Style, Select **New...**

The *New Text Style* dialog box will open.



Define the new Text Style by the options listed below and confirm with OK.

Modifying a Text Style Definition

An existing Text Style can be edited by highlighting the desired Text Style from the *Text Styles* dialog and then selecting **Modify…** The same *New Text Style* dialog will appear, allowing you to modify the following:

Options

- **Height** (point size) is expressed in drawing units. Entering a numeric value for Height gives the font a **fixed height** which can no longer be changed when inserting a text object using this font. If change in

the font height may be required, the user should keep the height parameter at a 0 value. The font height may then be defined in the dialog box *Text*, entered from the menu *Draw*, or entered by typing TEXT.

- **Width Factor** controls the width of the characters in relation to the height. This ratio is called the *Aspect Ratio*.
- **Oblique Angle** defines the inclination of the characters in relation to a horizontal (X axis) line. Input values between 0 and 180 result in an inclination to the right (forward), values above 180 to an inclination to the left (backward). Practically, values between 0 and 30 or between 180 and 210 are meaningful.
- **Backward**, when selected, will produce a mirror-image display of the text.
- **Vertical**, when selected, will product characters aligned vertically.
- **Bottom Up**, when activated, will produce characters displayed upside down.

It is possible to combine these settings, although in some cases the combination of properties may interfere with each other.

Note: Some type fonts do not support all of these options. For instance, some fonts can be aligned vertically. On others, there is no change in aspect ratio permitted.

Confirm the settings of the properties by selecting OK. Fonts added or changed in this manner can now be used for creating or editing text objects.

Note: Defined text styles are saved with the drawing file only with a reference to the corresponding font file. They are available any time that the drawing is opened if the corresponding font file is found in the programs directories.

Compiling a Source Shape File to a Binary Font File

Command: FCOMPILE

The program uses binary font files which allow to define text styles in the drawing (see command FONT).

Defined text styles can then be set and used with the commands TEXT, TEXTEDIT, ATTDEF, ATTDEDIT.

Binary font shape files (*.fsh*) provide less loading time and improved memory usage. A number of those FSH files are delivered with the program.

Application specific or customized font shapes can be defined in plain text in shape files of type **.shp** or **.fsh**.

Before they can be used in a drawing, it is required to compile these source files to binary font files.

To convert a source shape file to a binary font file

1. Call the command FCOMPILE
2. Specify the *.shp* or *.fnt* file name in the file dialog box,
3. In another file dialog box, enter the name of the binary font file of type *.fsh* to be created for later utilization in text commands.

Multi-line Text (MTEXT)

Command: MTEXT

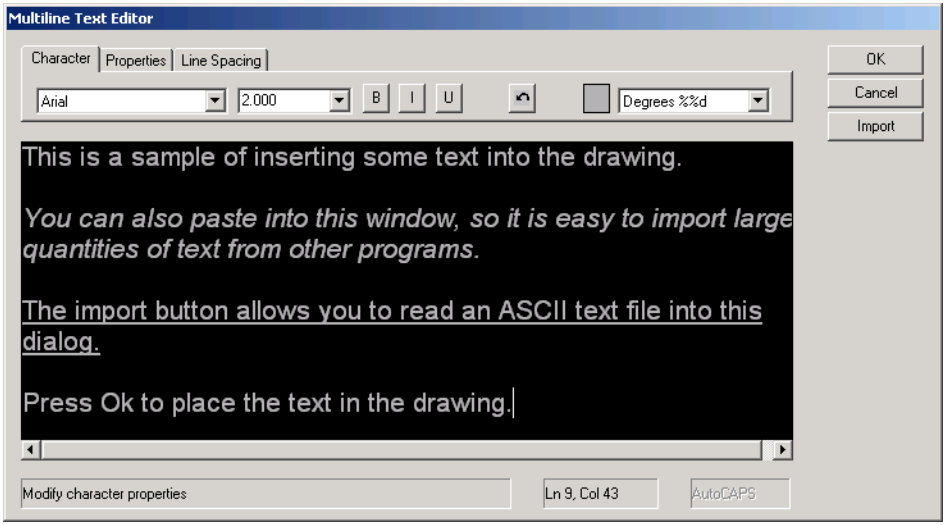
MicroSurvey now supports MTEXT. After issuing the command you are asked to specify the location of the new text by picking an upper left and lower right corner on the drawing. This window roughly locates the placement for the text but is essentially used to determine the line length. Text will be wrapped into the next line if the available width is insufficient. While specifying the two window points, you are presented with options to change Height, Justification, Line Spacing, Rotation, Style and Width.

You may choose to set these values here, however these options are also available in the dialog that follows.

Next, you are presented with a windowed dialog box in which you enter your text. Three tabs allow you to exercise control over all text parameters and a **Text File Import function** enables you to import an external text file into the window. These duplicate and add to the earlier

options that were presented and you can set your preferences in either manner.

Note: Text copied to the clipboard from any Windows application can be simply imported into MicroSurvey's MTEXT command dialog window, either through picking on the window and using Windows' "Ctrl V " function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry.



MTEXT Dialog showing Character Options.
Note: to allow the text to fill the indicated space in your drawing and to line wrap as needed within that designated width, be sure to continue the line entry beyond the visual extents of the entry window. Pressing the Enter button as we have done in the above example will allow you to see all the text without scrolling left-right, however it may not give the desired effect when this text is transferred to the drawing.

Most of these windows and buttons you are probably fairly familiar with if you do much word processing. Viewed left to right, these windows and buttons:

1. Specifies the required font
2. Toggles between bold and regular type
3. Italicizes text
4. Underlines Text

5. Removes the latest entry or reverses the latest action
6. Indicates the color to be applied to text. Click on this to change the setting

The last button allows you to call up special symbols (diameter, degree, plus/minus)

MTEXT Dialog showing Property Options.

Here, you can specify a named text Style, set Justification, override text Width originally set using the picked window points, and finally, Rotation, where you can specify an angle or bearing at which to place the entered text.

MTEXT Dialog showing Line Spacing options.

Here you can specify line spacing distance -either a minimum (At Least) or (Exactly)

Edit Multi-line Text

Command: MTEXTEDIT

This command predictably is used for editing multi-line text. When the command is issued, you are prompted to select the multi-line text that requires editing. This command will only work on text that has been entered with the MTEXT command. You must use TCORRECT or TEXTEDIT on other text entities.

When you select an MTEXT entity, the full multi-line entity is brought into the MTEXT dialog window, where you can edit any of the parameters that were originally available when the text was first created.

Cross-hatching

 **Command: HATCH**

Hatching (or cross-hatching) a drawing adds meaning to the drawing, and helps to differentiate the materials, the areas, or any of a number of other unique attributes.

Applying hatch marks or filling certain areas with distinctive patterns can greatly increase the clarity and legibility of a drawing. In a number of specialized applications there are norms and specifications which demand the use of specific hatch markings within a construction drawing.

Hatch... enables the user to use pre-defined hatch patterns to fill selected areas of the drawing. With the aid of a dialog box, the user is able to select the desired hatch pattern from a list and display this example in the preview window. The preview function simplifies the creation of new patterns and makes it easy to alter existing ones.

Hatch Patterns

A Hatch Pattern is comprised of a number of elements. This program already contains a number of typical hatch patterns used in various standard fields of application. The user may also choose to use other hatch patterns for specialized circumstances.

Hatch patterns are stored using the **.pat** file extension name.

Hatch Pattern Elements

Hatch Lines

A hatch pattern can contain a number of individual or diverse line types.

Hatching Angle

The lines of a hatch pattern can be arranged at any chosen angle. Standard hatch patterns are usually either made up of horizontal or vertical lines, but may also be drawn at a 45° angle, or at any other angle. A hatch angle of 0° defines a horizontal line. A positive angle value will cause the lines to slant in a counter-clockwise direction, while a negative value will cause a slant in a clockwise direction.

Line Spacing
The spacing between the lines of a hatch pattern is measured in drawing units. The default line spacing is set to one (1) drawing unit. Altering the line spacing is possible to adjust a pre-defined hatch pattern to suit the proportions of any given drawing element.

With the command QHATCH, experienced users can choose to bypass this option completely and use the keyboard to input the values directly.

Boundary Hatching

Command: BHATCH

The command BHATCH fills a region bounded by a closed curve just by pointing inside it.

The BHATCH command defines a boundary by creating a polyline from the entities that make up the boundary. Select each entity to define a boundary around the area you want to hatch.

Modify Hatching...

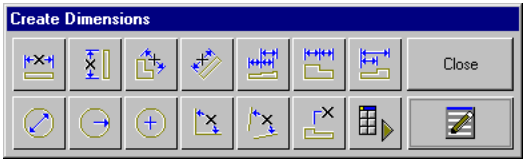
 **Command: HPEDIT**

You can change an exiting associative hatch by choosing this command or entering HPEDIT.

It allows you to add additional objects to the set of entities to be hatched, as well as change the hatch pattern.

CAD Dimensioning

Command: DIMDRAW
This Dimensioning command will activate the CAD Dimensioning Palette.

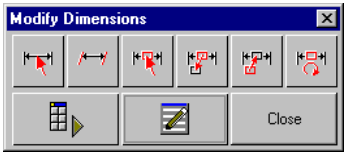


The buttons are pretty much self-explanatory.

Note: Dimensions added to your drawing using these buttons are considered to be CAD dimensions and will not be added to the MicroSurvey database.

Modify Dimensions

Command: DIMEDIT
This Modify Dimensions command will activate the Modify Dimensions Palette.



The buttons are pretty much self-explanatory.

Parts Menu

Part/ Symbol Library...

Command: PARTLIB

The command PARTLIB in the menu *Symbols* allows the user to manage drawing files such as project drawings, template drawings, Symbol files, or symbol files.

A library is defined by a parameter file (of file type **.plb**) and a set of drawing files assembled in a directory where the library parameter file is located.

The command PARTLIB called with the name of a parameter file (for example, C:\MSCAD2002\drawing.plb) will display a dialog which allows you to select a drawing to be inserted. The advantage of the command is that it displays all drawings with a preview bitmap, with verbal information in a list box.

Within the dialog box the user will see the preview of the drawings and information about the drawing.

There are two modes for the list box:

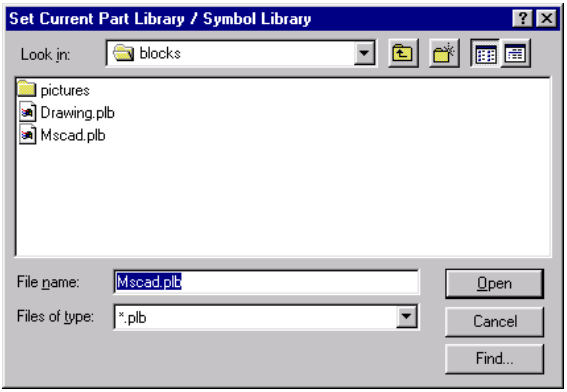
- In **Info Mode** the drawings are displayed together with the verbal information.
- In **Quick Mode** the drawings are displayed only with their preview bitmap.

The other buttons of the dialog have the following functions:

- **Close**: Closes a dialog without further action
- **File Info**: Opens a dialog to display and to change the file information.
- **Open**: This opens the selected file as separate drawing.
- **Insert**: This button lets the user insert the drawing into your current drawing.

Set Part / Symbol Libraries

Command: SETLIB



This command permits you to locate and select the Part/Symbol library that you wish to use as the default.

Symbol Library Setup...

Command: PLBSETUP

The command PLBSETUP (Symbol Library Setup) allows the user to setup up new Symbol libraries and/or modify existing Symbol libraries.

The command provides dialog boxes that:

- Select the drawing files to be included in a library;
- Set preferences for part or symbol insertions from the library; and
- Set the parameters which determine the layout and options of the Symbol library Dialog Box.

PLBSETUP creates files of type **.plb**. These files store the parameters for a single Symbol library. A Symbol library is called by using the command PARTLIB, and then specifying the name of a .PLB file. No knowledge of structure or syntax of the .PLB format is required to use the PLBSETUP command.

Specifying the Drawing Files Belonging to a Symbol Library

The dialog box *Part Library Setup* allows the user to specify the drawing files to be incorporated into the individual library. He or she can only choose drawing files from one folder (directory). It is not

necessary to include all of the drawings in a folder. This allows the user to maintain all of the drawings used for part or symbol insertions within one directory, and at the same time permits the creation of multiple thematically separated" libraries.

To add drawings to the current library highlight the file names in the left side file list of the dialog box and then click to the button > to insert them to the file list making up the library. The buttons (<, >) permit removal of drawing files from the list of Symbol library files. When only one specific file is selected from either of the list boxes, the preview image of the drawing will be displayed in the Preview Area. The buttons >> and. << allow the user to easily move file names from one list to the other.

Insertion Parameters

It is possible to pre-define insertion parameters for the files added to a library. Select one, more than one, or all entries from the list of the library files and then select **Modify**. Next, activate the option **Activate insertion parameters** and enter the default values of the *Scale Factor* and the *Rotation Angle* for insertion.

Part Library Dialog Box Layout and Options

Using the *Part Library Options* dialog box, the user can determine the layout and the available buttons and options of the dialog box which comes up when calling this individual library with the PARTLIB command:

The edit box **Title** allows the user to enter a name for the dialog box of the Symbol library.

Preferences for the Preview List

Preview List Style lets the user specify whether the part library Dialog Box should start up with a single column or multi column style of the list showing the drawing previews)

Sort by... helps the user determine whether the drawing preview list should be sorted by name (usually determined for part libraries) or by date (usually determined for project libraries).

Buttons assists the user in deciding options provided in the Dialog Box for a specific Part/Symbol Library.

Open allows the user to open a selected drawing for modification.

Insert allows the user to insert a selected drawing into the current drawing as Part/Symbol.

Explode allows the user to specify whether the Part/Symbol should be inserted as complex object (block), or exploded (broken down to single entities).

Info allows the user to open the Drawing Information sub-dialog of the specific library.

Set "Explode" mode as default sets the preference (activated/deactivated) for the mode of the button Explode.

Allow drawing info editing specifies whether the user may alter drawing file information stored with the drawing database.

Enable preview list style switch determines whether the library Dialog Box should contain the Quick Mode / Extended Mode button, which, in turn, allows the user to toggle between single column and multi column style in the file preview list.

Use default info tags determines whether default info tags (Project, Name, Drawing No., Draftsman, Notes) or user defined info tags are used when drawing information is displaced in the Drawing Information Dialog Box of the PARTLIB command.

Modify, in the preferences area, allows the user to determine individual tags for the file info fields used in the specific library (for example; manufacturer, price, etc. in a library managing purchased parts).

Insert Part

 **Commands: INSERT, QINSERT**

The command INSERT offers broad support for the user in its Dialog Box. The command QINSERT is the command line version of the command. The command macro QINSERT ? allows fast insertion of internal parts (parts already defined in the drawing).

The command INSERT provides selection areas for:

- Selection of external parts;
- Definition of the insertion point, scale factors, and rotation angle, with future choices of using coordinate/ numeric input or cursor selection;
- All parameter values may be reset; and
- The Part may be exploded into its components.

Selecting a Part/Symbol

To select the Part to be inserted, type the name of the Part (or select an existing Part) in the field Part Name. Use the buttons External and Internal to select the correct Part listing.

Defining the Insertion Point

Defining of the Insertion Point either by entering the proper coordinate values, or by selecting Interactive, and picking the point with the cursor.

Scale Factor

The Scale Factor input areas will offer the 1.0 as the default XYZ scale factor. This will insert a Symbol at its original size. Remember that a scale factor greater than 1.0 will increase the size of the Symbol, and a factor less than 1.0 will reduce the size of the Symbol to the percentage of the original size specified by the Factor.

The option *Interactive* allows the user to define the size of the Symbol during the insertion as well. To insert the Symbol using the Interactive option, simply define by opposite corners the rectangle into which the Symbol is to fit.

Since dragmode is active during this operation, the user will have the benefit of seeing the selection results before confirming the selection.

Rotation Angle

The Rotation Angle can be set from the Dialog Box. Enter a value for the Rotation Angle in the proper input window. For counter-clockwise rotation enter a positive value, for clockwise rotation enter a negative value between 0 and 360 degrees

Default

Default allows the user to reset the parameters for the Insertion Point, the Scaling Factor and the Rotation Angle back to the original parameter values.

Exploded

Exploded allows the user to explode the Symbol into its component entities. Exploding will cause the loss of the characteristics that were retained as a Symbol.

Paste Internal Part 1:1

Inserts a symbol into the drawing without scaling. The symbol must have already been inserted into the drawing in order to use this command, because it only works with Internal Symbols

Merge External Part 1:1

This command will insert a symbol into the drawing that is currently external. External means that the symbol has never been used in this drawing. No scaling is applied to the symbol when it is inserted.

Edit Part Attributes

 **Command: ATTXEDIT**

If only the Attribute values shall be changed, please use the command ATTXEDIT or call the function **Edit Symbol Attributes...** from the menu *Symbol*.

After the calling of the command you will be requested in the command line area to select the Symbol of which the Attributes shall be edited. For this it is enough to click the desired Symbol with the cursor at any spot. You can always select only one Symbol.

After the selection of the Symbol the program calls a dialog box.

In the window the Requests (left) and the current Values (right) of the Attributes included in the selected Symbol are listed. Select the Attribute value to be changed. The selected value is transferred to the editing field (below) and can be edited.

Modify Single Symbol Attributes (one by one)...

 **Command: ATTEDIT**

Enter the command ATTEDIT in the command line area or select **Modify Attributes...** from the *Symbol* menu.

The first prompt will request that the Symbol be selected. Make this selection either by naming the Symbol, or by selecting the Symbol with the cursor.

After the Symbol selection, a dialog box will appear. The editing process from this point is very similar to the procedure followed in the creation of an Attribute, except that the name is now displayed, and cannot be changed.

If a Symbol contains several Attributes, individual Attribute values will be displayed in the input field *Value*. Clicking the right double arrow will display the next Attribute, while clicking the left double arrow will display the previous Attribute is displayed. Only the Attribute whose name is currently displayed will be edited.

Modify Attribute Text

Command: ATTVALUE, ATTMOVE, ATTROT

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.

Export Attribute Information

 **Command: ATTEXP**

Writes attribute information to a file.

With this function, the user can extract attributes for Symbols or entities from the user drawing and save these in a CDF-format file. This is important when transferring and analyzing information from the drawing to or from programs for further processing

Define Parts…

 **Command: PARTDEF**

Select the function **Define Symbol...** from the menu *Symbol*, or enter the command PARTDEF via the keyboard.

Symbol Name

Names of Symbols can contain up to 31 characters. In addition to letters and numbers the special characters **$**, **-** , and **_** are allowed. The box is not case-sensitive.

Enter the name of the Symbol in the input field of the Dialog Box. Existing Symbols are listed in the selection window below. If one of

those Symbols is required, select the Symbol from the Selection Window using the cursor.

If there are Symbols of the same name already in the selection box, a message will appear **warning the user that current symbols will be redefined**. The message will give the user the option of choosing a new name, or of re-defining the existing Symbol.

The user may also, in this Dialog Box, elect to delete the Symbol in the original drawing. The appropriate box should be checked to exercise this function.

After this choice has been made, verify the selections of the Dialog Box by OK. the Box will close, and the Symbol will have been created.

Insertion point

Each Symbol must have a defined insertion point.

The **insertion point** is a reference point that enables the user to position the Symbol properly during insertion. The insertion point of a Symbol is also the basepoint for changing the scale, or for use as a rotating point for rotation during the insertion.

Any point on - or even off of - the Symbol may be selected as the insertion point. Typically, users will select the left lower edge, the center of an object, or some other discreet point of the geometry as the insertion point. It is a good idea to use the Object Snap options for this selection, as it insures precision.

Object selecting

Next, the user must select the object or objects that will make up this Symbol. To do this, the Option Bar furnishes several options. After selection, confirm by pressing.

Write Part File

 **Command: PARTEXP**

Write Symbol File... works identically to the function **Symbol Definition...** (Command PARTDEF) but is used to export Symbols.

Both functions serve to define entities of a drawing or to define the whole drawing as a Symbol. The Symbol names, the insertion point of the Symbol and/ or the drawing entities are defined or selected.

Define Attribute...

 **Command: ATTDEF**

Attributes are, in general, text objects which are related to, or joined to, Symbols. They are treated together with the Symbol as a whole object. The text objects may be constant, or variable. They may be visible, or hidden. There may be a single Attribute with a Symbol, or several.

Using Attributes, the Symbols can be further defined by textual information. This information may be used to label Symbols in the drawing, or may be used to contribute to a data base program for analysis later in the engineering process.

An Attribute consists mainly of three components; its name; its request; and the value. There are also a series of options which can be elected during the creation of an Attribute.

The **Name** is the designation of the Attribute under which it is integrated in a Symbol. Especially if a Symbol contains more then one Attribute, the name is the most important criteria for the effective usage of the Attribute.

The **Request** is a component of the Attribute which can be displayed during the insertion of the Symbol in the command line area. Normally, the request contains the invitation to enter a variable Attribute value.

The **Value** is that portion of the Attribute which is written, together with the Symbol, into the drawing. It can be constant or variable, visible or transparent.

The following parameters for an Attribute may be defined during the creation of the Attribute, and edited later in the design process.

Edit Attribute Definition...

Command: ATTDEDIT

You can modify an existing attribute definition by using the ATTDEDIT command. Select an attribute definition in the drawing. Immediately a dialog box is displayed which is similar to the dialog to create attribute definitions.

If the check box labeled Interactive is marked, you will be prompted to relocate the attribute definition when exiting the dialog. Otherwise you can change the insertion point coordinates in the dialog box, if desired.

Grouping Entities

Command: GROUP

Drawing entities can be combined in Groups. Grouping entities allows the user to manipulate all entities within the Group in a single operation (Moving, Rotating, Mirroring, or Scaling, for example). Single entities of a Group may still be *edited* individually (for example, intersecting or stretching). Entities may be removed or added to a Group at any time. An object may be a member of more than *one* Group. *Groups* may be contained in *other* Groups. Group definitions are stored in a distinct table of the drawing database. Groups are maintained in this fashion from one drawing session to the other.

Differences Between 'Symbols' and 'Groups'

A Part (a block) has its own insertion point. A Group has none. A Part can occur multiple times within the drawing. If a block is redefined, all of its occurrences are updated. Groups are not treated like that. Groups can, however, be copied. A Group that originates from another Group that has been copied is a separate union of entities. Each Group has a unique name.

Note: If an entity which is member of a Group is included in a part definition, the entity is deleted from the drawing and removed from the Group**.**

The Command GROUP

To group entities, or to manage and manipulate Groups, use the command GROUP. The options within the command are:

Create	Brings entities into a newly named Group
Explode	Abolishes a Group definition
Rename	Alters the name of an existing Group, especially of anonymous groups
Add	Adds drawing objects to an existing Group
Remove	Removes drawing objects from a Group
Selectable	Determines whether a Group is selectable or not

Creating Groups

To create a group, first determine a unique name. The name will be required to identify the group later for a number of command options. In addition, specify whether the group should be selectable or not (see below).

To define a new group,

1. Call the command GROUP with option Create,
2. Specify a unique group name (max. 31 characters),
3. if desired, enter a description (max. 64 characters),
4. Determine whether the group should be selectable or not (the default is Yes), and
5. Select the drawing objects, which should belong to the named group.

To create a group using Command Line:

> GROUP
Select option: **Create**
Group name: **G1**
Description: **Screw**
Should the group be selectable (Yes/No)? **Yes**
Select objects: ***Select the entities that should belong to the group***

Exploding Groups

Group definitions may he abolished by using option **Explode** of the GROUP command. When a group definition is dissolved the corresponding entry in the group table is deleted in the drawing database.

The only way to abolish a group definition is by abolishing its name.

To explode a group,

1. Choose the command GROUP with its option *Explode*,
2. At the prompt "Group name: " enter the name of the group, to be exploded.

Because a single entity may belong to multiple groups and groups may be part of other groups, the **Explode** option does not allow the user to pick an entity within a group to explode the group.

Renaming Groups

A group may be renamed using the GROUP command **Rename**. It is useful to employ this method when anonymous groups created by copy commands exist. The system will automatically apply anonymous names to group copies (*A1, *A2, etc.) that exist through the use of one of the commands like COPY, MIRROR, or ARRAY.

To rename a group,

1. Call the command GROUP with its option Rename;
2. At the prompt "Current group name: " enter the name of the group (for example "*A0");
3. At the prompt "New group name: " enter the name of the specified group (for example "Screw_Copy1")

Add / Remove Entities

Two other options of the command GROUP are provided to allow the user to integrate additional entities into an existing group or to remove entities that are currently in the group.

After a group name is specified, the entities that belong to the group are highlighted.

- To add entities to an existing group, simply 'pick' the entities that are to be added (they will *not* be highlighted).
- To remove entities from an existing group, 'pick' the highlighted entities that are to be removed

Note: Should *all* entities be deleted from a group, remember that *the group will still exist as long as it has a name!*

Working with Groups

Editing commands will treat all entities in a group as a single object. Commands like MOVE, ROTATE, or COPY will act on the entire group.

If the user should select an object for one of these commands at the prompt, that selection will lead to the automatic selection of all group entities.

A typical prompt of a command like this would be:

```
Select objects:
```

The rule is: In commands that allow **multiple selection**, identifying a *single* entity of a group is a selection of *all* entities belonging to the group(s). When the user is *prompted* to select multiple objects, he or

she may choose the object selection option *Group* from the options bar, and enter a group name to select all of the entities belonging to the group. Groups, therefore, can also be named selection sets.

Other MODIFY commands (Offset, Lengthen, Intersect, Trim, etc.) request a single entity in one or more steps (usually to modify the geometry of drawing entities). A typical prompt for a command like this would be:

```
Select entity (Line, Circle, Arc, 2D-Polyline):
```

In this case **single selection** is being used. Any time the user is prompted to select a single entity, the group does not matter. These commands modify the geometry of single selected entities even though those entities belong to a group. The modified entities will remain members of the group. This feature makes allows great flexibility in working with GROUPS and provides significant advantages for the user compared to using BLOCKS.

The following table summarizes the editing commands and describes how entities that are members of a group will be processed when using each command.

DELETE

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. When a single entity of a group is deleted, the entity is removed from the group definition.

MOVE, ROTATE, SCALE, FLIP

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used.

MIRROR, COPY, ARRAY

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. The system will apply anonymous names to copied groups ("*A1", "*A2", etc.) that are created by the use of these commands.

STRETCH

Those entities belonging to a group that have been selected using the object selection modes *Crossing* or *Cpolygon* will be stretched.

TRIM, EXPAND

The prompt to Select cutting edges or ...boundary edges will include all entities of a group, unless the Object Selection Mode *Single* has been used.

OFFSET, FILLET, CHAMFER, LENGTHEN, INTERSECT

These commands always prompt the user to select a *single* entity to be modified. That an entity belongs to a group does not matter in these operations. Single entities altered by these commands will remain in the group. If *new* entities *originate from these commands* they will *not* be automatically included in the group.

DELPARTIAL, REJOIN

When an entity (line, circle, arc or 2D-Polyline) is broken into two or more entities by partial deletion, only one of the entities will remain in the group. When an entity that has been broken into two or more pieces is rejoined, the newly created entity is no longer member of any group, even if one or more of the pieces was part of a group.

PARTDEF, HATCH, HATCHEDIT

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. This also applies to the command HATCHEDIT when the user is prompted to `select additional objects:` to be included to the set of entities for associative hatch editing.

TEXTEDIT

The text entity retains its association to its group if the command TEXTEDIT is used to modified text.

Externally Referenced Drawings

Externally Referenced Drawings serve especially to

- Create assembly drawings from different files
- Insert detail drawings into a finished one
- Temporarily reference another drawing in the current drawing

Often, the biggest benefit of using externally referenced drawings is given within a running project with a workgroup creating several drawings and details in a network.

Another advantage is that within the current drawing only a link to a referenced drawing is stored but not the entire entity and symbol database information of the referenced file.

In several situations, the usage of externally referenced drawings is more convenient and applicable then inserting external Symbols into the current drawing as blocks.

XLINK command: Control externally referenced drawings

Command: XLINK

The XLINK command allows you to create a link to an external drawing and to control externally referenced drawings (also called Xrefs) in the current drawing.

Differences to Symbol Insertions

Although the treatment of linked drawings is basically similar to those of Symbol insertions (block insertions), the some fundamental differences are:

An Xref establishes a link to another drawing file, but it does not become a permanent Symbol of the current drawing.

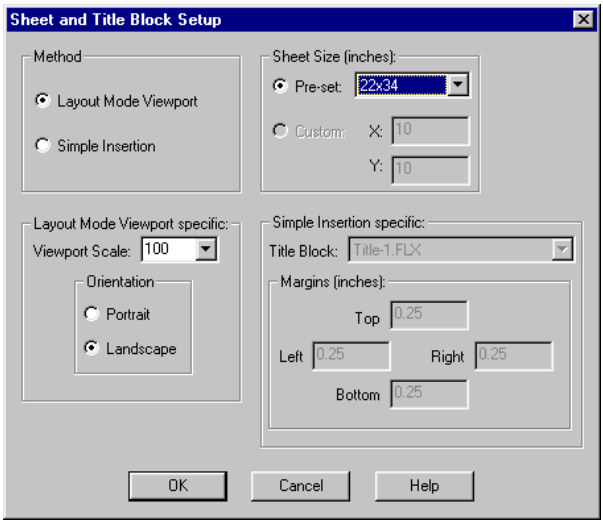
See the on-line help for a complete description of this command and how to use Xrefs.

Options Menu

Sheet and Title Block - Layout Mode

There are 2 options built into the Sheet and Title Block command, available from the options menu. This option is for the use of a viewport for plotting purposes. MicroSurvey uses the term layout mode which is the equivalent of AutoCAD's paper space.

First select the Layout Mode method from the upper left of the dialog. Then choose a Paper Size you wish to plot to, from the list on the upper right of the dialog box. Alternatively, you can specify a custom size. Then you pick the orientation by selecting either Portrait or Landscape. Landscape is oriented with the longer side horizontal and Portrait is oriented with the longer side vertical.



The last Item you pick is the drawing scale from the middle left of the dialog box. You may select one of the more common ones from the list or you may type in a different scale manually.

Pick insertion point:

At this point you have been flipped to another screen and the Title Block has been inserted onto a blank screen. A viewport has been

created to fit within the title block and your drawing is centered in this viewport based upon the drawing extents.

The drawing inside the title block, has been scaled up or down to match the drawing scale you set in the dialog box above.

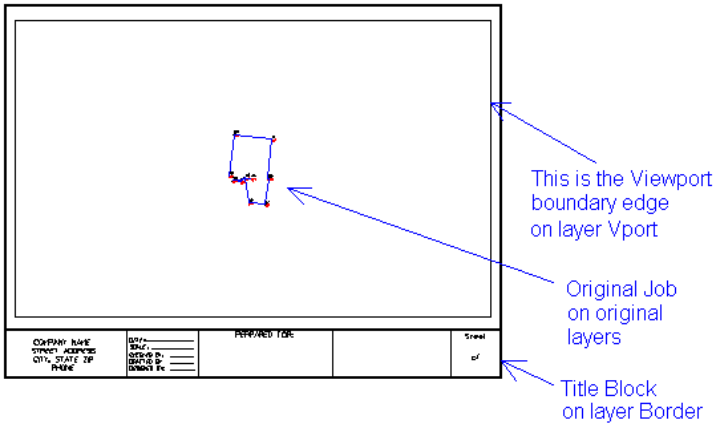
When you go to plot this job out to paper, Layout Mode assumes that you will now plot the job at a 1 to 1 scale. It has automatically adjusted the drawing to fit in the title block at the correct drawing scale, as you set it, and then has set the Layout Mode to require a 1 to 1 scale for plotting to achieve your desired final plotted result.

You are now in what is commonly called Layout Mode or Paper Space.

To switch back and forth between your original drawing mode and the new layout mode, you can type the command TLMODE, or pick the "Layout Mode/Model Space option from the File pull-down menu.

You can only edit the original job while in model space. You can only edit the title block while in paper space.

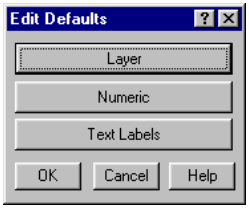
(For more details on Paper Space and Model Space, see that section in the manual or help file)



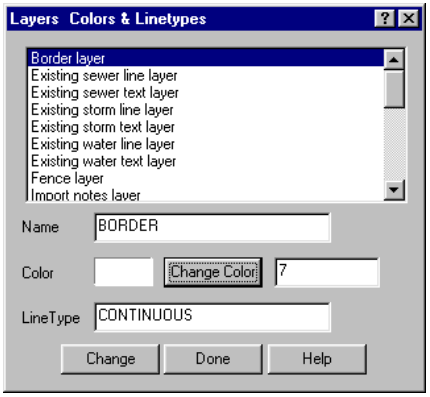
The Viewport is automatically placed on a new layer VPORT.
The Vport layer can not be changed, but you can freeze that layer, later for plotting purposes.

The Title Block is inserted as a part, on a new layer called BORDER, by default.

You can change this, prior to inserting the title block, by going under the Design pull-down menu and picking on Design Defaults – then pick on the button marked Layer.



Then scroll up or down to find the Border and make any changes you desire.



When you have made the changes desired, pick the Change button to record the changes, hit the Done button to exit without changes. This change will be made permanently for all jobs that follow, not just the current job.

Sheet and Title Block - Simple Insertion

The Simple Insertion method follows similar steps but excludes the use of a viewport and places the sheet and title block in model space. Complete instructions can be found in the online help system.

Settings

Command: SETTINGS

The command SETTINGS, which can be selected from the **Options** pull-down menu or by keyboarding **Settings**, is the main command to set the parameters and modes of the drawing environment.

Set Layer

Command: SETLAYER

This function allows the user to switch the active layer. It is not possible to create a new layer or to change any characteristics of the existing layer through the selection of this button. These changes must be made in the fashion described above in Layer Control.

Layer Configurations

Command: LAYCONF

This command allows you to save many different layer-setting files. Each one can have layers set to different values such as Frozen, Off and Locked or Thaw, On and Unlocked, or any combination.

Layer by Reference

Command: LAYREF

Change the current layer by selecting an item in the drawing. The current layer will be set to the layer of the selected item.

Load Linetype(s)...

Command: LINETYPE

LINETYPE allows the user to choose the linetype that will be displayed on the segment of the drawing that is currently active, and only on that segment.

Linetypes that were loaded with the drawing model are available in the present drawing.

Again, the linetype chosen with this option will not become the active linetype for all further drawings. It will only be added to the list of linetypes that are available.

In the dialog *Open Linetype File* the file filter has already been programmed to be **.lin**.

The dialog box *Linetype* will open. Choose the linetype(s) and verify the choice with OK.

Linetypes that were loaded with a drawing will be saved along with the drawing so as to be available at a later time.

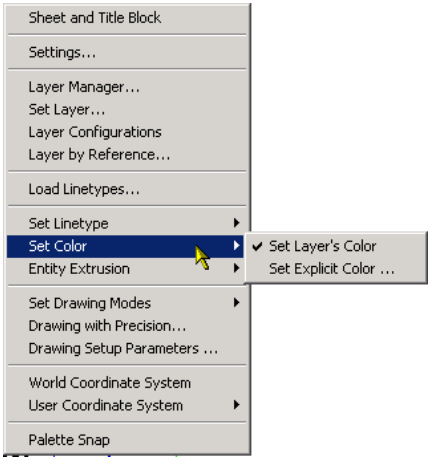
Set Linetype Bylayer/Explicit?

If you choose Bylayer, the linetype of lines added will be as specified in the Layer Manager for the current layer.

If you choose Explicit, the linetype will remain as currently specified, regardless of the linetype specified in the Layer Manager for the current layer.

See also SETLINETYPE and CELTYPE

Set Color Bylayer/Explicit



If you choose Bylayer, the color of drawing entities added will be as specified in the Layer Manager for the current layer.

If you choose Explicit, the color will remain as currently specified, regardless of the color specified in the Layer Manager for the current layer.

See Also SETCOLOR and CECOLOR

Entity Extrusion (THICKNESS)

Sets the value for the default thickness (extrusion). Specifying the value 0.00 means that no thickness is applied.

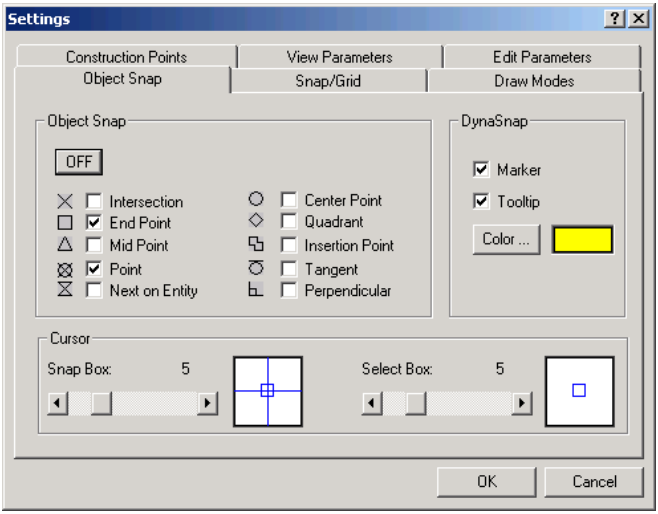
Properties

Type	Real
Range	Any positive or negative number
Default	0.00
Status	Read / Write
Saved	in the drawing file

Precision Aids Parameters

Command: PRECPAR

The precision aids can be controlled by the common dialog box, which is activated by selecting Drawing with Precision from the Settings menu or typing the command PRECPAR. The dialog box allows you to switch on and off the individual drawing precision functions.



Grid Display

Activate or de-activate the grid by clicking the check window on the upper left in the box **Grid Spacing**. A check mark in this check box indicates the grid display is active.

In the input area **Factor** enter the value that defines the relation of the grid points to the snap grid. Alternatively, the user can specify a predefined value by clicking available buttons.

Snap Grid

Activate or de-activate grid by clicking the check box on the upper left side in the area **Snap Grid**. A check mark in the check box signals that the snap grid is switched on. From the dialog box, select the input fields Width and Height, and enter values for the horizontal and vertical distances between the points of the snap grid. Units of measurement for these values are, again, drawing units. The value 1.00 represents the distance of one drawing unit.

Using the palette button (=) for the value Width enables the user to choose a vertical value equal to the horizontal value.

The input fields marked Origin in this box enable the user to define the origin of the snap grid on the X and the Y-axis. The desired coordinates for the origin should always be entered.

Orthogonal Mode

Activate or de-activate **Orthogonal Mode** by clicking the check box.

Object Snap

Using the dialog box Precision Aids, individual modes of the object snap function can be permanently activated (running object snap mode).

For this purpose use the control window for the desired mode. A check mark in the control window indicates that the mode is active. Clicking the switch OFF will de-activate all of the object snap modes.

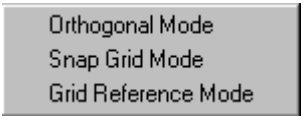
If several object snap modes are to be permanently activated, the user can differentiate between the modes by the selected point of a object. For example, if the modes Endpoint and Intersection are activated, then if the point selected on the entity is nearest an end point, that will be selected. If it the selected point is nearest the intersecting point of two or more entities, then Intersection will be applied.

Cursor

Allows you to specify the Select Box and Snap Box size.

Note: These precision parameters can also be accessed through the Settings dialog by clicking on the following button at the top of the vertical control panel.

Note: Orthogonal Mode, Snap Grid and Grid Display can be activated directly from the Options menu -> Set Drawing Modes submenu:



or by these buttons at the left end of the Properties Bar.

Drawing Setup Parameters

Command: UNITSETUP

The program allows you to set the number of decimal places of accuracy used in dimensioning and the type of angular units used by running the UNITSETUP command. These settings are saved with each drawing, so can be set differently in each drawing if desired.

The **Linear Units** can be set from zero to eight decimal places using:

- Decimal Units

- Fractional Units
- Architectural Units (Feet and Inches)
- Engineering Units (Feet and Inches)
- Scientific Units (Scientific exponential notation)

The **Angular Units** can be set from zero to eight decimal places using:

- Decimal Degrees
- Degrees / Minutes / Seconds
- Grads
- Radians
- Surveyors Quadrant Angles (N, S, E, W with deflection from North and South)

Orientation

In addition, the system allows you to modify which direction causes the angles to increase. The default setting is for Counter Clockwise increasing angles (CCW). You can set it to increase in a clockwise direction by selecting the Compass Angles (CW) radio button.

Angle Zero Direction

The system allows you to specify a different direction for Zero degrees. This is of interest to different special disciplines. For example Surveyors normally set zero degrees as up on the page. This allows for the correct display of angle and distance when drawing lines on the screen. The display on the status bar is updated depending upon the settings of the angle zero direction and the orientation.

Important note: MicroSurvey sets the orientation of the CAD system so that 0 degrees is North, and all angles are considered clockwise. Do not change this setting as this will interfere with the correct operation of many commands.

View Menu

View Manager... (Saving / Retrieving Views)

 **Command: VIEW**

It is convenient when working with different drawing portions to save and later to retrieve changes in angles of view, different drawing portions, and scales of view. This may be accomplished by using the VIEW command.

Pan

 **Command: 'PAN**

The command PAN moves the visible portion of the drawing, making it possible to look at objects that were previously not visible because they were off the viewing surface. In contrast to the ZOOM command, the apparent drawing scale does not change. The concept of PAN is to move the sheet of paper (your drawing) to a different position on the table (your screen).

Dynamic Pan

 **Command: 'DPAN**

As described above, panning means to move the display of the drawing in its current window without changing the magnification.

With the command **DPAN** you can move the current display in smooth operation until the resulting view fits to the needs to continue drawing or editing.

Dynamic Zoom

 **Command: 'DZOOM**

With the command **DZOOM** you can zoom in and out easily using one operation in the current drawing window.

To zoom in this manner, choose *Dynamic Zoom* from the menu *View*. The cursor changes to a magnifying glass with a plus sign and a minus sign. To start the dynamic zoom process, click the left mouse button and drag the mouse either up or down. Whether you zoom in or out depends on the movement of the cursor. To zoom in, drag up (in the direction of the plus sign). To zoom out, drag down (in the direction of the minus sign).

Zoom Window

 **Command: ZOOMWIN**

The ZOOMWIN command allows a chosen portion of a drawing to be shown in the largest possible scale. A box specifies the drawing portion. Identify opposite corner points with the cursor or enter the coordinates manually.

Zoom Back

 **Command: ZOOMBACK**

The command ZOOMBACK allows you to undo the last ZOOM command and recreate the prior drawing portion. The deleted scale is saved as the last used scale and may now be called again.

The program supports 20 previous zooms, so it is possible to zoom back many times.

Zoom to Fit (Zoom Extents)

 **Command: ZOOMFIT**

The command ZOOMFIT allows the user to show the drawing with all its elements as large as possible on the viewing area.

The user should remember that points of deleted or modified elements that may still be saved in the drawing can influence the results. These points will be completely deleted when a drawing is regenerated. Thus, before zooming to the largest possible drawing size, the user should consider doing a REGEN.

Zoom Factor

Command: ZOOMFAC

The command ZOOMFAC allows the user to zoom with specified scale factor.

Set Plan View to Construction Plane

 **Command: PLANVIEW**

The PLANVIEW command sets the view planar to the current User Coordinate System (UCS).

3D View Control...

 **Command: 3DVIEW**

It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the menu *View* provides the facility to make that definition.

The program uses as its standard viewing angle the top view (also called Plan View). At this angle the xy-plane is viewed from the direction of the positive z-axis, and the drawing or construction plane is parallel to the screen.

Additionally, FRONT, BACK, LEFT, RIGHT, and BOTTOM are possible standard views.

Animated Rendering

 **Command: ARENDER**

Rendering allows you to create an image of your 3D drawing that illustrates very realistically the model or plan.

ARENDER provides automatic rendering in a separate window of the program.

It runs without the need of setting lights.

The command allows animated rendering with fast zoom and rotation of rendered object.

Use the left mouse button to rotate on the x/y plane and the right mouse button to zoom on the z plane.

Use the Auto-Rotate option from the system menu to let the model rotate automatically. Use the left mouse button (drag with button hold down and then release) to determine rotation speed.

View Parameters...

 **Command: VIEWPAR**

Various parameters may be selected to define the direction and general size and shape of a given view.

There are values for ZOOM and PAN of the view on the screen, and selections to be made to determine visibility of the grid and the coordinate system.

Dialog Control Elements

Zoom / Pan

In the entry field *Zoom In Factor* and *Zoom Out Factor* the user may choose the factor by which the drawing on the screen is to be enlarged or reduced. These factors affect only the visible portion of the drawing.

In the entry field *Pan Factor* the user may choose the factor by which the visible portion of the drawing is moved using the command PAN. A pan factor of 0.5, for example, moves the drawing portion to about half of the visible drawing area.

Determining the Grid or Coordinate Symbol

It is possible to activate or deactivate the display of the Grid and/ or the coordinate system through the option *gridvisibility* and the symbol of a coordinate system.

To display the grid and the symbol of the coordinate system just click with your mouse on one of the two control fields. An x in the control field will indicate the active status of this option.

The determination of the grid, especially the distance between the gridpoint, takes place in the dialog box *Precision Aids*, opened by selecting its field or by entering the command *Precpar* by the keyboard.

In addition to the choice of whether or not to display the coordinate system the user may choose the colors of the coordinate axes and the symbols of the coordinate system. These choices are made available from the window called Configuration. Open this window by clicking on its symbol on the symbol panel or by entering the command CONFIG.

Before closing the window, the user must remember to verify the entries with OK or end the task or operation with the Cancel option.

Zoom In/Out/Pan Direct

Zoom In - Nearer to Objects

 **Command: ZOOMIN**

The ZOOMIN function enlarges the viewing window with each successive use by a factor determined by a given value.

Zoom Out - Away from Objects

 **Command: ZOOMOUT**

The ZOOMOUT function is the opposite of the zoom-in function. It reduces the viewing window with each successive use by a factor determined by a given value.

Pan: Portion Up

 **Command: PANUP**

After selecting this command, the visible drawing portion will immediately be moved up.

The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen.

Pan: Portion Down

 **Command: PANDOWN**

After selecting this command, the visible drawing portion will immediately be moved down.

The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen.

Pan: Portion to the left

 **Command: PANLEFT**

After selecting this command, the visible drawing portion will immediately be moved to the left.

The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen.

Pan: Portion to the Right

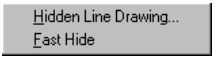
Command: PANRIGHT

After selecting this command, the visible drawing portion will immediately be moved to the right.

The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen.

Hidden Lines Submenu

Hidden Lines



Hidden Line Drawing...

Command: HIDE

The HIDE command allows you to create a separate drawing which contains a representation of the current view of the drawing with hidden lines removed.

Fast Hide

Command: FHIDE

In contrast to the HIDE command which creates a new drawing, the command FHIDE displays a hidden line view in the current drawing.

Render

Command: RENDER

Rendering allows you to create an image of your 3D drawing that illustrates very realistically the model or plan. To increase the realism the rendering may perform smooth shading on the objects that meet at a specific angle and evaluate the lights set in the drawing.

Render Settings

 **Command: RCONFIG**

The command RCONFIG displays a dialog box to view and set general preferences for subsequent rendering.

Render Lights: Inserting Light Sources

 **Command: LIGHT**

Ambient light is given any time, but linear, point and spot light sources must explicitly placed in the 3D model.

The command LIGHT is provided to insert light sources into the drawing.

Render Lights: Editing Light Sources

 **Command: LIGHTEDIT**

You may modify the settings of an inserted light source later with the command LIGHTEDIT.

This command requests you to select a light, represented by a block insertion.

MicroSurvey Menu

MicroSurvey Master Toolbox

We are trying to make your MicroSurvey Experience a little more pleasurable by adding many toolboxes (palettes), to help speed your drawing process up. Below is the Main Toolbox, which will call up all the other main MicroSurvey Toolboxes currently available.

MicroSurvey Defaults

Defaults Submenu

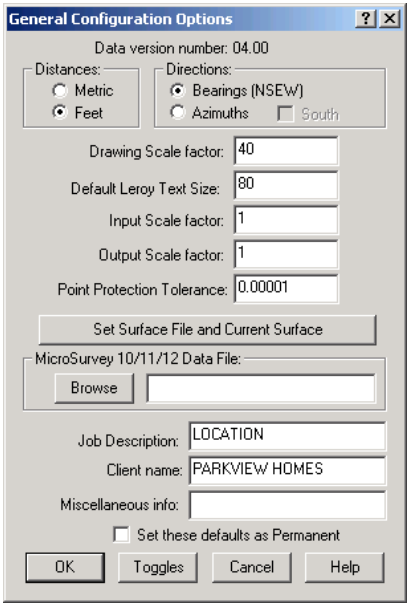
Many of the variables in MicroSurvey have default values you can set to make the construction of your drawing easier and more logical. These defaults are stored in the project directory (*.MSJ) of each job, in a file called INCAD.CFG. We chose this name so the product would be totally compatible with our AutoCAD based product called InCAD.

The System Defaults selection lets you edit the default values for the active job, and any changes you make will remain active when you reload the job in the future. You can cancel your changes by pressing **Esc** while in any of the configuration screens. The following selections are available:

Main Job Defaults

Each time you start a new job or re-enter an existing job, this dialog box will be displayed to allow you to either change or take note of the values.

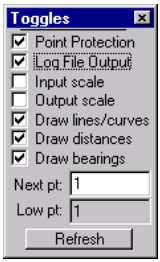
If this is a new job then the default settings will be those of the last saved settings (save option is at the bottom of the dialog box). If this is an existing job that has been worked on previously, then the settings shown are those that were saved with this job.



The most important setting to check or set are: Metric or Feet, Quadrants or Azimuths, and the Drawing Scale Factor. Please make an extra effort to ensure they are correct to help save tedious edits later if they need to be corrected.

Note: If you are viewing this in the manual, there is much more assistance in the on-line help system.

Hot Toggles Floating Dialog



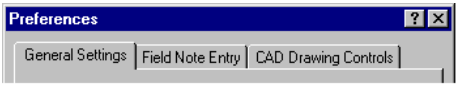
New for MicroSurvey is the Hot Toggles floating dialog to help you keep track of current point numbers and the most used System Toggles. It can be left on screen at all times much like a Tool Palette and allows you to make changes on-the-fly without exiting a command

to change a setting. This dialog is modeless which means that all other commands are available regardless of whether or not this dialog is active.

System Toggles Overview

The **System Toggles** are a set of **ON/OFF** variables that control how data is entered, the format of the prompts you will see, and the type of information drawn on the MicroSurvey screen. The most commonly used toggles can be accessed via the Hot Toggles floating dialog. Check it out.

There are 3 sections to this dialog box. Pick on the tab for the section of the dialog you wish to see.



IMPORTANT INFORMATION
MicroSurvey updates items in the drawing according to an established order of precedence, and points have the highest precedence. If you move a MicroSurvey point, all lines, symbols, bearings and distances, which depend on it will update automatically. You should leave the Draw Points toggle ON, because there are other programs that will analyze your drawing and update coordinates automatically if the program can find the MicroSurvey points that were inserted when the coordinate was originally calculated and drawn.

Toggle Shortcuts

The System Toggles can be changed in five different ways:

1) By selecting the **System Toggles** option on the Defaults Menu. This method allows you to change many different toggles quickly, and also is the only method that shows you the current setting of the toggle.

2) **Inside a command.** Many MicroSurvey prompts will present you with a quick toggle menu if you press **T Enter** or if you pick the Toggles button from immediately below the command entry window.

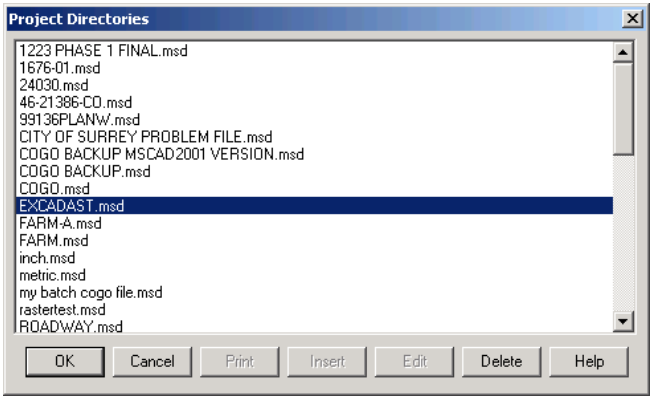
3) Enter the command Toggles from the keyboard

4) By picking the Toggles button from the Options Bar, across the top of the drawing, third from the left. You can also toggle individual options found within the toggles dialog box by selecting them here.

5) By accessing them directly via the Hot Toggles floating dialog.

Delete Projects

This option deletes projects of your choice from the hard drive. The deleted project file is **NOT** placed in the recycle bin. You will be shown all the jobs in the current project directory via this dialog box.

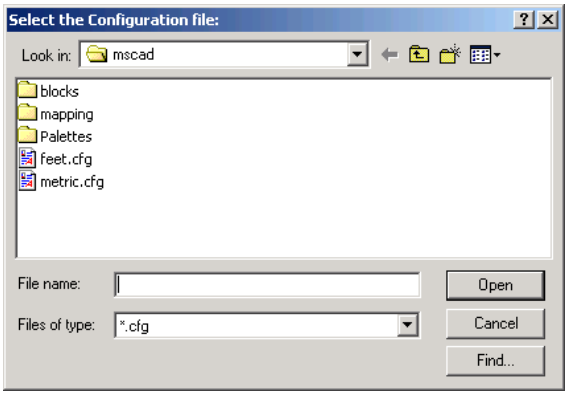


Read / Write Defaults

Read MicroSurvey Configuration File

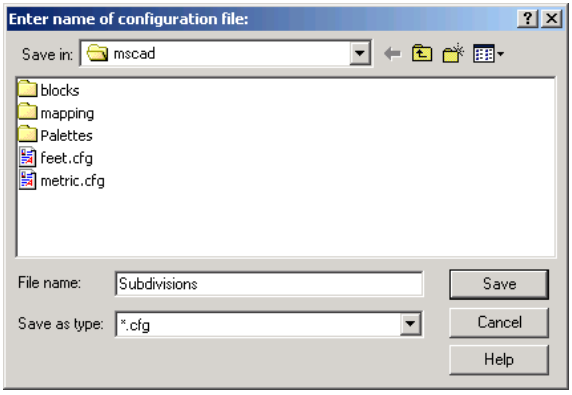
Allows you to read in a previously saved set of defaults. This will over ride any current settings.

There are two different configuration files supplied with the program. They are METRIC.CFG and INCH.CFG, and can be found in the MSCAD2002\MSCAD directory.

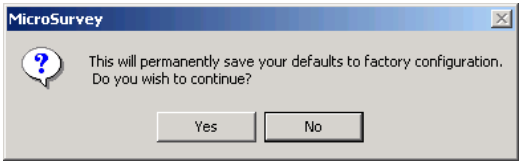


Save Configuration File

Allows you to save your current configuration of Defaults out to a file so that you can use it again later without taking all the time to redo the whole procedure. You may have many different configurations saved for many types of jobs or customers.



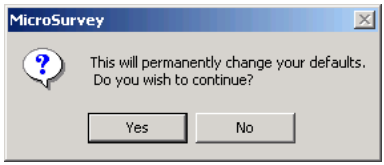
Reset to Factory Defaults



This option is a powerful one, allowing you to reset all of the defaults back to the way we ship the program. This means that you can really experiment with the defaults and always have a way to go back to the

Factory settings without having to remember or write them down ahead of time. This can also be dangerous in the sense that you may have taken a lot of time to set the defaults just the way you like them and if you pick this option you could erase this work. Of course if you have completed a job with these defaults you could always reload that job and then save the default settings at that time.

Save as Default Configuration



This option allows you to save the defaults specified in this job as the **global** defaults for **all** jobs. To save your defaults in connection with the current project only, simply exit the **Defaults** menu and they will be stored in file INCAD.CFG in the current project directory when you save and exit the job. To make the defaults global, select **Save as Default Configuration** and this file will also be copied into the MicroSurvey directory. Whenever you start a new project, MicroSurvey will look for the file in the MicroSurvey directory; if the file is there, it will be copied whenever a new project is started. Old jobs are not affected by changes made and saved after they were started.

This means that when you work on a project, MicroSurvey will set the defaults in this order of precedence:

1. Local values saved in your current project directory.
2. Global values stored in the MicroSurvey directory.
3. The nominal values built into MicroSurvey.

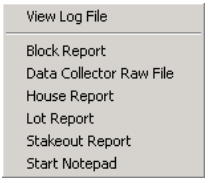
You can over ride all of this by reading in a previously saved configuration file.

Reports (Print/Edit/View)

Print / Edit / View Reports

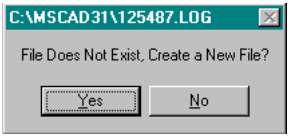
You can now print any of the standard reports that are calculated in MicroSurvey. Prior to printing you can edit the reports to make them look the way you desire. You can change the font, underline key information, make some text BOLD, change to italic lettering, change the margins, and a whole lot more. This routine actually calls up a small word processor and has many of the commands on its pull down menus that other popular word processors have. You now have control over what you print without having to cut and paste to Notepad. You can save the log file into a separate file in several different formats.

The width of the page is normally fine for most systems but may have to be changed depending upon the font size you use and also the type of printer you have setup in Windows. Some drivers, such as Fax printer drivers, can really make the output look weird. Please make sure that you have the correct printer driver loaded for your system so as to keep the output looking correct and printing correctly.

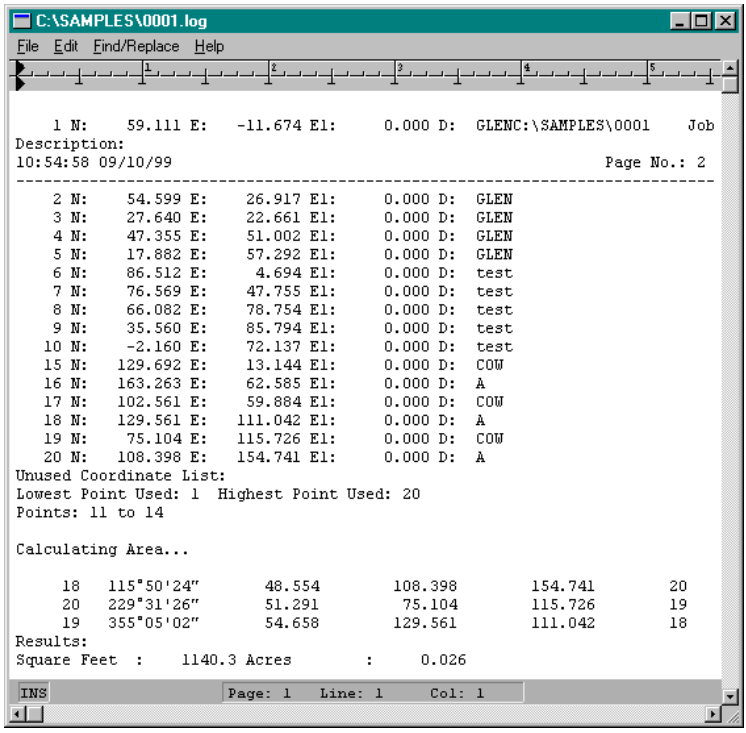


View Log File

The LOG file is only generated if the Toggle is turned on. You can clear the Log file by pressing the Clear File button on the Toggles box. Make sure that the Log file has been created before you attempt to do any work as the file may not be turned on. If it has not been created and you choose one of the options to view the Log file, you may see the following question.



Here is a sample of what the log file looks like:

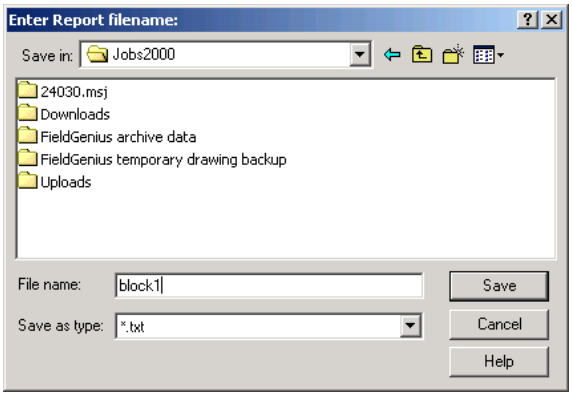


You may edit this file at any time and when finished you can print the entire file by going to the FILE menu and picking the PRINT command. If you wish to only print a portion of the file then simply highlight that portion by picking and dragging your mouse over the text. Then go to the print command and choose the option of printing the highlighted data rather than the entire document.

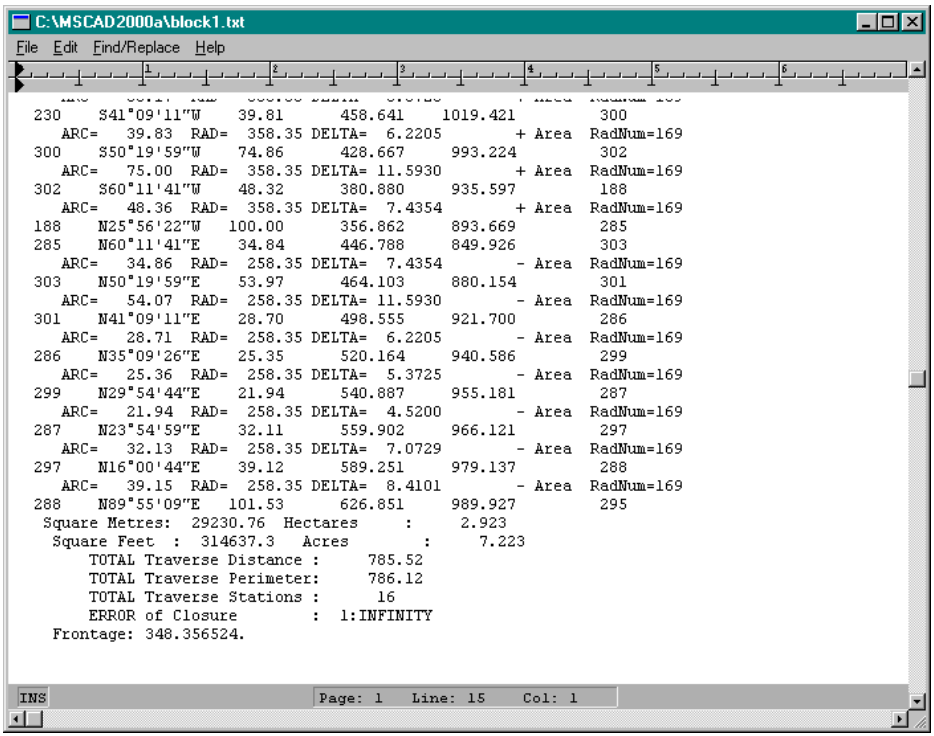
The MicroSurvey Log File Editor has it's own built in Help File.

Block Report

Give the file a name to save the block information.

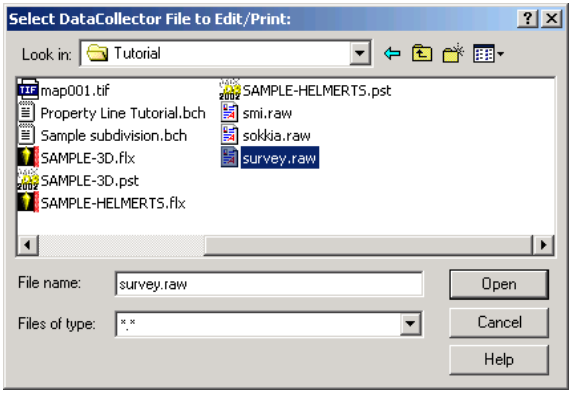


Pick entities from the Blocks on which to report: Select objects:

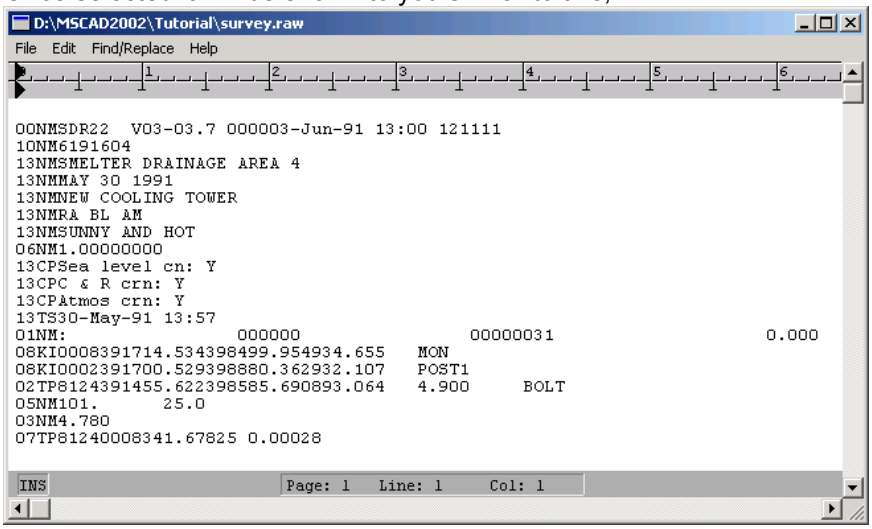


Data Collector Raw File

You will be asked which collector file you wish to view/edit.

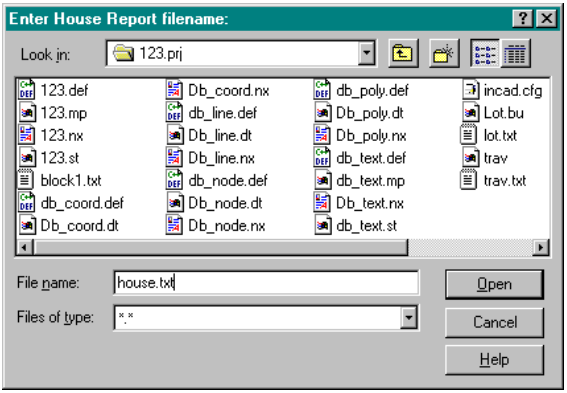


Once selected it will be shown to you similar to this,

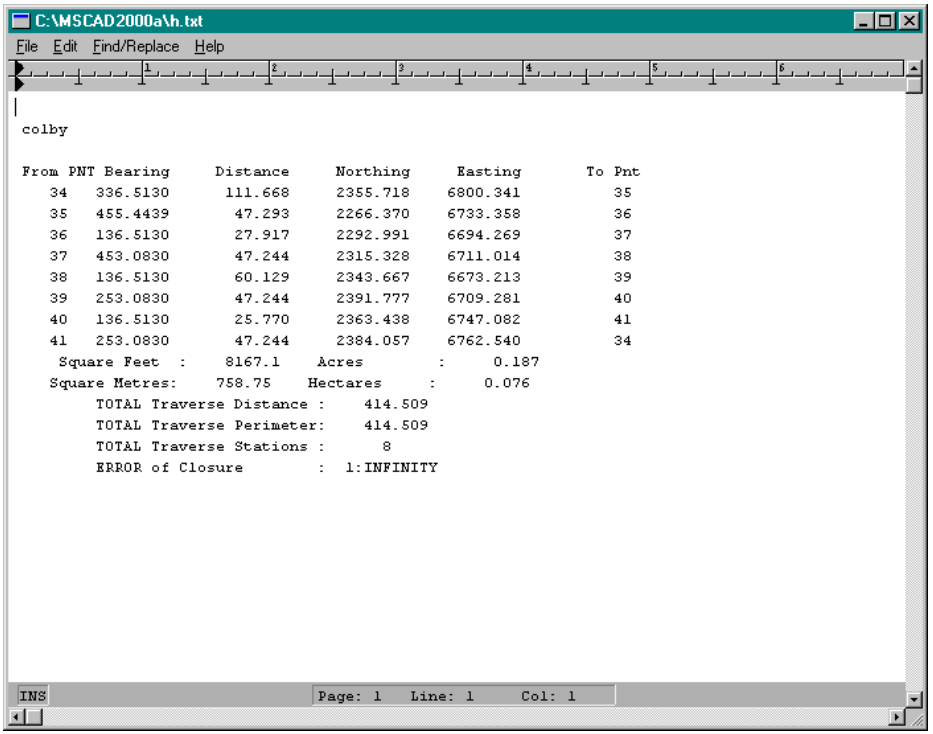


House Report

You will be asked for a file name for the information about the houses.

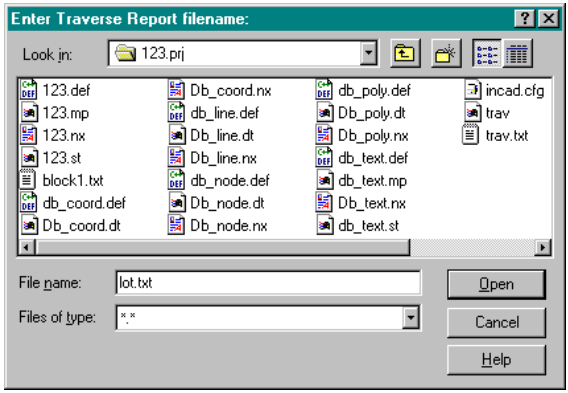


Pick entities from the Houses on which to report: Select objects:

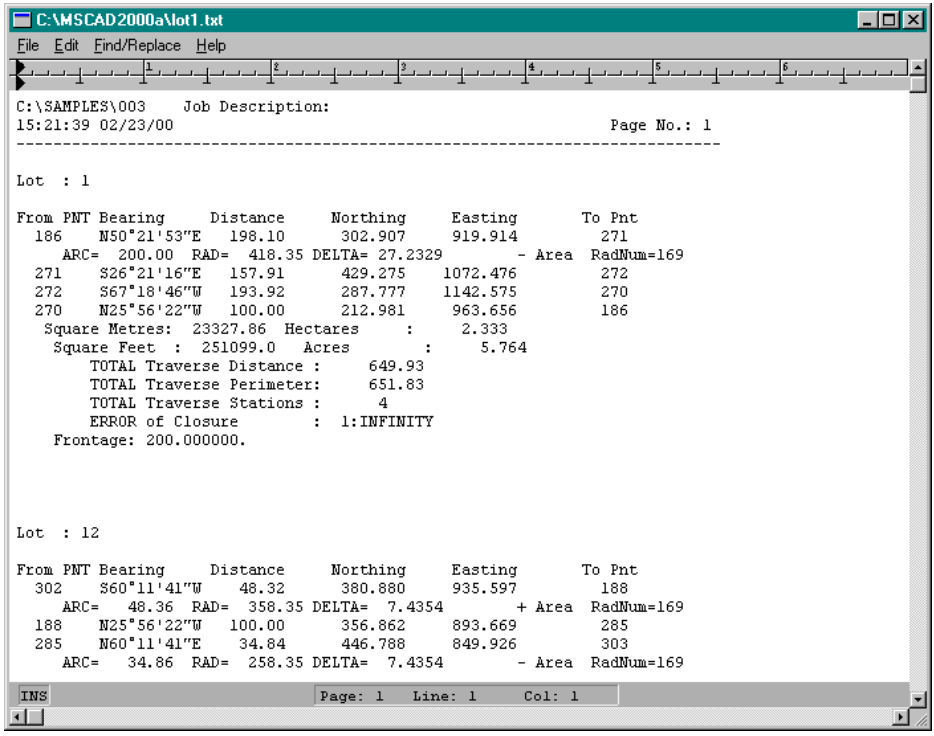


Lot Report

You will be asked for a file name to save the information about the lots.

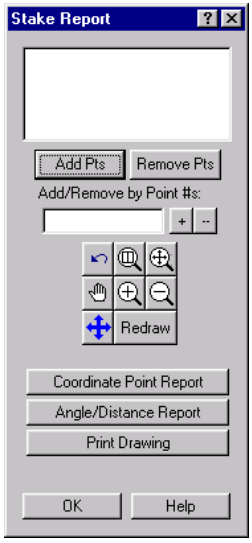


Pick entities from the Lots on which to report: Select objects:



Overview – Stakeout Reports

Prepare Stakeout Reports with this simple two-step process:



1. Choose the stakeout points to include in the report by one of the following two selection methods (do not include the setup point in this list):

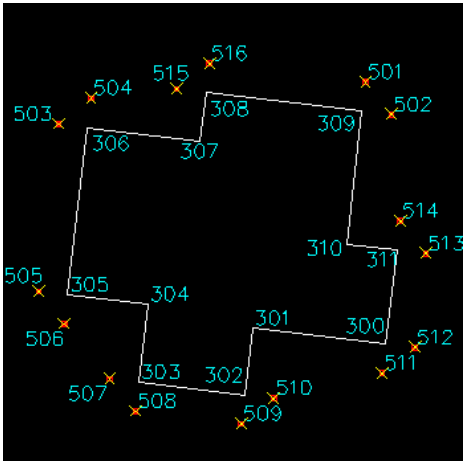
Method 1: Selecting Objects

1. Pick the Add Pts button to Add or the Remove Pts to Remove points from the Report List.
2. Using any of the Object Selection methods, pick the dot that is located where the coordinate point is located. This will be in the center of a stake, or at a building or lot corner. You need to make sure you are selecting the dot. Use the Wpolygon, Window, Crossing, and Cpolygon methods to select points.
3. When you have selected objects, press Enter to add them. The Report List points will be indicated with a small, yellow X on the drawing. This X is temporary. If you exit this routine, the X will disappear.

Method 2: Entering Point Numbers

1. Enter a single number or a range of points into the edit box. If you are entering a range, separate the starting point from the ending point by two dots (..)
2. Pick the + button to add points to the Report List. Pick the -- to Remove points from the Report List.

A small yellow X on screen indicates points included in the Report List:

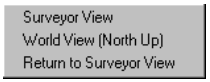


2. Choose the type of report you would like from one of the three available MicroSurvey Stake Report formats:
   - Coordinate Listings of field points to use with field equipment
   - Angle / Distance report showing setup, backsight and field data
   - Graphics Printouts to take in the field showing different views of the points to stake

MicroSurvey generates reports and places the results in the Internal Windows Text Editor. The program has print previews and supports all Windows printers.

Drawing View Rotation

MicroSurvey normally displays in “World View with North oriented toward the top of the page. You may alternatively select “Rotated View and re-orient North to any direction on the plot.



Surveyor View

Selecting MicroSurvey Surveyor View allows you to change the orientation from the “World View (where North is oriented pointing toward the top of the page) to an arbitrary Rotated View where north can be oriented in any direction. Your points will not be altered in the database and all calculations will be correct even after north has been re-oriented. Your cross hairs will continue to Point straight up the page and not follow the rotated view.

You will be prompted for two points that will define a horizontal line on the new Surveyor View orientation. All subsequent displays will honor the new orientation.

If you know you are going to use this option please make sure that the rotation values for Point Numbers, Elevations and Descriptions are set correctly to look correct after the new orientation. If you do not do this ahead of the new orientation then you will have to do the changes to the default settings to set the rotation angles and then do an update of the drawing from the database, or do a Rescale of the entire drawing, to correct the text orientation.

The view may be set back to North pointing toward the top of the page by selecting World View.

World View

This is the default view orientation for MicroSurvey. In World View, North points towards the top of the page. Picking this command will take your drawing, no matter what orientation it was placed in by the Rotated View command, and re-orient it so north is straight up the screen again.

Return to Surveyor View

This allows you to flip back to the Surveyor View you had previously set but without having to run the setup again.

Text Entry and Editing

The Text Entry and Editing submenu lets you add and edit text in your drawing, and control the fonts and sizes in which it is entered. This menu offers two parts: one for Entering Text and one for Editing Text.

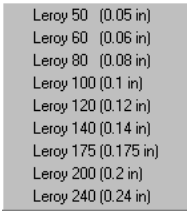
Pre-Sized Text Styles

Add New Text

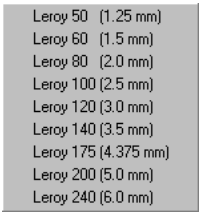
When you click on one of the font names

a selection box will appear with the available Leroy sizes, from 50 to 240. Click on your choice of size (between Leroy 50 and 240) and MicroSurvey will initiate a Text command, with your selected font.

On this menu we also show you the size of the text on paper (when plotted) in metric and imperial (based upon which units you chose to work in at the beginning of the drawing).



or



Here is an example of each font style we provide you with.

The Text command pulls up a dialog box that allows you to enter multiple lines of text. It allows you to set the justification, the height, the style, and the alignment. After you have entered the text, pick the **OK** button to place the text.

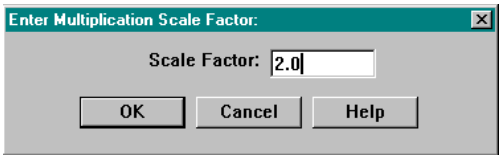
After you have placed the text on the screen you will now be asked for the rotation of the text to allow you to align the text with anything on the screen. All lines of text being entered in this command will rotate the

same amount. If you press enter at this prompt, zero degrees will be assumed.

Scale one Text Size to Another

This command re-scales all selected text objects to a new size. In order to use it, you must either know the present and desired text heights in drawing units, or have samples of text of these heights onscreen. See the on-line help for more details.

Scale Selected Text by Factor



This command will do a global re-scale of all the text you select at one time by a scale factor. This is extremely useful for re-scaling a drawing to a different scale factor. It should not be used to re-scale MicroSurvey Text items like bearings, distances, point numbers, elevations, or descriptions. These items will be replaced with better results if you use the Drawing Rescale program under Drawing Utilities. This program has only one prompt to select all the text you wish to re-scale. Then it displays a dialog box asking for the text scale factor. When you enter the scale factor, the program will replace all the text at the original text insertion point.

Rotate Text 180 degrees

This command rotates a text line by 180 degrees. You are prompted to select the objects. On completion, you are asked for another text object; you can select another to rotate, or terminate the command by pressing Enter. This program is mainly for rotating bearings and distances that are not in the correct rotation.

Slide text along line

This program slides text along the line parallel to the text rotation. Pick the text you want to slide, and drag it to the new location. The program sets the ortho and snap modes to rotate the cross hairs to the rotation

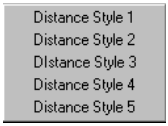
of the text. This program makes it much easier to place text along a line where it cannot be located in the middle of the line.

Bearing/Distance/Curves

This selection brings up a submenu of options for adding bearing and distance labels to lines and point pairs in your drawing. It has submenus called Styles, Lines, Curves, Manual Labeling and Two Points. Options for automatic labeling of Lines and Curves are shown on the Bearing/Distance Labels submenu itself; the other items appear on separate submenus.

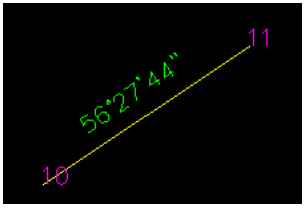
Bearing and Distance Styles

This option brings up a submenu of five styles each for bearing and distance labels. Clicking on one of these styles will place that font, and its associated MicroSurvey layer, in effect.



For example, if you click on Distance Style 2, all subsequently entered bearings will appear in style DISTANCE2 and will be drawn on layer DISTANCE2. You can edit the text styles with the Font command and set up the defaults settings for each distance style under the Labeling Defaults for distances.

Lines: Auto Bearing

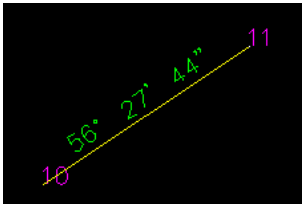


This command adds a bearing above or below a line. You can select as many lines as you like, with the usual MicroSurvey object selection methods. When you press Enter to indicate your selections are complete, MicroSurvey will designate each line in turn with an arrow and ask you to indicate on which side to place the bearing; pick a point

on the appropriate side. When all the lines are done, you will be returned to the object selection prompt; you can designate more lines or press Enter to terminate the command.

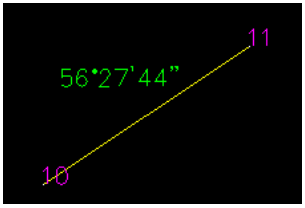
See the on-line help for more details.

Lines: Auto Split Bearing



This command is identical to Auto Bearing, except that it splits the bearing into three separate entries for degrees, minutes and seconds, spacing them along the line at one-quarter, one-half and three-quarters of its length. Since these entries are separate text entities, they will NOT update if the line is moved.

Lines: Auto Bearing Off Line

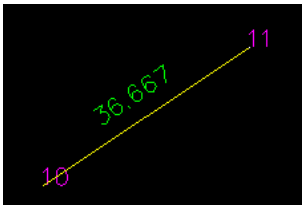


This command is identical to Auto Bearing, except that it places the bearing anywhere you specify on the drawing. You will be prompted for a location and rotation angle for the bearing text.

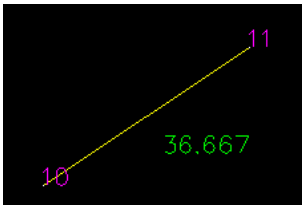
A bearing placed with this command is related to the line and points; if the line is moved, the bearing will update automatically.

Lines: Auto Distance

This command enters a distance above or below a line in the same way that Auto Bearing enters a bearing. The distance is related to the line, and if the points move, the distance will update automatically. The distance is calculated by inversing between the two coordinates. The distance is always the horizontal distance between the points. If the line is trimmed in AutoMAP to make room for symbols at the endpoints, the distance will still be correct.

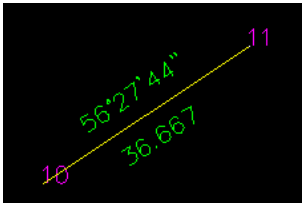


Lines: Auto Distance Off Line



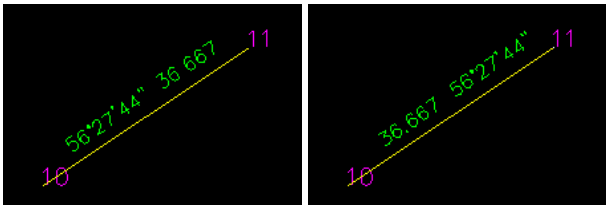
This command is identical to Auto Distance, except that it places the distance anywhere you specify on the drawing. You are asked for the location and rotation angle of the distance text. The distance is related to the line and its defining points; if you move the points, the distance will update.

Lines: Auto Bear/Dist



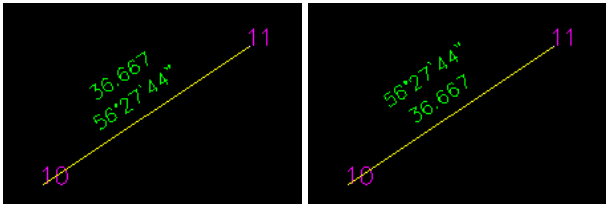
This command adds both the bearing and distance to the line, one above and one below. You are prompted for the side where the bearing will be located. Both the bearing and distance are related to the points; if you move either point, the bearing and distance will update automatically.

Lines: Auto Bearing/Distance or Distance/Bearing



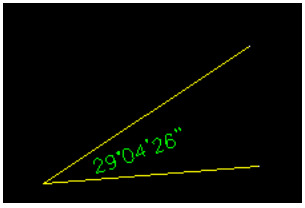
These commands are identical to Auto Bear/Dist, except that they place the bearing and distance side by side. Auto Bear-Dist places the bearing first, and Auto Dist-Bear places the distance first. Both the bearing and distance are related to the points; if either point moves the bearing and distance will update automatically.

Lines: Auto Dist/Bear/Line



These commands are identical to Auto Bear/Dist, except that they place both bearing and distance on the same side of the line. You are prompted for which side to use. Auto Dist/Bear/Line will place the distance farthest from the line; Auto Bear/Dist/Line will place the bearing farthest. Both the bearing and distance are related to the points; if the points move, the bearing and distance will update automatically.

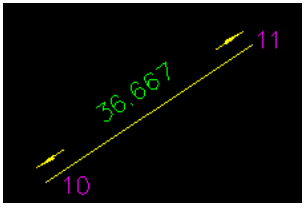
Lines: Auto Add Angle



This command adds an angle between two lines. You define the angle by picking a point on each line, and their intersection point. The order in which you select the lines is critical, since the angle will be calculated clockwise from the first selected line to the second. On completion, you will be prompted for another line; you can continue labeling angles, or terminate the command by pressing Enter.

The angle entered by this command is static, and will NOT update if the lines are moved. The angle will be added by selecting clockwise.

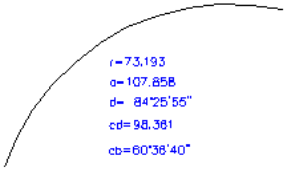
Lines: Auto Arrows on Line



This command will add a set of arrows to one side or the other of a line you choose. The program will prompt for the name of the arrow block to insert. MicroSurvey is supplied with 3 different arrow types: MMARR, MMARR1, and MMARR2. These are standard MicroSurvey drawing files. You can make a copy of one of the files, manipulate it, and save it for future use. You can make your own arrow with a similar scale. The arrows are automatically re-scaled when inserting into the drawing based upon the Drawing scale factor in the Main Defaults.

The distance shown here was added separately.

Curves: Label OFF Curve



This program allows for very flexible curve labeling. The prefix for each item can be placed on the drawing or skipped.

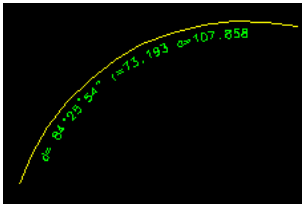
The delta prefix can be regular text, or it can be the small triangle delta symbol. This symbol is only available if you include the font file MSURVEY.FSH. You are granted the legal right to distribute this font file, with your drawings if necessary to other companies working with your drawings.

This command enters a variety of curve information anywhere on the drawing. You can position each item independently if you wish.

Hot tip: When prompted for text rotation, simply right-click or press the enter key to place the text horizontal.

Note: Information entered by this command does not update if the arc is changed.

Curves: Label ON Curve



This command adds curve information (radius, arc length, delta angle, chord bearing and chord distance) to a curve on the plan by writing text in a curved format.

Note: Information entered by this command does not update if the arc is changed.

Curves: Text on ARC

This command enters a text string, which you supply, along an arc. An arc does not have to be present in the drawing; you are simply prompted for a center point and the center of the text string, and asked

whether the text should be base-in or base-out. The distance from the center point to the text center point will specify the arc radius of the text.

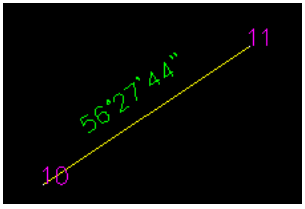
On completion, you will be prompted for the center of another text string to be entered around the same radius point. You can continue to enter text strings, or terminate the command by pressing Enter.

Manual Labeling

Lines: Manual Labeling

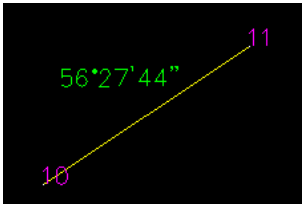
The Manual Labeling selection pops up a submenu which offers options for entering both line and curve information which you type in, instead of having it computed by MicroSurvey.

Lines: Man Bearing



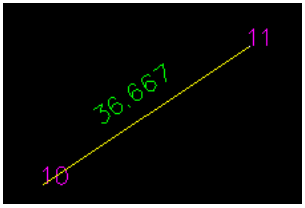
This command lets you add a bearing or azimuth by typing in the data. You are prompted to select a line; to choose on which side to place the bearing; and to type in the value. On completion, you are asked for another line; you can continue with the command, or terminate it by pressing Enter.

Lines: Man Bearing Off Line



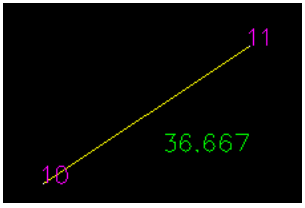
This command is identical to Man Bearing above, except that it places the bearing anywhere on the drawing. In addition to the prompts described above, you will be asked for a location and rotation angle for the bearing.

Lines: Man Distance



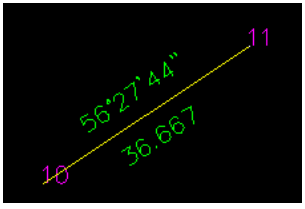
This command adds a typed-in distance to a line. You are prompted to select a line; to choose on which side to place the distance; and to type in the value. On completion, you are asked for another line; you can continue with the command, or terminate it by pressing Enter

Lines: Man Distance Off Line



This command is identical to Man Distance, except that it places the distance anywhere on the drawing. In addition to the prompts described above, you will be asked for a location and rotation angle for the distance.

Lines: Man Bear/Dist

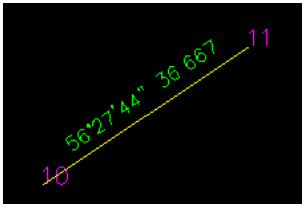


This command adds a bearing and distance on opposite sides of a line, with the values entered manually. You are asked to select a line; to choose the side on which to place the bearing; and to enter the bearing and distance values.

The distance will be entered on the side opposite the bearing. On completion, you will be asked for another line; you can continue with the command, or terminate it by pressing Enter.

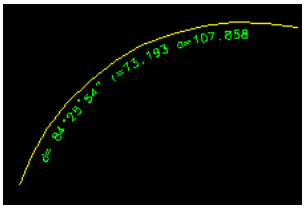
If the MicroSurvey Units is set for Surveyor's Bearings, MicroSurvey will expect the bearing entry to be prefixed with a quadrant digit of 1 for NE, 2 for SE, 3 for SW or 4 for NW. For example, a bearing of N42d21'32"W would be entered as 442.2132.

Lines: Man Bear-Dist



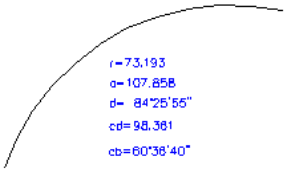
This command is identical to Man Bear/Dist above, except that it places both bearing and distance on the same side of the line. Bearings and azimuths are entered in the same manner as Man Bear/ Dist.

Man Label ON Curve



This command enters curve information along a curve in the same manner as Auto Label ON Curve, except that you type in the information. In addition to the automatic labeling prompts, you are prompted for values for the parameters.

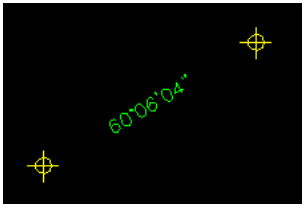
Man Label OFF Curve



This command enters curve information anywhere on the drawing in the same manner as Auto Label OFF Curve, except that you type in the information.

Two Points Labeling

Two Points: Bearing

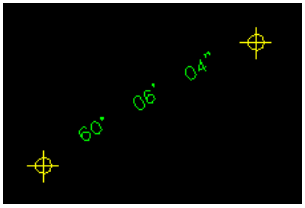


This command enters a bearing between two selected points. You are simply prompted for the two points in sequence. The command will display the information as a quadrant bearing or an azimuth depending on the setting of the Reference North toggle.

On completion, you will be asked for another set of points; you can continue with the command or terminate it by pressing Enter.

The bearing entered by this command is NOT related to the points and will not automatically update if they move.

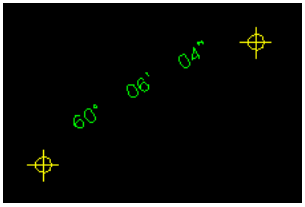
Two Points: Split Bearing



This command is identical to Two Points: Bearing, except that the bearing appears in three segments for degrees, minutes and seconds, placed respectively at the one- quarter, one-half and three-quarter points along the distance between the defining points.

The bearing entered by this command will not update automatically if the points change.

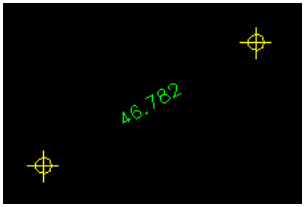
Two Points: Split Bearing Offset



This command is similar to Split Bearing, except that the bearing information will be offset from the line connecting the two defining points. It is useful when you want to annotate a long line by picking the endpoints.

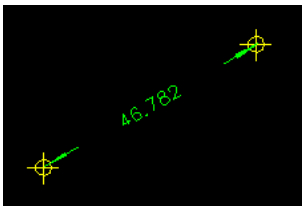
You will be prompted for the side on which the bearing is to be placed; pick a point on the appropriate side. The bearing entered by this command will not update automatically if the points change.

Two Points: Distance



This command enters a distance between two points. You are prompted to select two points. On completion, you are asked for another set of points; you can continue with the command, or terminate it by pressing **Enter**.

Two Points: <Distance with Arrows>

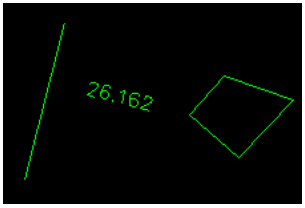


This command enters a distance between two points and indicates it with dimension arrows. You are prompted to select two points and choose an arrow style. On completion, you are asked for another set of points. You can continue with the command, or terminate it by pressing Enter.

Three arrow styles are included with MicroSurvey: a standard arrow called MMARR; a backward-headed arrow called MMARR1, for restricted spaces; and a long-leader arrow called MMARR2, for longer distances. These arrows are contained in files MMARR.FLX, MMARR1.FLX, and MMARR2.FLX, respectively; you can customize the arrows with MicroSurvey.

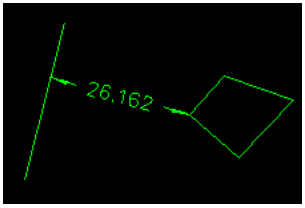
Ties/Leaders/Symbols

Auto House Tie



This command is identical to <-Auto House Tie->, except that it does not add the dimension arrows.

Auto House Tie with Arrows



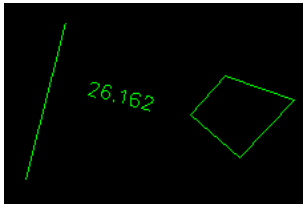
This command places a distance with dimension arrows on the plan, measured from a corner to a property side perpendicular to the property side. You are prompted to select the corner and the lot side, and to choose the type of arrow you want. On completion, you will be

asked for another tie location; you can continue with the command, or terminate it by pressing Enter.

Three arrow styles are included with MicroSurvey: a standard arrow called MMARR; a backward-headed arrow called MMARR1, for restricted spaces; and a long-leader arrow called MMARR2, for longer distances. These arrows are contained in files MMARR.FLX, MMARR1.FLX, and MMARR2.FLX, respectively; you can customize the arrows with MicroSurvey.

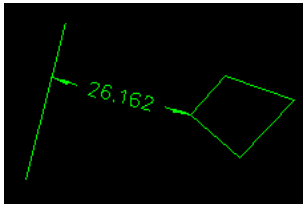
Since this command uses highly specific entries, it automatically places MicroSurvey in the correct OSNAP mode for each pick: INT for the house corner and PERP for the property side. The distance will be centered between the house and property line, and will be drawn in the current text style. The number of decimal places is determined by the current setup parameters. The distance placed by this command will update automatically if the house corner or property line moves.

Man House Tie



This command is identical to <-Man House Tie->, except that it does not place the dimension arrows.

Man House Ties w/Arrows



This command is similar to <-Auto House Tie->, except that you enter the distance manually.

Three arrow styles are included with MicroSurvey: a standard arrow called MMARR; a backward-headed arrow called MMARR1, for restricted spaces; and a long-leader arrow called MMARR2, for longer distances. These arrows are contained in files MMARR.FLX,

MMARR1.FLX, and MMARR2.FLX, respectively; you can customize the arrows with MicroSurvey.

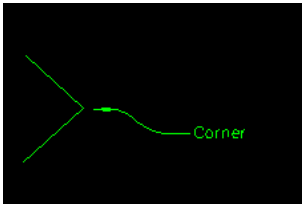
Multiple Perpendicular Ties

This command allows you to specify a baseline by typing in the two point numbers, as shown below, and then specify the range of points to calculate distances along the baseline and then perpendicular out to the points. A nice report is generated for you showing the baseline and offset distances. If the baseline distance is negative then the direction of travel is backwards. If the offset distance is negative then it is to the left of the baseline.

NOTE: The report can be generated in two formats, 1) Bearing/Distance along the baseline and Bearing/Distance off the baseline to each point, or 2) as a baseline/offset distance.

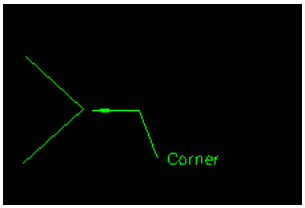
See the on-line help for more details.

Curved Leader



This command draws a curving leader to point to an item, using an Arc command to create a polyline made up of arc segments. You are prompted to select a start point for the leader, and to specify the direction of the arrowhead; then you can simply “pick the path of the leader until you are satisfied with the layout. When done, terminate the polyline by pressing ESC.
Note – The size of the arrow head is controlled by the Dimensioning variable in the Dimension settings. The command to set the dimension variables is DIMTYPE. Then pick the Lines and Arrows button and experiment with the value on the right called Block Size. If you change the Block Size, any existing Leader Arrowheads will NOT be updated. You would have to redraw the Leader to get the new size. Normally you would enter very small numbers in to this value, but the size is scale dependent.

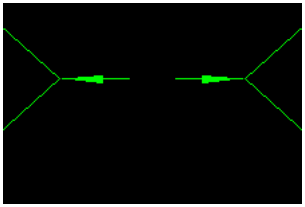
Straight Leader



This command draws a straight- segmented leader to point to an item, using a MicroSurvey command to create a polyline. You are prompted to select a starting point for the leader, and to specify the direction of the arrowhead. Then you can draw a series of line segments back from the arrowhead until you are satisfied with the leader layout. Terminate the leader by pressing Enter.

Note – The size of the arrow head is controlled by the Dimensioning variable found under the Detail menu -> Dimension Type… -> Lines and Arrows -> Block Size. If you change the Block Size, any existing Leader Arrowheads will NOT be updated. You would have to redraw the Leader to get the new size.

<-Arrows->



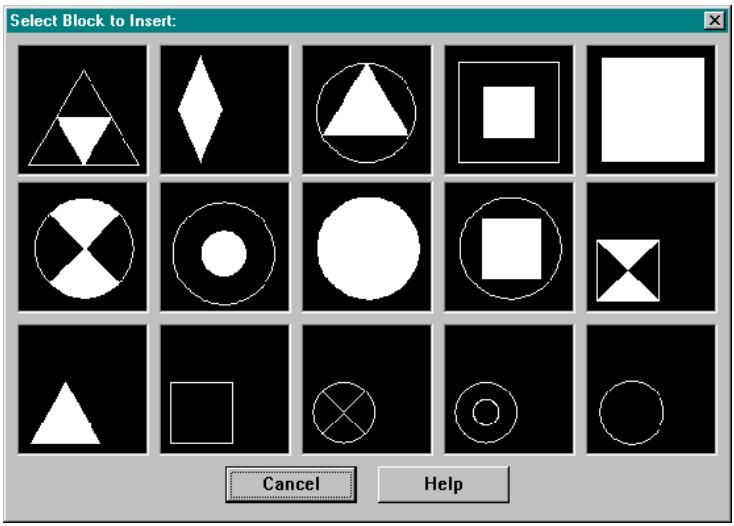
This command adds arrows to the drawing to indicate a distance or bearing. You are prompted to select the two endpoints, and to choose the type of arrowhead you want. On completion, you will be asked for another set of points; you can continue with the command, or terminate it by pressing Enter.

Three arrow styles are included with MicroSurvey: a standard arrow called MMARR; a backward-headed arrow called MMARR1, for restricted spaces; and a long-leader arrow called MMARR2, for longer distances. These arrows are contained in files MMARR.FLX, MMARR1.FLX, and MMARR2.FLX, respectively; you can customize the arrows with MicroSurvey.

Posts Icons #1

Metric and Inch Post Icons are identical except for scale. The symbol names on the hard drive start with ici_ for imperial symbols and icm_ for metric symbols.

This command enters one of 12 common symbols on the plan. Initiating the command pops up an icon menu of symbols, which are resident in MicroSurvey's AutoMAP block database. You may modify how the symbols are placed by going to AutoMAP and modifying the options for each symbol. They are saved in the MSCAD.CSV file by default. Select one of the icons and click on OK.



You then receive the prompt

Entity/Pick/Number:

If you enter a point number, MicroSurvey will draw the symbol at the point bearing that number. You can also press **E** and then pick the point number on the drawing, or press **P** and then pick the point itself. You can, of course, use the OSNAP commands to pick the point accurately. You are then prompted for rotation angle and the block is inserted.

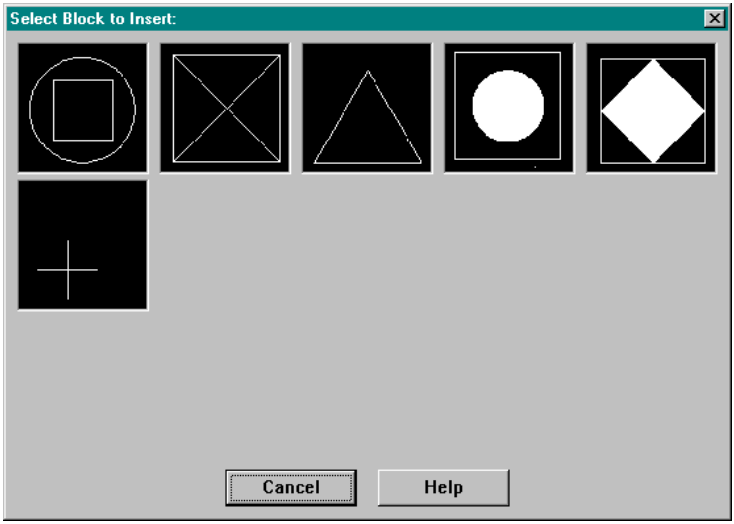
Each time you pick a point, you will be returned to the same prompt. You can continue picking points, or terminate the command by pressing Esc.

If you pick a point that is not listed in the coordinate database, MicroSurvey will add it to the database automatically. If you re-compute the point later, the symbol will automatically move with it.

Posts Icons #2

Metric and Inch Post Icons are identical except for scale

This command is identical to Metric Posts Icons #1, except that the popup icon menu offers an additional set of common symbols:



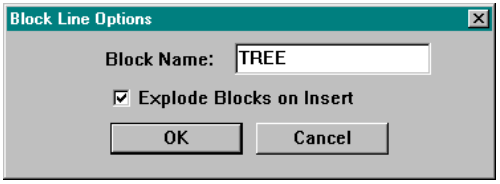
Custom Posts Icons

This command is an example of how you can modify the menus to insert all your own block symbols with automatic scaling.

See the on-line help for more details.

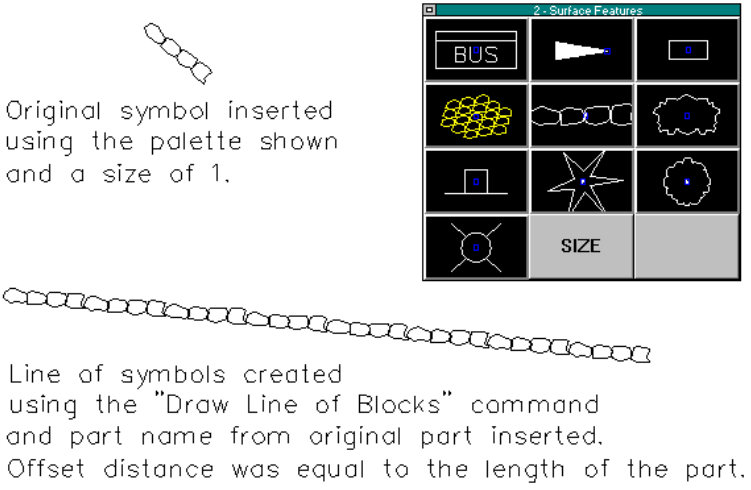
Draw Lines of Blocks

This program will draw a series of repeated blocks in a line defined by you. When you start the program, the following dialog box appears:



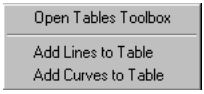
Enter the block name and indicate whether the block should be exploded upon insertion.

You will be prompted for two points defining the path along which the blocks will be inserted, then you will be prompted for an offset distance. The offset distance is the distance between adjacent blocks.



Line and Curve Tables

MicroSurvey has a powerful set of program to build line and curve tables automatically.



Click on the command to display the topic.

Curve and line table examples:

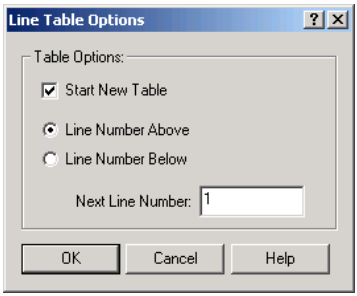
NUM	DELTA	RADIUS	ARC	BEARING	DISTANCE
C1	51°38'13"	524.443	472.648	323°01'38"	456.814
C2	36°55'38"	826.459	532.656	330°10'35"	523.485
C3	109°56'22"	42.000	80.590	84°00'46"	68.783
C4	81°52'12"	42.000	60.014	348°06'28"	55.037
C5	96°46'12"	42.000	70.936	77°25'41"	62.800
C6	96°46'12"	25.000	42.224	77°25'41"	37.381
C7	73°29'02"	42.000	53.867	352°18'03"	50.250
C8	96°27'06"	42.000	70.703	282°36'56"	62.645
C9	300°00'00"	133.000	696.386	299°02'35"	133.000

NUM	DISTANCE	BEARING
L1	937.524	348°38'24"
L2	895.289	29°02'35"
L3	860.064	348°38'24"
L4	860.064	348°38'24"
L5	633.246	29°02'35"
L6	864.154	29°02'35"

Line Tables

This command will build a line table. The line table will contain the line number, the bearing, and the distance of the line. The Start New Table option will determine whether a header is drawn for the table. The Lines on the drawing will be labeled with an L followed by the line number assigned by this program, either above or below the line based upon your choice in the dialog. The lines will be numbered in the order that MicroSurvey returns them in the selection set. If you want to control the numbering precisely, select the lines one at a time instead of using the crossing command. The program uses the current bearing text style for the line table. The bearing label default, Text Offset, is used to place the L1, L2, etc. on the lines at the correct distance from the line. The table is placed on the current layer.

The Next Line Number option determines the line number to use on the table and plan. An example of the output from this program:

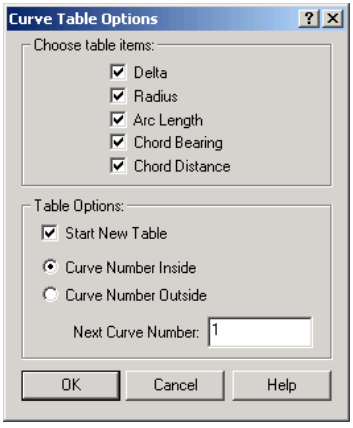


NUM	DISTANCE	BEARING
L1	937.524	348°38'24"
L2	895.289	29°02'35"
L3	860.064	348°38'24"
L4	860.064	348°38'24"
L5	833.246	29°02'35"
L6	864.154	29°02'35"

Adding to Line Tables

Adding to a line table is easy. You will be prompted for the bottom left corner of the existing table. If your scale has not changed, and the text size is the same, then the program will continue the table with exactly the same text size and spacing as before.

Curve Tables



You can choose any one of the Delta Angle, Radius, Arc Length, Chord Bearing, and Chord Distance for your table. The Start New Table option will determine whether a header is drawn for the table. The

Curves on the drawing will be labeled with a C followed by the curve number assigned by this program, either on the inside or outside of the curve depending on you choice in the dialog. The curves will be numbered in the order that MicroSurvey returns them in the selection set. If you want to control the numbering precisely, select the curves one at a time instead of using the crossing command. The program uses the current bearing text style for the curve table. The bearing label default, Text Offset, is used to place the C1, C2, etc. on the arcs at the correct distance from the arc. The table is placed on the current layer.

The Next Curve Number option determines the curve number to use on the table and plan. Here is an example of the output from this program:

NUM	DELTA	RADIUS	ARC	BEARING	DISTANCE
C1	51°38'13"	524.443	472.648	323°01'38"	456.814
C2	36°55'38"	826.459	532.656	330°10'35"	523.485
C3	109°56'22"	42.000	80.590	84°00'46"	68.783
C4	81°52'12"	42.000	60.014	348°06'28"	55.037
C5	96°46'12"	42.000	70.936	77°25'41"	62.800
C6	96°46'12"	25.000	42.224	77°25'41"	37.381
C7	73°29'02"	42.000	53.867	352°18'03"	50.250
C8	96°27'06"	42.000	70.703	282°36'56"	62.645
C9	300°00'00"	133.000	696.386	299°02'35"	133.000

Adding to Curve Tables

Adding to a curve table is easy. You will be prompted for the bottom left corner of the existing table. If your scale has not changed, and the text size is the same, then the program will continue the table with exactly the same text size and spacing as before. The table is placed on the current layer.

Layer Control

Mass Layer Conversion

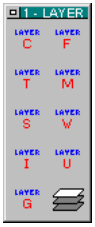
Originally created for Alberta Land Title Office digital submissions, this tool is very powerful and can convert any set of layer names to another. This is to allow you to submit drawings using the layer structure your client desires but still allow you to work with your own layer structure in your own office.

Very simply, you set up the layer mapping using the Layer Table Editor, run the conversion and create a report. Then you may check the objects on each layer to make sure they are correct and change them if required. You now have a drawing with the layers you require to submit to your client.

See the on-line help for more details.

Layer Tools Palette

This invokes the Layer Control Tools Icon Palette of icon shortcuts for rapid access to the layer control commands.



Copy Layers

Copies a group of selected entities to a layer specified by picking an object on that layer.

You will be prompted:

Select item on target layer: Choose an object on the target layer

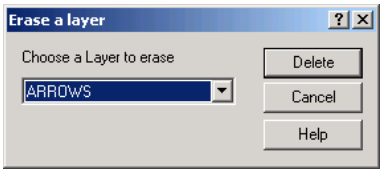
Select items to COPY to layer POINTNUMBER: Select objects to copy

In this example a point number residing on the POINTNUMBER layer was chosen as the target, so all selected entities will be copied to the POINTNUMBER layer.

Erase Layers

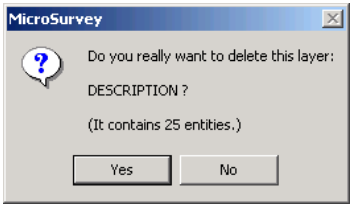
Erases one layer at a time. This function will completely erase the layer definition and **all** objects that on that layer. **Note:** There is no UNDO for this function.

Select the layer you wish to delete from the pull-down list in the dialog:



Very Important WARNING: Erasing the MSPOINT layer will destroy your coordinate database!

You will get a chance to change your mind!



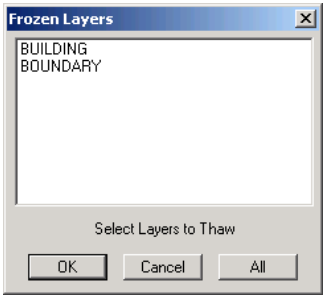
Click OK to complete the layer erase function. **Remember:** You cannot UNDO this function!

Freeze Layers

Freezes the layer of the selected item.

Thaw Layers

Displays a dialog of frozen layers and lets you select one or more to thaw. The dialog only displays layers that are currently frozen. The ALL button makes it easy to thaw all frozen layers at once rather than picking each one separately.



Move Layers

Moves a group of selected entities to a layer specified by picking an object on that layer.

Set Layers

Sets the current layer to the layer of the selected item.

What Layers

Reports the layer upon which the selected object resides. You may query any number of objects in succession. Press Esc to terminate the command.

Isolate Layers

This command turns off all layers except for the layers used by the objects you choose.

Un-Isolate Layers

This command turns back on all the layers you have turned off by the isolate command.

Group Layers

This command is the most powerful of all the layer commands. It allows you to group layers into named selections. These named selections are stored on your hard disk. They can be recalled at any time.

You need to start this command by picking Create. The Create command will prompt you for the name of the layer group. The layer group file is a standard ASCII text file with the default extension GRP. You can store the layer group files anywhere on your system. The standard file selection dialog box appears to store and read the group files. You add the layers to the group list by selecting objects on the screen.

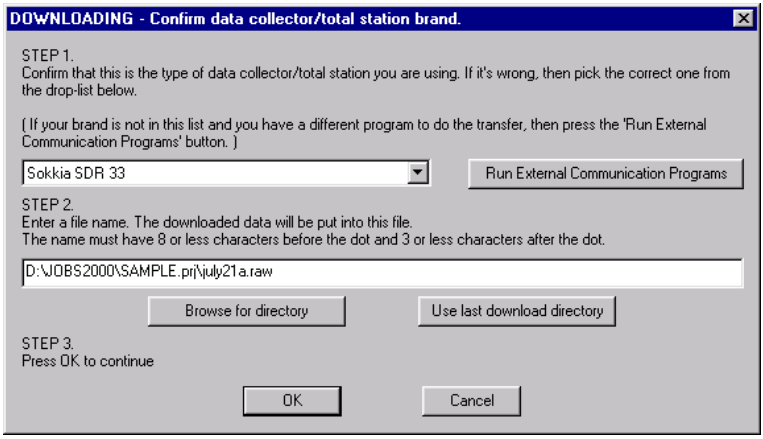
Once you have built the group, you can freeze, or thaw layers quickly. The View command lists the layers in the group file. The Edit command allows you to add or delete layers from the group list.

Download Data Collector

Step 1 of a 2-step process for downloading data and bringing it into your drawing:

Download Data Collector: We download the data from the equipment and store it on your hard drive. We prompt you to save it in the current project directory but you may browse to another folder and store it wherever you like. We also make a backup copy of the data file and store it in a safe backup directory. The stored files are normally in a pure ASCII Text format.

MicroSurvey supports direct serial communications to many data collectors. When you choose this option from the MicroSurvey menu you are presented with this dialog. Follow the instructions on the dialog:



Note: Some data collectors include the filename with the data download. When available, this filename will be used and MicroSurvey will not prompt you to enter a filename. You will still have the option to browse to an alternate directory in which to save your download file.

You will next see the confirmation dialog. This dialog shows the data collector and the current communication parameters specified. If you need to modify these settings, click on Change Current Parameters.

When you are satisfied that your settings are correct, click on the Download Now button. You will be provided with a dialog containing instructions for downloading the specified data collector. Simply follow the instructions in the dialog.

Keyboard command: DOWNLOAD

Now go to step 2 which will read your downloaded file into your drawing. Look for "Read Data Collector File."

Upload Data Collector

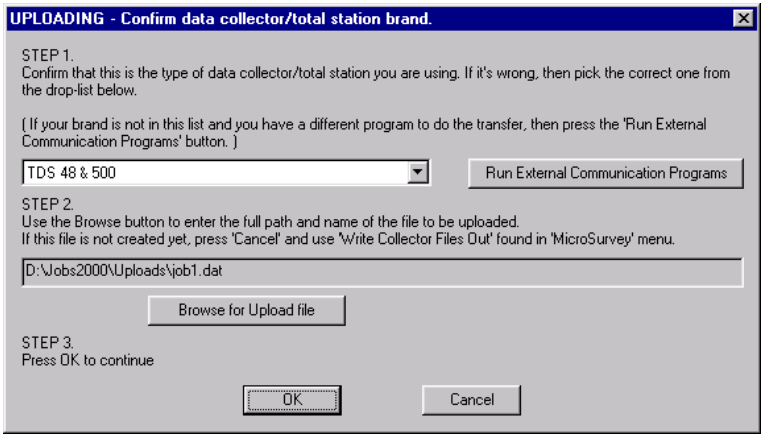
Uploading your data collector is a 2-step process.

1. **Write Data Collector file:** Create a coordinate file to upload to your specific data collector from the current drawing.

2. **Upload Data Collector:** Send the file to your instrument or data collector.

See the section on Writing Data Collector Files now if you have not already done so.

MicroSurvey supports direct serial communications to many data collectors. When you choose this option you will be presented with the following dialog. Follow the instructions on the dialog.



Keyboard command: UPLOAD

Read/Write Data Collector Files

Data Collector Formats

This section of the help file gives detailed descriptions of the file formats MicroSurvey uses to communicate with data collectors. **If you are reading the manual, please refer to the on-line help for all of the detailed documentation on the data collector file formats.**

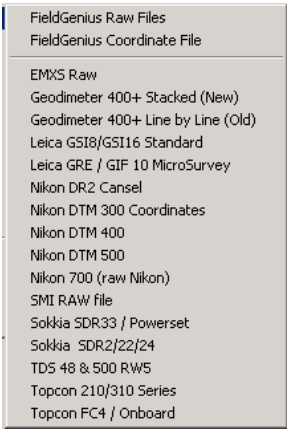
Read Data Collector Files

Step 2 of a 2-step process for downloading data and bringing it into your drawing. Please see the section on Downloading Data Collectors now if you have not already done so.

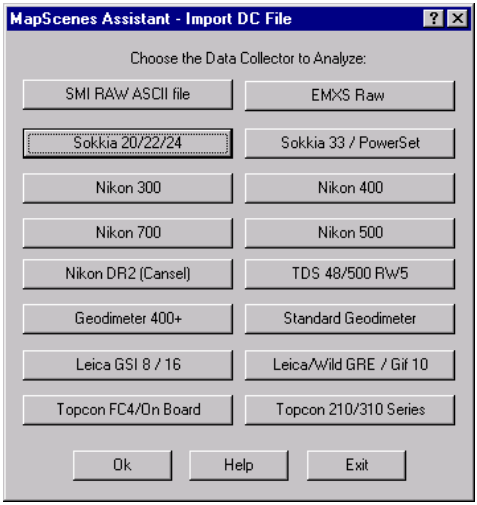
After your data collector has been downloaded, we move to the data processing routines to analyze the data that has been downloaded and stored in an ASCII file. This function can import downloaded coordinates directly into your drawing and adds the raw observations to a traverse file in preparation for processing. The traverse file can be edited prior to processing. Once any edits have been made, use Re-coordinate Traverse to compute and store coordinates for each point in the drawing and in the drawing coordinate database.

This selection provides you with a range of options for reading ASCII files that you have already downloaded from your data collector.

If you are working from our pull-down menus then you will see this menu.



If you are working from the Assistant you will see this dialog box.

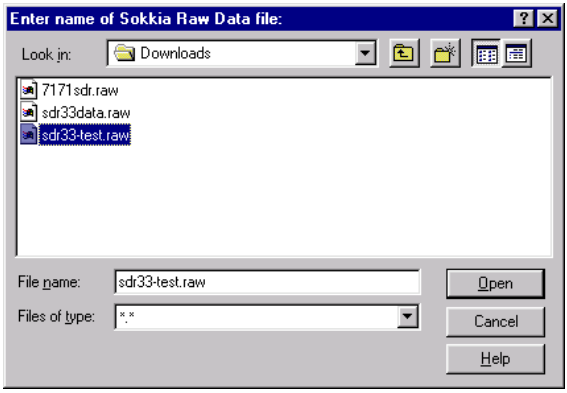


Pick on the menu or dialog box above to see collector file samples and more information about each format listed.

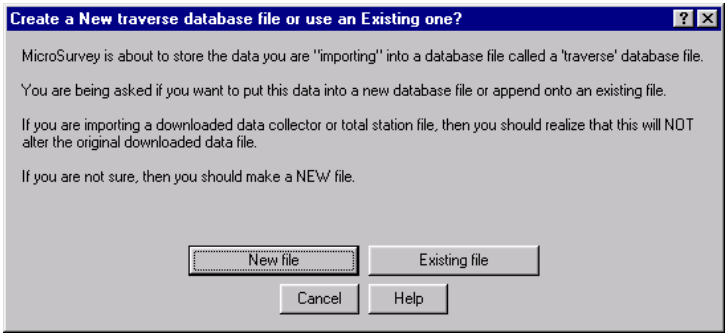
When selecting any of the options above, we immediately pop up a small dialog box called Import Toggle Check to let you know the status of, and let you update the status of, the Toggles that affect the reading of the data collector file.

Next we will ask you to locate and select the collector file, on your hard drive, that matched the format you chose above.

Note: The title bar of this dialog will depend upon the data collector make and model you are importing. The Example below shows the header if you are reading a Sokkia data file:

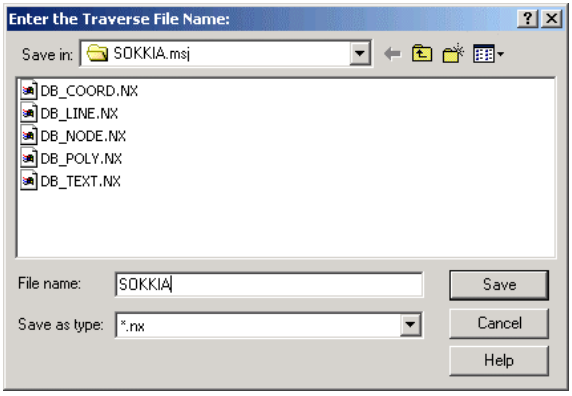


Then you will be asked,



In most cases you will choose the New Traverse File option as each file you read in will need to be calculated and edited separately. If you are merging 2 collector files, or adding to a manually entered file, then you may try the Existing Traverse option.

Now you need to identify the information by giving us a Traverse File Name. This allows us to go back to this set of data at any time to edit or coordinate it.



At this point the Collector file you chose is now read into the Traverse File you named. The information will scroll by in the command prompt area and then this routine is done and you are either returned to the previous dialog box (as seen at top of this section) or to a blank screen if you were working from the pulldown menu.

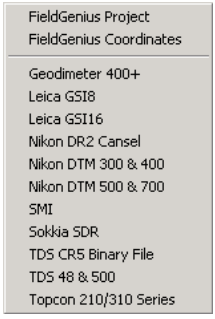
Write Data Collector Files

Step 1 of a 2-step process to upload a coordinate file to a data collector using MicroSurvey.

1. **Write Data Collector file:** Create a coordinate file to upload to your specific data collector from the current drawing.

2. **Upload Data Collector:** Send the file to your instrument or data collector.

This selection pulls down a menu of options for writing ASCII files for upload to your data collector



Traverse Coordinating & Balancing

Traverse Calculations Overview

When you enter raw traverse observations in MicroSurvey, either by downloading a data collector or by manually typing in the data, the data is stored in a traverse database file. Each traverse should be contained in a separate database file to make the best use of the conventional balancing routines. (This is not necessary if you plan to use the Least Squares Method that can use the raw observations in any input order) These database files are stored in the project directory when your file is open in MicroSurvey and running, and they are automatically compressed into the .MSD file when the file is closed.

There is a set order that you should use when working with raw data information in MicroSurvey:

1) Download your instrument or data collector file.

2) Manually Enter or Read Download data into a traverse file. If you are entering the data manually, points will be coordinated and drawn onto the MicroSurvey screen as you enter the data. If you download the data from a data collector, any coordinates in the ASCII file will be added to your drawing and stored in the drawing coordinate database. Raw data shots are not automatically processed and stored but are written to the traverse file so you have the opportunity to make corrections.

3) View and edit the raw data by selecting Edit Traverse. Enter any starting setup and backsight coordinates not already in the coordinate database. You can do this by editing the setup record in the Active Traverse Editor.

4) When you are satisfied that your raw data is correct, select Re-Coordinate Traverse from the Edit Menu in the Active Traverse Editor. There is also a short cut key available (Ctrl-R).This will compute coordinates for the points. If there is observation information in your traverse file to points already in the coordinate database, you may see the Point Protection dialog when they are re-calculated. If so, it indicates that the coordinates read in to the file vary with those computed from your raw data. It is up to you to determine which values are correct.

5) The calculated coordinates will be listed in the right columns in the Active Traverse Editor. Examine the coordinates and drawing for errors. If you find any problems, simply re-edit the traverse file and re-run the Re-coordinate traverse routine.

6) At this point, you may want to show or draw your traverse on screen.

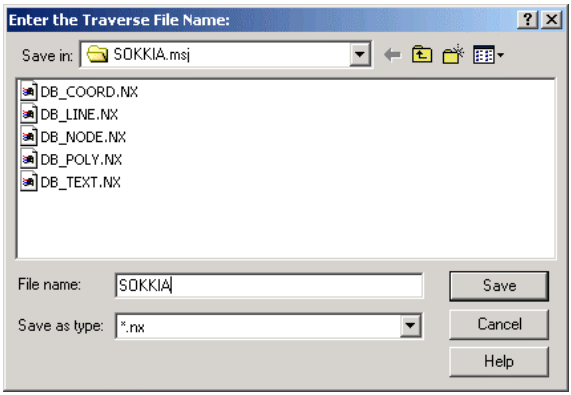
7) Now you should decide whether you wish to use conventional methods or Least Squares to balance or adjust your traverse if necessary. If you wish to use the Least Squares method, go directly to that function. For the conventional methods, go to Compute Closure to begin.

See the on-line help for more details.

Set Current Traverse File

This command allows you to set which Measurement file you are working with. This name will be used automatically when coordinating, editing or printing the measurement file.

This is the dialog you will see – DO NOT CHANGE THE FOLDER – just select or enter the current traverse file.



Note: Do not touch or use the file names already in this folder.

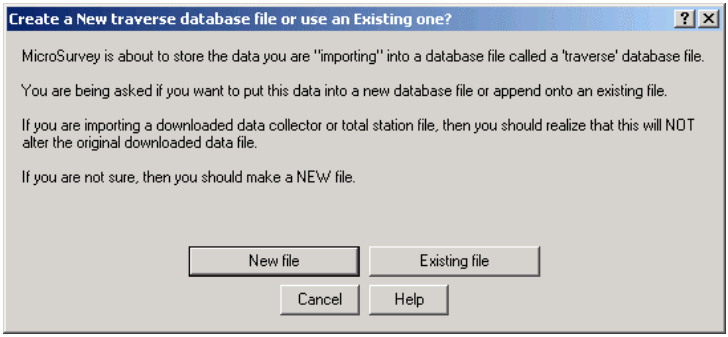
Manual Enter Traverse Data

This selection reads manually entered field note data into a Traverse data file. You begin by specifying an instrument setup, and then enter as many measurements from that setup as desired. You can re-specify the instrument setup at any time and continue entering measurements. All the observations are entered in a Traverse data file in the project directory along with the coordinate, connectivity and special text files.

When you re-coordinate your traverse, MicroSurvey will analyze the traverse file for multiple shots to any point, shots to the backsight, and doubled angle methods. This is very powerful and means that you can enter many shots to the same point from one setup. We will give you several options for averaging the multiple shot data to give you the best coordinate that can be calculated. You can enter several sets of shots with doubled angles, for surveys such as Third Order or better survey loops. Once you move to a different setup then the routine will start the

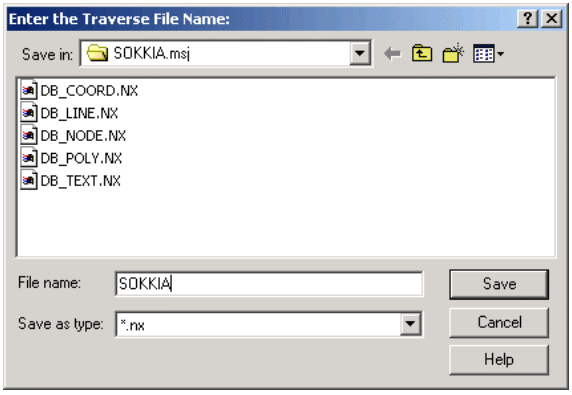
search again looking for repeated shots, doubled angles, flipped faces, etc.

You are first prompted whether you are starting a new Traverse file, or continuing with an existing Traverse file.

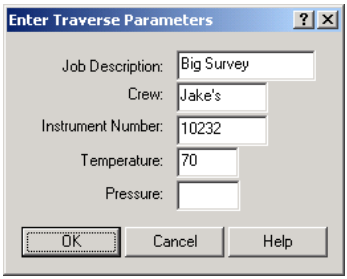


In most cases you will choose the New Traverse File option as each file you read in will need to be calculated and edited separately. If you are merging 2 collector files, or adding to a manually entered file, then you may try the Existing Traverse option.

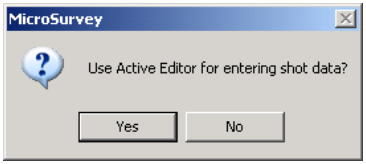
Now you need to identify the information by giving us a Traverse File Name. This allows us to go back to this set of data at any time to edit or coordinate it.



Enter a value with no extension; any extension you enter will be ignored. If the file already exists, the new information will be appended to it.

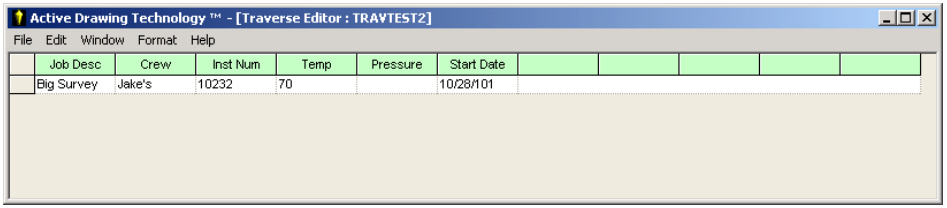


Next you will be asked to enter general information about the traverse. This information is for your use only. No calculations are performed based upon any values you enter into the Job Information Dialog.



The Active Editor prompt will appear next. This allows you to enter the data into a new spreadsheet type format. It is the preferred and fastest entry method.

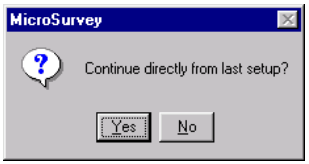
If you choose the Active Traverse Editor, then a window will appear that looks like this:



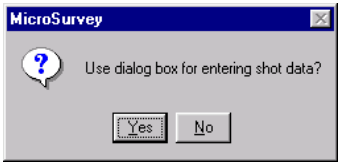
Please refer to the on-line help for more detailed information about how to use the Active Drawing Technology Editors.

If you choose not to use the Active Traverse Editor, then you will be asked the following questions:

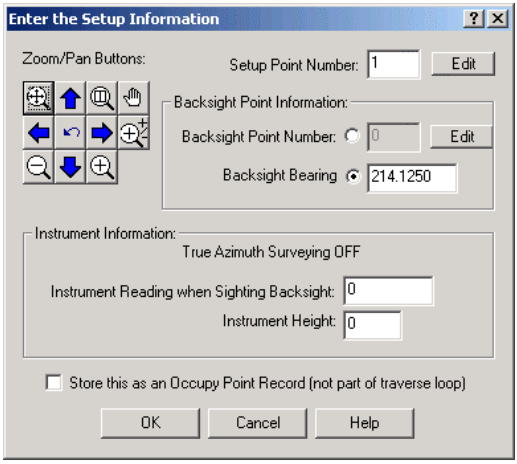
You are offered a choice of starting with the last setup in the Traverse file (if it already exists) or specifying a new one (which is most common).



Next you will be prompted whether you want to use a dialog box for entering the shot data.



To specify a setup, you enter the following information:



Setup point number: If you enter an existing point number, the setup will be placed there; if you enter a nonexistent number, you will be prompted for information to define a new point.

Edit: the edit button allows you to edit the northing, easting, elevation and description of the points without having to exit back to Cogo. This is available for both the setup and backsight points. The same dialog box appears as can be seen in the Store + Edit Points dialog box.

Backsight Point Information a radio button allows you to choose between the following two options.

Backsight Point Number: if this option is selected you simply enter the number of the backsight point. Edit options are available if the northing and easting values need to be changed. The backsight azimuth will be calculated between the setup and backsight points.

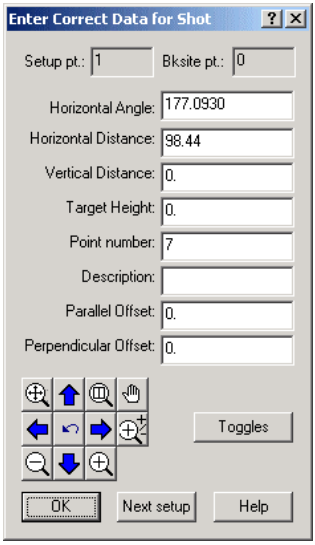
Backsight Azimuth: if this option is selected, you will be asked to input the actual azimuth direction from the setup towards the backsight. No backsight point number needs to be defined with this option.

Instrument Information

Instrument Reading When Sighting: Enter a value for the instrument reading on the backsight. This value will be subtracted from the foresight angle on each shot. This allows you to run true azimuths in the field if you wish.

Enter instrument height: This prompt will only be active if the Elevations toggle is ON. MicroSurvey always calculates the elevations of shots; however, if an elevation has not been set for the starting point, the elevations will start from zero.

The data entry for each shot will present the following dialog box:



Enter the Horizontal Angle: Enter the instrument horizontal angle reading.

Enter the Slope/Horizontal Distance: This prompt will ask for Slope or Horizontal distance according to the setting of the **Horizontal Distance/Vertical Distance** toggle. Enter the appropriate reading.

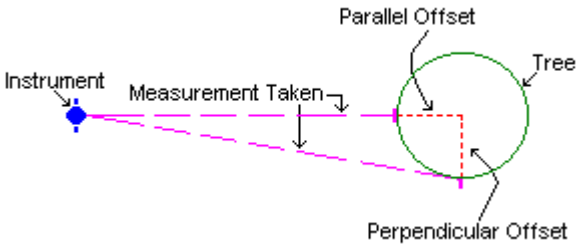
Enter the Vertical Angle/Vertical Distance: This prompt will ask for the Vertical Angle or Horizontal Distance according to the setting of the **Horizontal Distance/Vertical Distance** toggle found in the System Toggles. Enter the appropriate value.

Enter the Target Height: This prompt will appear if the **Elevations** toggle is **ON**. Enter the target height if appropriate.

Enter the Point Number: This prompt will appear if the **Auto Numbering** toggle is **OFF**. Enter a point number if appropriate.

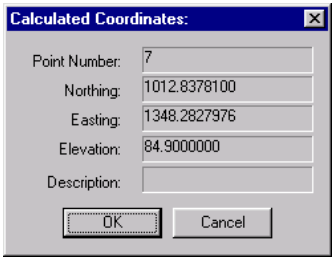
Enter the Description: This prompt will appear if the **Descriptions** toggle is **ON**. Enter a description if desired.

Parallel Offset / Distance Offset: See the sketch below if needed.



If you are entering the data without dialog boxes, then once all the shot information has been entered, you will receive a summary of it on the text screen and will be asked if you want to accept it. If you press **Esc**, MicroSurvey will return to the Horizontal Angle prompt, so that you can re-enter the values. The values you previously entered will be offered as defaults, so you will only have to retype the ones you want to change.

If the **Auto Coordinate** toggle is **ON**, MicroSurvey will now calculate coordinates for the shot and ask if you want them stored. A dialog box will show you the computed values:

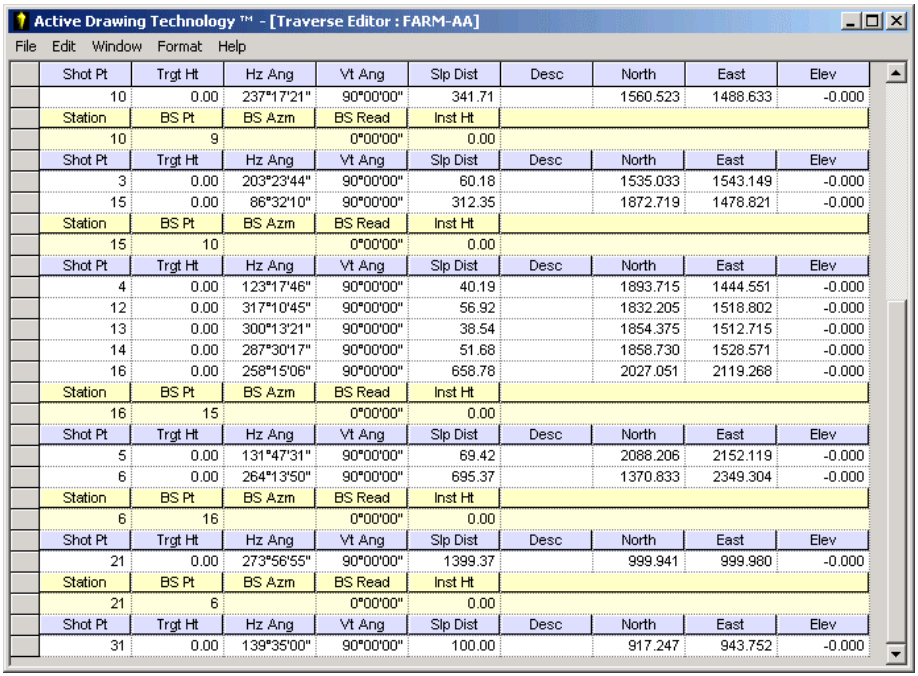


If you accept the coordinate, and the **Point Protection** toggle is **ON**, MicroSurvey will then check the database to see if the point already exists. If it does, you will receive the Point Protection warning.

The program will cycle back to the raw data entry dialog, or the **Horizontal Angle** prompt, ready for another shot. If you wish to enter a new setup, pick the **Cancel** button on the dialog, the **ESC** key or press **B** at the horizontal angle prompt.

Edit Traverse File

This option lets you edit a Traverse (raw observations) file. It will open the current traverse file selected using the Set Traverse File command. Next a window will appear with a list of all the notes, setup records and shots.

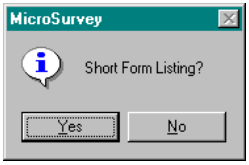


Shot Pt	Trgt Ht	Hz Ang	Vt Ang	Slp Dist	Desc	North	East	Elev
10	0.00	237°17'21"	90°00'00"	341.71		1560.523	1488.633	-0.000
Station	BS Pt	BS Azm	BS Read	Inst Ht				
10	9		0°00'00"	0.00				
Shot Pt	Trgt Ht	Hz Ang	Vt Ang	Slp Dist	Desc	North	East	Elev
3	0.00	203°23'44"	90°00'00"	60.18		1535.033	1543.149	-0.000
15	0.00	86°32'10"	90°00'00"	312.35		1872.719	1478.821	-0.000
Station	BS Pt	BS Azm	BS Read	Inst Ht				
15	10		0°00'00"	0.00				
Shot Pt	Trgt Ht	Hz Ang	Vt Ang	Slp Dist	Desc	North	East	Elev
4	0.00	123°17'46"	90°00'00"	40.19		1893.715	1444.551	-0.000
12	0.00	317°10'45"	90°00'00"	56.92		1832.205	1518.802	-0.000
13	0.00	300°13'21"	90°00'00"	38.54		1854.375	1512.715	-0.000
14	0.00	287°30'17"	90°00'00"	51.68		1858.730	1528.571	-0.000
16	0.00	258°15'06"	90°00'00"	658.78		2027.051	2119.268	-0.000
Station	BS Pt	BS Azm	BS Read	Inst Ht				
16	15		0°00'00"	0.00				
Shot Pt	Trgt Ht	Hz Ang	Vt Ang	Slp Dist	Desc	North	East	Elev
5	0.00	131°47'31"	90°00'00"	69.42		2088.206	2152.119	-0.000
6	0.00	264°13'50"	90°00'00"	695.37		1370.833	2349.304	-0.000
Station	BS Pt	BS Azm	BS Read	Inst Ht				
6	16		0°00'00"	0.00				
Shot Pt	Trgt Ht	Hz Ang	Vt Ang	Slp Dist	Desc	North	East	Elev
21	0.00	273°56'55"	90°00'00"	1399.37		999.941	999.980	-0.000
Station	BS Pt	BS Azm	BS Read	Inst Ht				
21	6		0°00'00"	0.00				
Shot Pt	Trgt Ht	Hz Ang	Vt Ang	Slp Dist	Desc	North	East	Elev
31	0.00	139°35'00"	90°00'00"	100.00		917.247	943.752	-0.000

The Active Traverse Editor is described in detail in the on-line help.

List Traverse

This option lists a measurement file to the screen command prompt area as well as into the Log file, if it is active.
It will use the Measurement file set in the Set Measurement File command.
You will be asked if you want a Short Form Listing.



The Short Form looks like this;

```
                        Traverse Print Out
================================================================================
Job Description:    Crew:   Inst:          Temp:          Press:

Setup    Backsight    BS Azimuth       BS Reading       Instrument Height
1            2       N00d00'00"E       29.4930             1.524
```

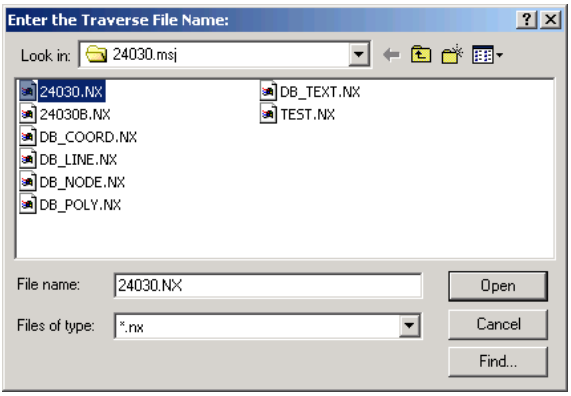
```
    1 N:   100.000 E:   100.000 El:     0.000 D:  EOP
    2 N:   252.530 E:   187.443 El:     0.000 D:  BS
Pt#   HZAngle   SlpDist  VTAng   ParOff   PerpOff TgtHt  Description
2      29.4931    84.357  91.1610   0.000    0.000    0.914    MH
Setup   Backsight   BS Azimuth      BS Reading      Instrument Height
1            2      N00d00'00"E      29.4930            1.524
    1 N:   100.000 E:   100.000 El:     0.000 D:  CL
    2 N:   252.530 E:   187.443 El:     0.000 D:  BS
Pt#   HZAngle   SlpDist  VTAng   ParOff   PerpOff TgtHt  Description
3      107.0721  32.836  90.2507   0.000    0.000    0.914    TREE
7      167.1547  46.345  88.5730   0.000    0.000    0.500    HP
8      209.4716  14.402  90.5712   0.000    0.000    0.500    BAR
9      203.5255  18.429  91.3338   0.000    0.000    0.500    BAR
```

And the Long Form looks like this:

```
================================================================
Automatic setup:
Rec Num :  74000 Rec type:   220 Point :  2215 Inst  :  0.000
BS Pt :  2214 BS dir:       0.0000 BS read:   0.0000
Status  : GOOD
Time: 07/05/97 17:08

================================================================
Foresight shot:
Rec Num :  75000 Rec type:  310 Point   :  2216
Horz Ang: 173.4349 Vert Ang:  90.0000
Distance:   129.461 Hz. Dist:   129.461 Vt dist :     0.000
T Height:  0.000Par. Off: 0.000 Perp Off: 0.000 Description: CP2
Status  : GOOD
Method: Horizontal Distance & Vertical Distance
Time: 07/05/97 17:08

================================================================
Automatic setup:
Rec Num :  76000 Rec type:   220 Point :  2216 Inst  :  0.000
BS Pt :  2215 BS dir:       0.0000 BS read:   0.0000
Status  : GOOD
     Time: 07/05/97 17:08

     ================================================================
     Foresight shot:
     Rec Num :  77000 Rec type:  310 Point   :  2217
     Horz Ang: 180.4001 Vert Ang:  90.0000
     Distance:   717.785 Hz. Dist:   717.785 Vt dist :     0.000
     T Height:  0.000Par. Off: 0.000 Perp Off: 0.000 Description: CP2
     Status  : GOOD
     Method: Horizontal Distance & Vertical Distance
     Time: 07/05/97 17:09

     ================================================================
```

Delete Traverse File

This option deletes a measurement file from the project directory. You will be prompted for a measurement file name; on entering it, you will be given the option of proceeding with removing the measurement file, or canceling the operation. The deleted measurement file is NOT placed in the recycle bin.



BE VERY CAREFULL and DO NOT ERASE any of the files named below! If you do then you will be destroying other valuable information for your job and you may have to start over.

DB_POLY.NX
DB_COORD.NX
DB_LINE.NX
DB_NODE.NX
DB_TEXT.NX

Show / Draw Traverse

This program will graphically display the observations of the current traverse file on the screen. You can "show" the observations, which will display the shot vectors until the next redraw, or you can "draw" the observations which will add lines to the drawing on the current layer. These lines will automatically be labeled according to the System Toggles settings.

New for MS2002: You now have the option of drawing only the control traverse, excluding the side shots.

Note: If the draw option is chosen and lines already exist connecting the points, new lines will not be drawn

Calculations

Re-Coordinate Traverse

This option coordinates all uncoordinated points in the current traverse file or re-coordinates the file after changes were made.

When you coordinate a traverse file, MicroSurvey will search the database for coordinates for the initial setup and backsight points. If these do not exist in the current job's database, you will be prompted with the Store and Edit Points dialog so you can enter them without exiting the coordinate traverse routine. Coordinates for subsequent points will be calculated using the observations in the traverse file. Should your traverse file contain additional traverse observations, not connected to the initial traverse, you may be prompted for starting coordinates again.

See the on-line help for more details.

Compute Traverse Closure

This option computes closure on the current traverse (two traverse types and 7 scenarios are supported – 1 closed loop and 6 open traverse). All report the error distance and direction, closure ratio, total angular error, and angular error per angle where appropriate.

See the on-line help for more details.

Angle Balance

Apply the angle correction calculated with the Compute Closure routine to the set current traverse. **Note:** Before running this command, you must have first coordinated the traverse and run the compute closure routine.

See the on-line help for more details.

Traverse Vertical Balance

The Vertical balance will simply distribute the elevation error throughout the traverse in proportion to each traverse leg's slope distance over the total traverse slope distance.

See the on-line help for more details.

Compass Balance

Use the Compass Rule to balance your traverse when the precision of your angle and distance observations is approximately equal. This is usually the case with modern theodolites and electronic distance measuring.

See the on-line help for more details.

Crandall's Method Adjustment

Crandall's Adjustment is based on the angular misclosure. It should only be used when the angular misclosure is known as in a case where the last setup is on a known point and has a closing shot to another known point.

See the on-line help for more details.

Least Squares 3D Adjustment

3D Least Squares Adjustment for traverses and networks.

See the on-line help for more details.

Existing Point Traverse

This option adjusts a string of points to make them fit between two known points.

See the on-line help for more details.

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Traverse File Utilities

Export a Traverse in TDS

Sometimes it is necessary to be able to export a traverse file and re-import it into either the same job or a different job. This can now be done via this routine. We generate a TDS formatted raw data file that can be read into any MicroSurvey job just like any other TDS RW5 file.

See the on-line help for more details.

Star Net Interface

This routine exports reduced raw observations from a MicroSurvey traverse file into a .DAT file that StarNet can read.

See the on-line help for more details..

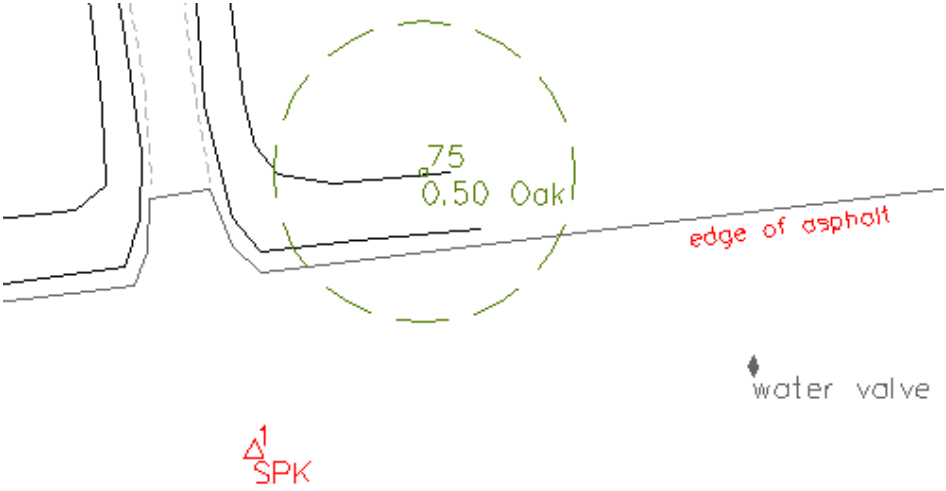
AutoMAP System

The AutoMAP System in MicroSurvey is a system of automated drafting tools that can help you to organize many tasks. It can be used to display a more detailed description in the drawing while maintaining a simple description for field coding. It can be used to automate the placing of symbols in a drawing and for setting up line connectivity based upon point description. One symbol may be used by several descriptions but customized by scale for each use. The line connectivity can be further customized with specific layers and linetypes to minimize the editing needed after processing.

Alternatively, you may use XYZ , SDR or AIMS coding for line connectivity and the AutoMAP system for symbol placement. There are also settings available to customize the layer for the display of points and their respective point numbers, descriptions and elevations.

The AutoMAP system can be used to give your drawing a very professional appearance with a minimum of effort. In the small sample that follows, all items except the text that identifies the edge of asphalt were automatically drawn based upon the AutoMAP library setup.

Sample drawing prepared using the AutoMAP System:



Have a quick look at the example above. Note that the layers DESCRIPTION, MSPOINT and POINTNUMBER have been frozen for clarity.

The field description for the tree was 50OAK but we changed the plotted description to 0.50 oak. The symbol inserted consists of only 2 circles, representing the trunk and canopy of the tree, inserted at point 75, which corresponds to an identity tag on the tree. The point number was moved to the description layer so that both items appear in the same color.

The water valve symbol is inserted on a layer called UTILS and the description has been plotted to this layer as well.

The triangular symbol at point 1 indicates a control point (steel spike). The symbol drawn was created on a layer called CONTROL and the point number and description have been moved to this layer. This is so we can display the control point number and description without having the screen filled with the point numbers of all of the other points in the drawing.

The edge of asphalt linework was created using the AutoMAP linework connections specified as 2D lines and connects 9 points that were shot in sequence with the field description EP.

As you can see, the AutoMAP System has the ability to save you a great deal of time in preparing your drawings and can help you to view only the most important point details. It is well worth the effort to set it up.

See the on-line help for more details.

Connect Points by XYZ Coding

This command causes lines to be drawn connecting points in accordance with a system of prefixes which you add to your point descriptions. By entering descriptions with this code, you can gather data in the field, transfer the data to the computer, and have lines drawn immediately.

The way that the linework appears can be controlled in AutoMAP to allow placement of the lines on selected layers with the proper line or polyline drawn between the points.

See the on-line help for more details.

SDR Connection Method

COMMAND: SMAP

The SMAP command will draw linework connecting points in accordance with a system of control coding which you include with your point descriptions. By recording field data using these control codes with your descriptions, you can have most of your linework drawn with the execution of a single command in MicroSurvey

See the on-line help for more details.

NIKON / AIMS Connection Method

Command: NMAP

The NMAP command causes linework to be drawn connecting points in accordance with a system of control coding which you include with your point descriptions. By recording field data using these control codes with your descriptions, you can have most of your linework drawn with the execution of a single command in MicroSurvey.

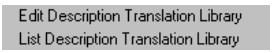
See the on-line help for more details.

Note: SDR and NIKON / AIMS linework will honor AutoMAP settings for Layer and Linetype. Lines will be drawn in 3D and polylines will be drawn in 2D. This may be expanded upon in future releases to provide additional control over 2D and 3D.

Description Translation Library

MicroSurvey maintains a translation table which lets you store frequently used numeric descriptions and converts them to alpha descriptions. If the toggle is turned on then when you read in a data collector file or an ASCII file, the numeric descriptions will automatically be converted to the equivalent alpha descriptions and saved this way in the database. Common collectors that work in this format are the Leica and Geodimeter collector formats.

The next two options let you maintain this table.



No spaces allowed. Here are a few examples of how to use numeric descriptions and translate them into alpha and be able to use our XYZ coding.

Simply add 100, 200, 300 and 400 appropriately to the sub 100 descriptions normally used day to day. If there are more than 100 normal descriptions then bump the numbers up to 500, 600, 700, and 800 or whatever is appropriate.

If 1 was a ditch
101 would be a Zditch
201 would be a Xditch
301 would be a Yditch

If 22 was a CL
122 would be a ZCL
222 would be a XCL
322 would be a YCL

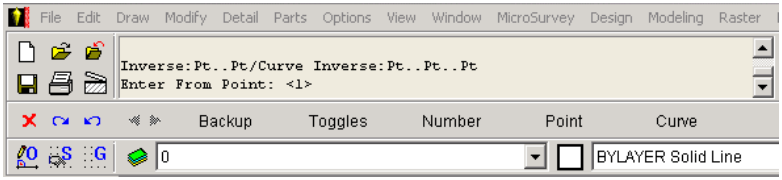
If 89 was a BLD
189 would be a ZBLD
489 would be a .BLD

Points/Lines/Curves

Calculate Points

COGO

Selecting **COGO** causes the following prompt to appear on the MicroSurvey text screen.



Data Input Options

Depending on how you answer this and subsequent prompts, you can do any of the following calculations:

New Point
Traverse
Inverse
Curve Inverse
Baseline Offset
Three Point Angle
Bearing/Bearing Intersection with Offsets
Bearing/Distance Intersection with Offsets
Distance/Distance Intersection with Offsets
Distance/Angle Intersection with Offsets

All of these commands have been designed for transparent entry of the zooming, panning, and toggle commands. You can pick any of the zoom/ pan icons at the left side of the screen in the middle of these commands. You can also pick the Toggle button below the entry window.

On completion, you are returned to the same prompt for another calculation. Pressing **Esc** will terminate the command.

Some of these calculations create new points, while others only display calculated bearings, angles and distances on the text screen. Click on green COGO commands above to see the descriptions of each type of calculation prompt by prompt.

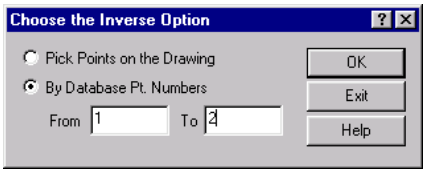
There are many options for entering data at COGO prompts to speed up your calculations.

COGO: Bearing/Azimuth Options
COGO: Distance Options
COGO: Using Offsets
COGO: Batch Recording and Playback

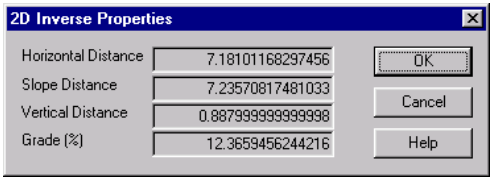
The Active Batch COGO features are described in detail in the on-line help.

2D Inverse

This command will quickly compute the horizontal distance, slope distance, vertical distance and percent grade between 2 points in your drawing. Choose two points by Picking on the Drawing (use osnap functions) or by specifying two database point numbers.



After selecting the points, you will get the following information:



Note: For bearing and distance, use the COGO Inverse function.

Radial Inverse

This program will compute radial inverses from one point to a single point, or to many points at once. The program will honor the toggles for Drawing Lines, Bearings, and Distances. Following is an example of how to use the program:

```
Enter the point to inverse from: 109
Enter Range like: Point#..Point# to inverse to:  (press <Esc> to
finish):110

  109  N27°15'29"W      30.494       5011.670       5014.444    110
Enter Range like: Point#..Point# to inverse to:  (press <Esc> to
finish):245..250

Start Point:245    End Point:250
  109  N19°02'05"E      65.069       5022.156       5025.169    245
  109  N29°47'09"E      49.599       5016.528       5026.211    246
  109  N03°21'28"E      69.973       5024.699       5019.950    247
  109  N12°07'53"E      38.895       5014.998       5021.192    248
  109  N06°05'00"E      57.132       5020.723       5020.546    249
  109  S18°26'31"W     169.839       4954.299       5002.325    250
```

```
Enter Range like: Point#..Point# to inverse to:  (press <Esc> to
finish):*** Cancel ***
```

The output is placed in the LOG file if log file recording is ON.

Point on an Object

This option sets a point on an object at a chosen distance from one end.

Proportioning

This program will proportion a line or curve in 2D or 3D with a variety of options for controlling the calculations.

See the on-line help for complete instructions.

Point by Turned Angle

This option sets a point at a specified distance and angle from a reference point.

Select either: Points/<Line>: To use a line as reference, select it with the

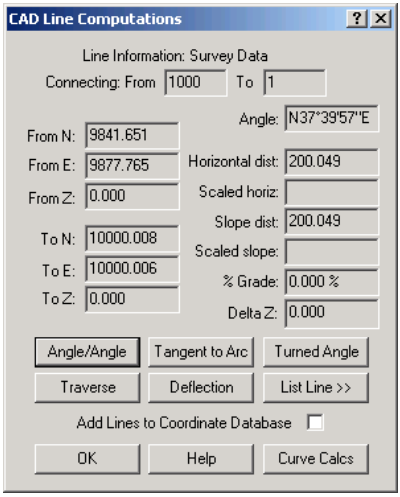
Point by Station/Offset/Elevation

This option creates a point at a station number on, and offset from, a line or curve.

Calculate Lines

CAD Lines Dialog

A feature that allows you some powerful methods of inputting line data without having to coordinate points on the ends of all the lines, as you do in COGO. If you do wish to have the new points added to the database then this option is available. It is also a powerful and easy way to list information about any line on screen. You do not have to pick from the menu or type a command to start this routine – simply pick a line, while no command is running.



See the on-line help for more details.

Right of Way Design

This program will compute lines parallel to a center alignment. It is primarily used for utility lines. If you need to design a road right of way, it is better to use the Design → Auto Route Programs.

See the on-line help for more details.

Line Tangent to Curve

This option computes a line off the end of a curve, tangent to the curve at that point.

Select the End of the Curve: Select a point on the curve. The tangent will be drawn from the end of the curve nearest the selected point.

Enter the length of the Tangent: Enter the tangent length in feet or meters according to the Units toggle.

MicroSurvey will enter the line on the drawing and return you to the starting prompt. Pressing Enter will terminate the command. If the curve is not in the coordinate or connectivity databases, it will be added to them. The line will be added to the line database, and a point will be stored at its far end if the “Add lines to Coordinate Database” checkbox is selected.

Lines by Inversing (Dot to Dot)

This routine will draw 2D lines while accessing 3D points accurately on screen.

Inversing is a "lean and mean" technique for connecting any two points whether they presently exist in the MicroSurvey database or not. It ignores the Descriptions toggle and does not prompt for point descriptions when it creates a new point.

Num/Select the starting point:(N or pick)

If you want to start at an existing MicroSurvey point, press N, followed by Enter and then enter the point number.

Press ESC to get out of Number Mode...
Enter the Point Number to connect:

If you have many point numbers to join, the routine now stays in point number entry mode until you press ESC to exit back to the previous prompt. If you enter a nonexistent point number, it will be rejected and the calculation will terminate.

Num/Select the next point (N or pick):

If you use the pick option you will be creating new points both on the screen and in the database. The point being assigned will be the next available point number. When picking points you may use any of the object snaps that may assist you. If you pick a location that is already coordinated then no new point is assigned and the original point at this location will be used.

With either method of entry, if the Draw Lines toggle is ON, a line will be drawn connecting the two points, and it will be labeled with its bearing and distance depending on the settings of the Draw Bearings and Draw Distances in the system toggles. A rubberband line will appear from the last chosen point to the present cursor location.

Inversing between points will now give you the inverse information in the command prompt area and will also keep a running total for display at the end of the routine to show the total distance inversed.

You can switch between picking locations on the screen and entering point numbers as often as you require.

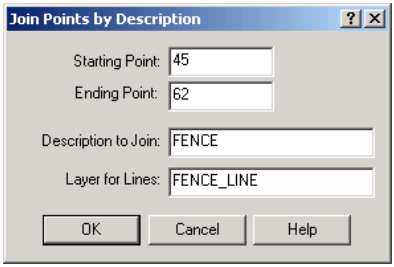
Num/Select the next point (N or pick):

An additional feature that has been added allows you to pick the button in the Options Bar that says RANGE. This Range button is available anytime the above prompt is on screen. This option allows you to enter a range of numbers to join together. 200..250 as an example, would join from 200 to 201 to 202 increasing by one each time, right up to 250. If a line is already drawn between any of the numbers then the routine simply carries on to the next point. Each line will be labeled according to how the system toggles are set (same as above).

Pressing ESC will end the section of the routine you are in and pressing ESC a second time will terminate the routine completely.

Lines: Connect Points by Description

This program will connect points with the same description by lines.



The starting point and ending point define the range of points, which will be examined for the specific description. The points will be connected in order of ascending point numbers. The lines will be placed on the layer indicated.

Best Fit a Line

This option computes a best-fit line through a series of points. It is ideal for fitting a line to points picked up along a road or fence line where the starting point is known. Use the OSNAP to pick the points as point numbers can not be used.

Select the Starting Point: Select the first point in the sequence.

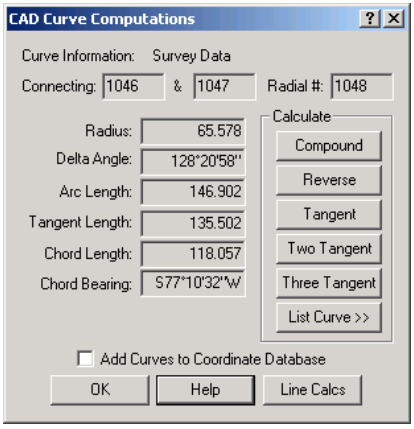
(This point will be held)
Select Next Point: Select as many points as desired. Press **Enter** to indicate completion.
(A best fit will be calculated along these points from the fixed point)

MicroSurvey will report the number of points used and enter the line on the drawing.

Calculate Curves

CAD Curves Dialog

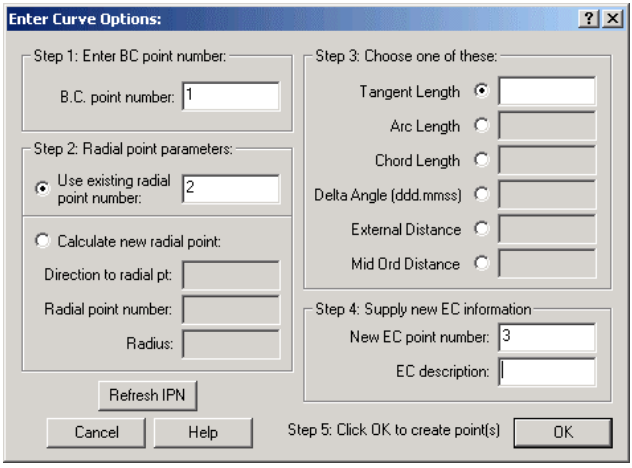
CAD Curves is a program that allows you some powerful methods of inputting curve data without having to coordinate points on the ends of all the curves, as you do in COGO. If you do wish to have the new points added to the database then this option is available. It is also a powerful and easy way to list information about any curve on screen. You do not have to pick from the menu or type a command to start this routine – simply pick a curve, while no command is running.



See the on-line help for more details.

COGO Curve Calculations

The Curve Calculations selection on the main menu extrapolates along a circular arc from a given point and creates a new point on the arc.



Start at the top left of this dialog box. Enter the start of curve point. There are two options when entering the radial point. If you have a pre-computed point, then you can enter the number, and the program will use it. If you do not have a point, you can enter the direction to radial point, radius, and point number. The program will compute a radial point.

You next have a series of options on how to compute the end of curve point. Choose one of Tangent, Arc, Chord Length, Delta Angle, External Distance or Mid Ordinate Distance.

Enter the point number to compute, and description for the point.

BC = Beginning of Curve
EC = End of Curve

Best Fit a Curve

This option fits a curve to a sequence of points. It is useful for fitting to points picked up in the field on a street curve or property fence.

The curve fit is always done in a ***counterclockwise*** direction, and you must enter the beginning and ending points in that order. Other points

may be entered in any order. For the most accurate curve fit, use the MicroSurvey OSNAP functions to select points as point numbers can not be used.

Enter the Beginning of the Curve: Select the point at the clockwise end of the curve.

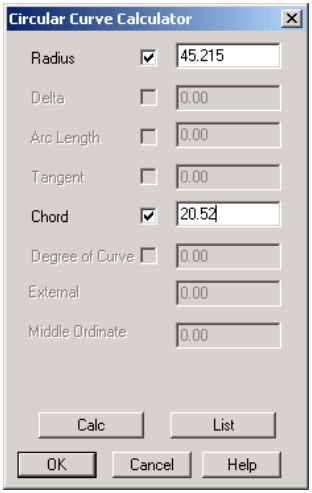
Enter the End of Curve Point: Select the point at the counterclockwise end of the curve.

Select a Point: Select as many points as desired. Press **Enter** to indicate completion.

MicroSurvey will compute the curve and enter it on the drawing. You will be returned to the beginning point prompt; pressing Enter will terminate the command.

Curve Problem Solver

This option computes all the characteristics of a curve based on values you enter for any two, as long as these are sufficient to define the curve. To fully define the curve, one of the values you enter must be either the radius or delta angle. Also, the radius and degree of curve are not sufficient. Pick the boxes on the left to turn the prompts on and off.



Pressing the **Calc** button computes the characteristics.

Pressing the **List** button allows you to pick a curve graphically and display its characteristics.

Curve Off One Tangent

This option computes a curve off the end of a tangent. You define the curve by a radius and either a tangent distance, chord length, delta angle or arc length.

Select the Tangent: Select the line off which the curve will be drawn. The end of the line nearest to the selection point will be used.

Enter the Radius (Positive=CW, Negative=CCW): Enter a value for the radius. A positive value will give a curve moving clockwise off the tangent; a negative value will give a counterclockwise curve.

Choose a method: Tangent/Chord/Delta angle/<Length>: If you want to use the default of arc length, press Enter. You will be prompted to enter a value. Otherwise enter T, C or D and press Enter. Then enter a value for the tangent, chord length or delta angle at the subsequent prompt.

MicroSurvey will compute the curve and enter it on the drawing. You will be returned to the tangent prompt, and pressing Enter will terminate the command.

Curve On Two Tangents

This option computes a curve tangent to two lines with a specified radius.

Select the first tangent: Select a line.
Select the second tangent: Select a line.
Enter the radius: Enter a value.

MicroSurvey will compute a curve of the given radius and tangent to the selected lines, and enter it on the drawing. The tangent lines will not be removed.

Reverse or Compound Curve

This option computes a curve on the end of an existing curve, mutually tangent at the intersection.

Select the Curve: Select the curve on which the new curve is to be drawn. The end nearest the selection point will be used.

Choose the Type: Compound/<Reverse>: Choose whether the new curve will be compound or reverse with respect to the existing curve.

Enter the Proposed Radius: Enter a value for the radius.

Choose the method: Tangent/Chord/Delta/<Length>: If you want to define the curve by arc length (the default), just enter a value. Otherwise answer T, C or D and give a value at the subsequent prompt.

MicroSurvey will compute the curve and enter it on the drawing, and you will be returned to the curve selection prompt. Pressing Enter will terminate the command.

Coordinate Point Utilities

Graphical Coordinate Editor

Overview

The Graphic Coordinate Editor provides you with a powerful tool to manipulate the data in your drawing; however, this once powerful tool has been overshadowed by the incredibly easy to use Active Coordinate Editor. We have left the Graphical Coordinate Editor in the program for legacy reasons, but we suggest very strongly that you investigate the utilities in the Active Coordinate Editor.

With the Graphical Coordinate Editor you can pick points, by many different methods, and then change many different options for those points. This combines all of our previous editing methods, and adds many more, into one simple, yet powerful routine.

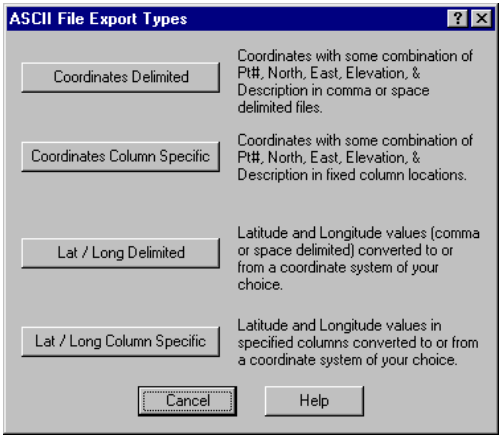
To accomplish this we are using the Group command. If you wish to read more on the Group command then please check the online help files.

A group can be thought of like a basket. You keep adding points to a group, or adding them to a basket. You can choose to add points or remove points from this basket, as many times as needed until you get the group of points that you wish to work with. Once you have the basket or group of points that you wish to work with, you can then tell us what to do with those points. All the points in the group or basket will be treated the same way. You can modify or work with the points in many different ways, as a group being treated as one object.

Note: if you are reading the manuals, please go to the on-line help system for a complete explanation of how to use the Graphic Editor.

ASCII Export of Points

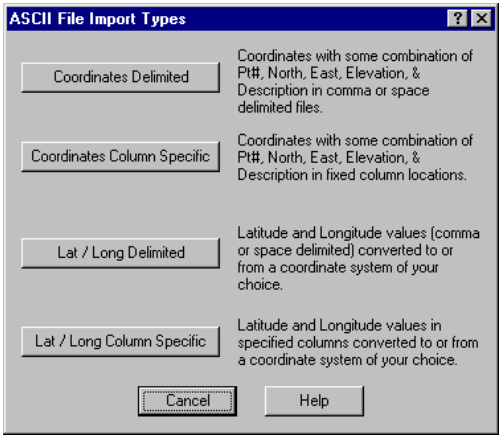
This routine has been enhanced with many new options. The four different buttons will take you to four different methods of defining the type of ASCII file you are exporting to. Each of these options has the ability to apply transformations to the coordinates and export them into a file of points for the new zone.



Note: These are hot links in the on-line help. See the on line help for more information on these functions.

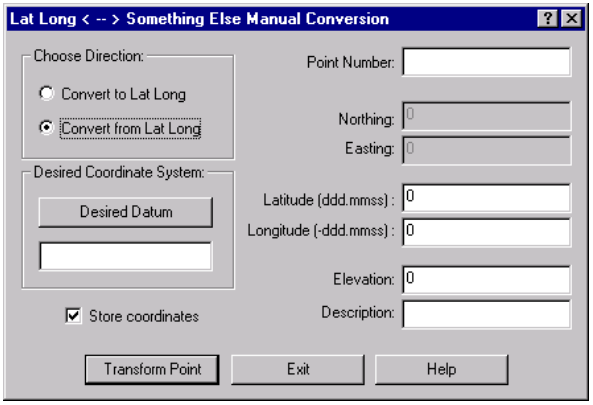
ASCII Import of Points

This routine has been enhanced with many new options. The two different buttons will take you to two different methods of defining the type of ASCII file you are importing from.
Each of these options has the ability to apply transformations to the coordinates and then import them into your job.



Note: These are hot links in the on-line help. See the help for more information on these functions.

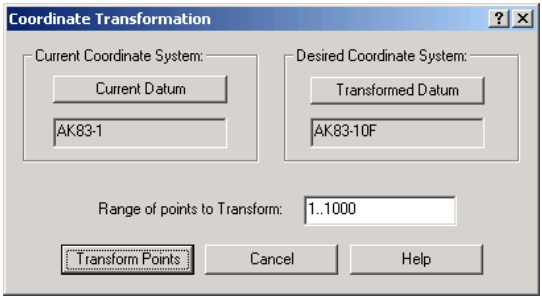
Manual Coordinate Conversions



This allows you to enter in Latitude and Longitude information and have it converted to coordinates and saved in the drawing. You can also do a conversion to the points to translate them to a particular datum.

You can also enter in coordinates one at a time and have them converted to Latitude and Longitude values.

Transform Database



This allows you to convert a range of points in your drawing from one datum to another datum. Simple yet powerful!

NOTE: Canadian clients will have to obtain the National Transformation Database from Geomatics Canada to be able to convert NAD27 to NAD83 or NAD83 to NAD27. The database file (NTV2_0.GSB) would then be copied into the \MSCAD2002\MSCAD\MAPPING\DATA directory. We will automatically recognize the fact the files exist and use them as required.

For more information on the database, their web address is:

http://www.geod.nrcan.gc.ca/products/html-public/GSDapps/English/NTv2_Fact_Sheet.html

Auto Add Points to Objects

This option adds Points to a selection set of objects in your drawing. You are prompted for the objects to label; select as many as desired with the standard MicroSurvey object selection methods (windows, crossing, etc.).

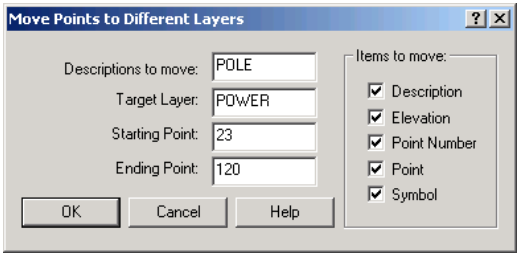
All ends of Lines, Arcs and Points will be added to the coordinate database. The objects will be numbered starting with the "Next Point

Number" and annotated with Bearings and Distances, as configured in either the System or Hot Toggles Dialog box.

Note: If you need to place points at the vertices of a Polyline then you will have to explode it first so it becomes regular Line segments.

Change Point and Object Layer

This program will move points and their components to new layers based on their descriptions.



The program prompts for a Description – this is the description that will be searched for through the database, a Target Layer – this is the layer to move the chosen items on the right side of the dialog box to, a starting and ending point to determine the range of points to be considered. You then may choose any combination of the items on the right to move to the target layer as described above.

Compute Stakes

MicroSurvey will compute stakes in 5 different ways:

- Single Offset
- Double Offset
- Property Line Intersection
- Arc Staking
- Multi-point Line Staking

See the on-line help for more details.

Delete Points

This option deletes a specified point or range of points from the database. Enter a point number, or a range of numbers in Point#..Point# form at the prompt. If you enter a single point number, it will be deleted; if you enter a range, you will be asked to verify that it is to be deleted.

Bear in mind that since lines connecting the points will have no meaning if the points are deleted, the lines and all related bearings, distances, point numbers, elevations, descriptions and symbols will disappear too. Use this option with care!

Example output:
Enter the point, or range you wish to delete: 2..33

Delete Points from 2 to 33? (Y/N) y

Identify North/East of Point

This option reports the Northing and Easting of any point in your drawing on the text screen.

Select the Point: Select any point using any of the MicroSurvey selection methods which will accurately identify the point. The Northing and Easting will be listed to the screen. The point selection prompt will then be repeated, to list more points, until you terminate the command by pressing Enter.

Example output:
Select the Point: (pick a point on screen with your mouse)

Northing = 205.637 Easting = 890.598

Helmert's Similarity Coordinate Transformation

The 2D Helmert's Transformation included in MicroSurvey is a powerful tool that can be used for a variety of coordinate transformations. With the Helmert's Transformation, you are able to rotate, shift and scale up to 5000 points based upon a best fit solution using 2 to 500 pairs of common coordinate points, all with the execution of a single command!

The program assumes the following:

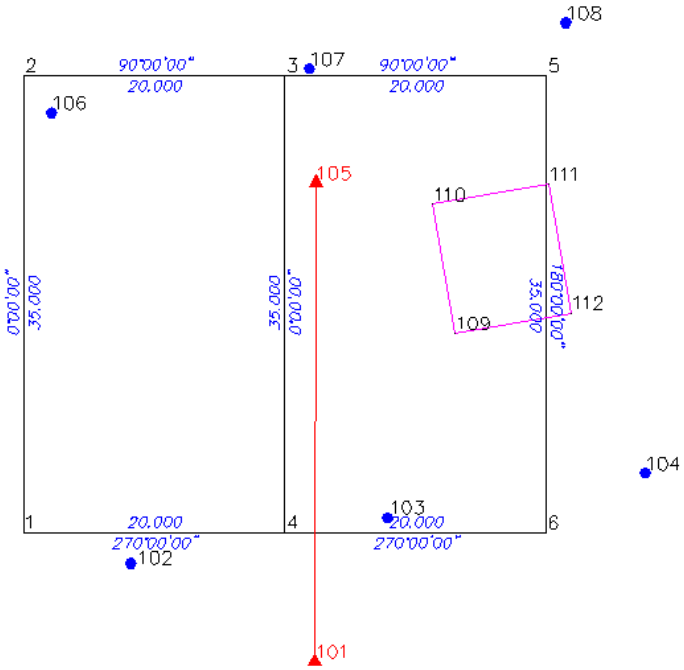
1) The coordinates of the “Plan system are considered fixed and error free.

2) Equal weighting is applied to all “Local coordinates used to calculate the solution

A minimum of two points common in both the Plan and Local systems are needed to be able to solve the four unknown transformation parameters. This gives what is known as a unique solution (only one possible set of transformation parameters can be solved for). When more than two points are common in each system, redundancy exists and a Least Squares method is used to determine the optimum (best-fit) transformation parameters.

We’ll work through an example as we explain the function of the Helmert’s Transformation routine.

In the example below, you’ll see a plan of 2 simple lots with lot corner point numbers 1 through 6. These will be our Plan (fixed) points. You will also see our field ties (Local points) for all six corners and 4 points on a simple building. Points 101 and 105 are traverse points, points 102-104 and 106-108 are corner ties and 109-112 represents ties to the building.



The objective here is to transform our local field survey into the plan system and, at the same time, check to see if the corner posts tied are located where the previous surveyors plan says they are!

See the on-line help for the complete worked through example.

Label North/East of Point

This option labels any point in your drawing with its Northing and Easting. The routine uses the current text style in the drawing. If no text style is set as current then the labels will automatically use the Default Leroy Text Size from the Main Jobs Defaults dialog box. You should set the correct text style that you wish to use, prior to running this command.

Select the Next Point: Select a point to be labeled, with any of the selection methods, which will accurately identify the point.

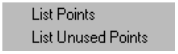
Select the Labeling Point: Select the point at which the label is to begin.

Pick the Rotation Angle: Enter a rotation angle or pick it with the cursor. The coordinates will be entered on the drawing in a stacked format,

Northing over Easting. The point selection prompt will be repeated until you terminate the command by pressing Enter.

Label Example:

List Points (Used and Unused)



Choose between listing points in the database or unused points. Note: These are hot links in the on-line help. See the help for more information on these functions.

Protected Points List

The Protected Points function found under the Coordinate Point Utilities submenu. If you set a point(s) as protected, the coordinates will be held as fixed regardless of whether or not Point Protection is toggled on and you will not be permitted to change the point description. Protected points cannot be deleted or changed in any way with CAD commands either. The Active Coordinate Editor has a column that allows you to set or remove points from the Protected Point List.

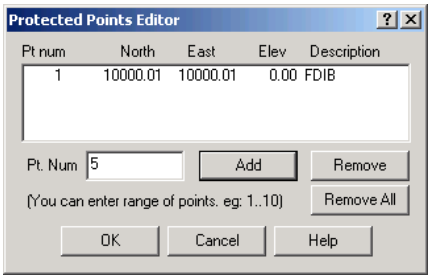
This feature is designed to be used as a second level of point protection to avoid accidental movement of control points when coordinating traverse data but it will also protect points from accidental adjustment by CAD commands. **Note:** You may see points move during execution of a CAD command but they will return to the protected location upon completion of the function.

For example: Say you have a 10 station closed loop traverse that you have balanced and adjusted. The next day you are asked to send the field crew out to do a topographic survey of the same site. You have the control in place so your crew will do the topo from these control points. When the data is downloaded into MicroSurvey and you want to coordinate the shots, there may be check shots to your control points. To avoid accidental re-coordination of your control stations, add them to the Protected Points List prior to coordinating the traverse. Upon

coordinating the traverse, if MicroSurvey encounters a shot taken to a protected point, the following message will appear in the command line and be written to the log file:

```
4406 is a protected point. This point will not be moved.
```

Add or remove points using the following dialog box.



Add Points: Enter the point or range of points to add, press the ADD button. OK to finish.
Remove Points: Select/highlight in the list, the points you wish to remove, Click the REMOVE button. OK to finish.

Renumber Points

This option renumbers a range of points beginning with a specified point number. It automatically updates the connectivity database to retain information on which points are connected.

See the on-line help for more details.

Rotate Points

This option rotates a range of points around a specified point.

See the on-line help for more details.

Scale Points

This option performs a 3D scaling operation about a specified point. You are prompted for a base point, scale factors for Northing/Easting and elevation; and a range of points to scale.

See the on-line help for more details.

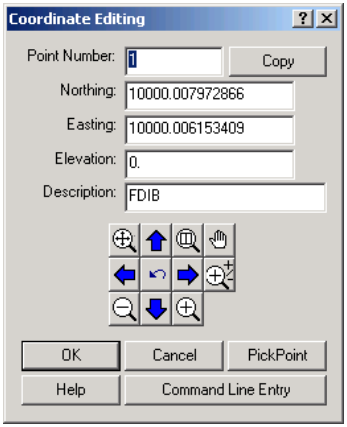
Shift Points

This option shifts a specified range of points through specified increments in Northing, Easting and Elevation.

See the on-line help for more details.

Store and Edit Points

This option lets you enter points in the database and edit existing points. When editing a point, simply input the point number in the dialog box and press the TAB key to see the remaining fields show the correct data for that point.



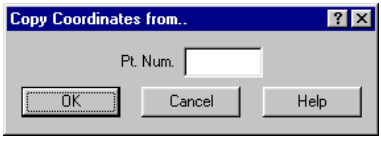
Note: Do not to use the ENTER key to try and move between the fields in the dialog box. ENTER is the same as pressing the OK button and is used to complete the edit function. Be sure to either use your Mouse to pick the fields to change, or use the TAB key to move between the fields.

You can optionally pick a point on the screen (using the pick point option) to establish the Northing, Easting, and Elevation of the point. When picking a point visually on screen you may also make use of the Object Snaps.

Command Line Entry option: If you prefer, you can pick the Command Line Entry button from the Store and Edit points dialog. The command sequence will be like the following:

```
Enter the point number to Edit:7
Enter the Northing: <39.008>:39.000
Enter the Easting: <-0.941>:-1.000
Enter the Elevation: <0.000>:
Enter the Description: <>:sample

Point Number: 7 North: 39.000  East : -1.000  Elev : 0.000  Desc
: sample
Is this Ok (Y):y
    7 N:    39.000 E:    -1.000 El:     0.000 D:sample
```

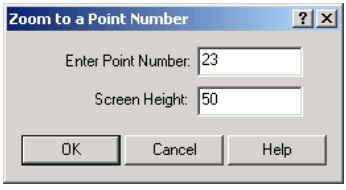
New for MicroSurvey: You now have the option of copying the coordinates of another point into the N, E, and Elevation fields. You may find this useful when computing a traverse closure.



HOT TIP - If you wish to edit an existing point, and you do not have any command active, simply pick on screen the point, point number, description or elevation of an existing point. This will activate the Active Coordinate Editor that allows you to edit anything related to the point. At the same time the point on screen will now be highlighted and a large X will be placed at the point to help show you if you have selected the correct point to edit.

Zoom to a Point

This option will find and zoom to a point in the coordinate database.



You can enter a screen height in ground units to narrow down the focus of the zoom. For example, entering a screen height of 50 would zoom centered on the point such that the vertical screen height would equal

50 ground units. This can be very handy if you get a point that is very far away from the center of the job and you have done a Zoom Extents. This would result in 2 small specks on the screen and you don't know where anything is. Running this command gives you direct control over where you zoom to and how much ground you see on screen. This is also useful when you have a large number of points in a small area and you need to locate a specific point – zoom in close and it will be in the center of the screen, and easy to find.

New for MicroSurvey: This function will now remember the last screen height used, saving you the step of entering it every time the function is used. The function has been added to the default Option Bar. Look for the ZOOM Point button.

GeoConverter Datum Editor

Overview

Coordinate System Editor Overview

Conversion of coordinates from one coordinate system to another is a three step process, where the second step can have several 'sub-steps' to it. While it might seem strange at first, we'll first describe basic steps One and Three first, deferring Step Two. After we've seen what Step One and Step Three do, we'll explore how Step Two links One and Two together to form a complete coordinate conversion system. How users provide these elements the information necessary to perform their functions is then described, followed by a description of a little programming trick used to enable this basic process to efficiently and effectively deal with geographic coordinates (i.e. latitude and longitude). Finally, we wrap this overview up with a description of the naming convention the programmers used to name the thousand or so coordinate system definitions provided with the coordinate conversion system.

See the on-line help for more details.

Lot Closure / Legal Description

New Compute Lot Closure (Map Check) function

This program is new for MicroSurvey and is designed to be more flexible and easy to use. It uses the same functions as Creating a Lot Manually, saving you the need to learn 2 separate routines. It now has the option of drawing the lot(s) as you enter data and provides the ability to back up if you make a mistake.

At startup, enter a lot number and starting location.

Note: If you wish to do a map check in an existing job, we recommend you do not draw lines or store coordinates. The program will give you a closure based on the data you enter.

See the on-line help for more details.

Legal Description Writer

This program will build a legal description by entering a series of point numbers that define the perimeter of the lot. Curves are entered by prefixing the radial point number with the + sign if the curve is clockwise and - if the curve is counterclockwise.

See the on-line help for more details.

Subdivision / House Design

Lots & Blocks

Overview of Lots & Blocks

Lots & Blocks are treated identically in our system, the only difference is that a Block can contain Lots. A Lot can contain a Building but a block can not. We will discuss creating a Lot. Block prompts are virtually identical.

Here is our definition of a Block*:*

A multi-sided polygon made up of lines and curves that may hold lots.

Here is our definition of a Lot:

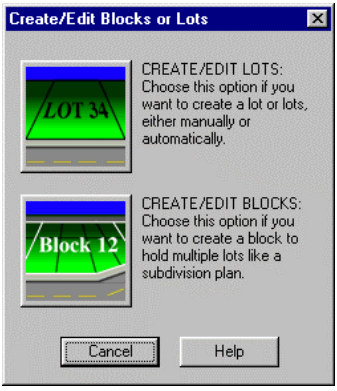
A multi-sided polygon made of lines and curves.

Lots can be created in 3 different ways:

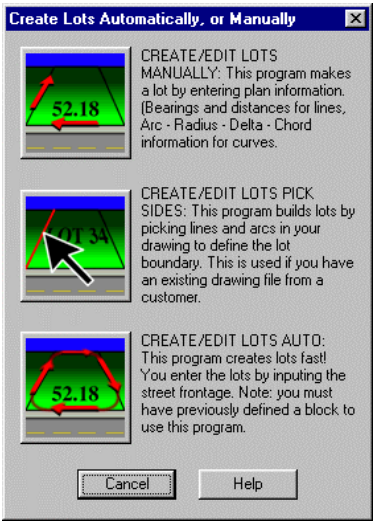
1) By manual entry – you have a plan that is showing lines and arcs with bearings, distances, and curve information.
2) By picking lines and arcs from an existing drawing – you have a drawing with lines and arcs that define the lot.
3) By subdividing a block using our Auto-Lot creation program – you have entered a block and wish to split it into lots.

Create/Edit Lots Manually

Choose CREATE / EDIT LOTS: from the dialog:



Choose **CREATE /EDIT LOTS MANUALLY:**

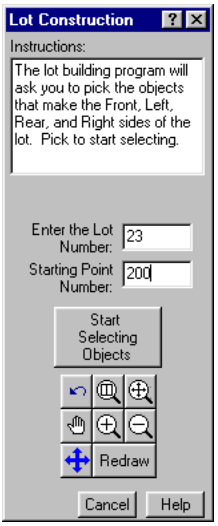


See the on-line help for more details.

Create/Edit Lots - Pick Sides

MicroSurvey can create lots from existing drawings.

1) Pick the Create / Edit Lots & Blocks button on the MicroSurvey -> Subdivision / House design submenu.
2) Pick CREATE/EDIT LOTS:
3) Pick CREATE/EDIT LOTS PICK SIDES:
4) You will see the following dialog:



1) Enter the Lot number and the starting point number. The program will start numbering the lot line and curve ends at this number.
2) Pick Start Selecting Objects.

See the on-line help for more details.

Create/Edit Lots - Automated

The Auto Lot program is designed to be used under a special case where you have many lots to enter all along a street. This program will make the data entry faster than manually entering the bearings and distances that define the perimeter of each lot.

See the on-line help for more details.

Editing Lots and Blocks

Any Lot or Block that was entered manually (using “raw data: bearings, azimuths, distances, curve data etc.) can be edited later. The raw data is stored in a separate file in the project. When you enter the Lot or Block Manual entry program, the Starting Information dialog will list the existing Lots and Blocks that were entered as raw data. You can select one to edit the data. MicroSurvey will open the entry screen with the first segment loaded in the dialog. You can move forward through the data with the Next button and move backwards with the Prev button.

When you are finished editing the data, press the Finish button to end the entry. A closure and error report will be generated. You will then be asked to pick the sides that define the front, left, rear, and right sides of the Lot or Block.

Generate Block Report

This option will generate a report of the block and write it to the Log File as shown below. To start the report, pick a line on the perimeter of the block that is not also part of another block.

Block : PlanABlockC

FromPNT	Bearing	Distance	Northing	Easting	ToPnt
100	260.4840	52.996	160.241	259.394	101
101	289.4046	93.131	151.778	207.079	102
102	246.2319	52.202	183.141	119.387	103
103	284.4636	64.411	162.232	71.556	104
104	218.3720	72.639	178.660	9.275	105
105	298.5421	64.883	121.909	-36.065	106
106	29.2122	69.111	153.272	-92.865	107
107	64.4806	94.711	213.508	-58.985	108
108	113.0134	72.545	253.831	26.714	109
109	75.2138	78.789	225.456	93.479	110
110	98.3217	97.239	245.368	169.710	111
111	185.1407	70.987	230.932	265.872	100

Square Meters: 25467.07 Hectares : 2.547
Square Feet : 274125.2 Acres : 6.293
TOTAL Traverse Distance : 883.644
TOTAL Traverse Perimeter: 883.644
TOTAL Traverse Stations : 12
ERROR of Closure : 1:INFINITY
Frontage: 335.378888.

Generate Lot Report

This option will report the inverses, curves, area, perimeter, and frontage of a defined lot. To choose the lot or lots to report, pick a line or curve on the perimeter of the lot(s).

The report will be written to the log file like this:

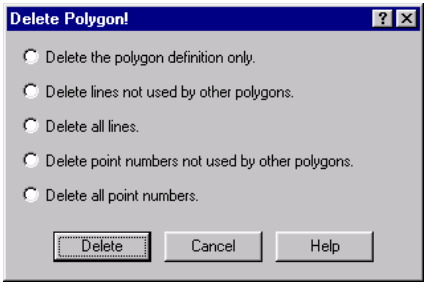
```
Lot  : 1

From PNT Bearing     Distance     Northing     Easting       To
Pnt
    9     270°00'00"   21.000       20.000       31.000       1
    1       0°00'00"   35.000       20.000       10.000       2
    2      90°00'00"   18.062       55.000       10.000      10
   10     175°12'07"   35.123       55.000       28.062       9
   Square Metres:    683.59  Hectares     :     0.068
    Square Feet  :    7358.1   Acres        :     0.169
        TOTAL Traverse Distance :    109.185
        TOTAL Traverse Perimeter:    109.185
        TOTAL Traverse Stations :      4
        ERROR of Closure        :  1:INFINITY
    Frontage: 21.000000.
```

Delete Block

This command is used for deleting Blocks and has several options regarding what exactly you wish to delete. In all cases, the block definition is deleted but you get to specify which drawing items you wish to delete. You have to be careful which you choose because it is easy to destroy the integrity of neighboring lot or block definitions. **Note:** There is no undo for this function.

Select the block you wish to delete from the list in the first dialog you see:

Now select the block components you wish to delete:

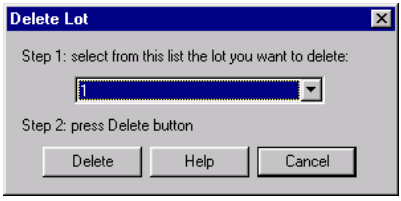


See the on-line help for more details.

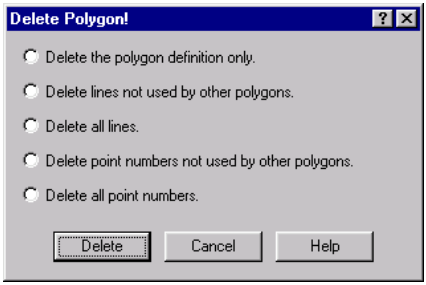
Delete Lot

This command is used for deleting Lots and has several options regarding what exactly you wish to delete. In all cases, the lot definition is deleted but you get to specify which drawing items you wish to delete. You have to be careful which you choose because it is easy to destroy the integrity of neighboring lot or block definitions.
Note: There is no undo for this function.

Select the lot you wish to delete from the list in the first dialog you see:



Now select the lot components you wish to delete:



See the on-line help for more details.

Label Lot Area and Number

This will place the lot number and area on the lots that you pick. You can pick any line on a lot and it will be labeled. The items placed on the plan are controlled by the Area Defaults dialog box.

Houses & Buildings

Overview - Create/Edit Buildings

MicroSurvey contains a Building Library. You can add, edit, delete, copy, and rename your buildings. Reference information can be stored about your buildings. There is no limit to the number of buildings that can be stored.

Buildings are available to all Projects, because buildings are “global. MicroSurvey stores a drawing preview of each building. The drawing preview is available for buildings that are create by entering “raw data. We define raw data as bearings, azimuths, distances, and curve information.
See the on line help and movies for a full description of how to use this feature.

Overview - Building Placement

Buildings are placed in, what is typically, a three-step process:

Step 1. Choose the Building from the Library. Choose the starting point number. Set the Building elevation. Choose the Lot to locate the building.
Step 2. Drag the Building Pad in the Lot until it is approximately in the correct position. Pick with left mouse button.
Step 3. Use the fine adjustment capability to locate the building exactly where you need it.

There are also 2 special cases that require attention:

Case 1 Two Points Given on the Building
Case 2 Two Points Given that are Perpendicular at some distance

Tip: If you have difficulty, try setting your direction setting to Azimuths for this function.

See the on-line help for more details.

Generate House Report

This option will generate a report listing the coordinates of the house and the area of the house.

Example:

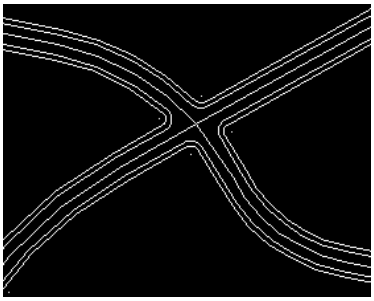
```
House: new

From PNT Bearing     Distance     Northing     Easting       To
Pnt
   34   250.3424        5.222       32.407      39.853       35
   35   295.3424        2.350       30.670      34.928       36
   36   340.3424        1.582       31.685      32.808       37
   37   250.3424        1.503       33.177      32.282       38
   38   160.3424        1.424       32.677      30.864       39
   39   250.3424        4.668       31.334      31.338       40
   40   340.3424        5.618       29.781      26.935       41
   41    70.3424        8.545       35.079      25.067       42
   42   160.3424        3.244       37.921      33.126       43
   43    70.3424        4.510       34.862      34.205       44
   44   160.3424        4.194       36.362      38.458       34
    Square Meters:     67.72    Hectares     :     0.007
    Square Feet  :     728.9    Acres        :     0.017
        TOTAL Traverse Distance :     42.862
        TOTAL Traverse Perimeter:     42.862
        TOTAL Traverse Stations :      11
        ERROR of Closure        :  1:INFINITY
```

Streets and Intersections

Create Streets from Centerline

Streets can be built up from lines and arcs (no polylines permitted). The design is done from the centerline of the street. The program will compute all the offset lines automatically. Intersections must be initially drawn to overlap completely, and they will be automatically trimmed.

Here is an example of a street centerline and the completed street network:



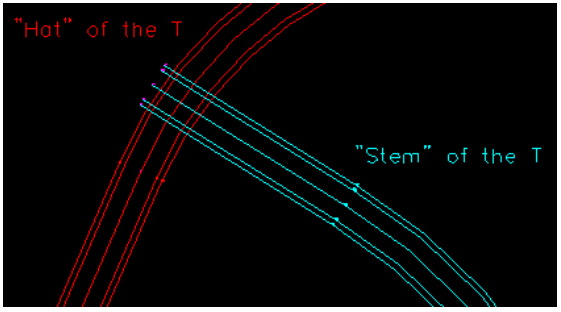
See the on-line help for more details.

Edit Streets

See the on-line help for more details.

T Intersection

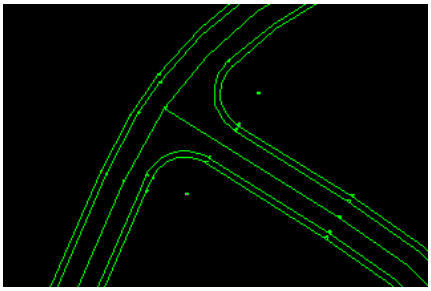
This intersection program will ask for the radii of the two internal corners. The first prompt in the program is to pick the street that forms the "stem" of the T. You can pick any line or curve on the street that will terminate at the T intersection. Here is an example of two streets ready for a T intersection:



The second prompt will ask you to pick the street that forms the "hat" of the T. Pick any line or curve on the street that does not terminate.

If the two streets intersect in more than one place, the program will prompt you for the intersection you wish to trim.

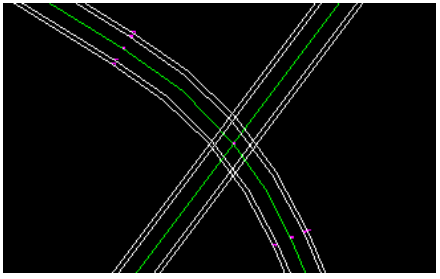
The program will prompt you for the end of the street to cut off. Here is the finished product.



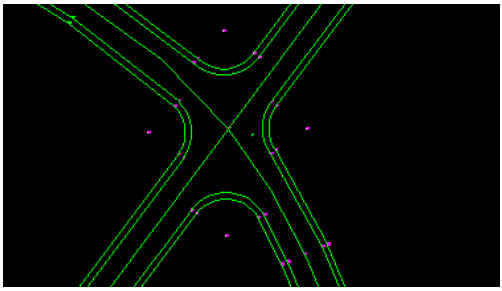
The intersection programs perform literally hundreds of calculations. Intersections can consist of either line or arc segments for the joining streets. You are prompted for the radius of each corner of the intersection. The radius is always from the radial point to the centerline of the street.

X Intersection

This intersection program will ask for the radii of all four corners. If the two roads that are intersecting cross at more than one point, then the program will prompt you for the intersection you wish to work with. The program prompts you to pick one line or arc from each street. Here is an example of how the two crossing streets should appear before the X intersection program is run:



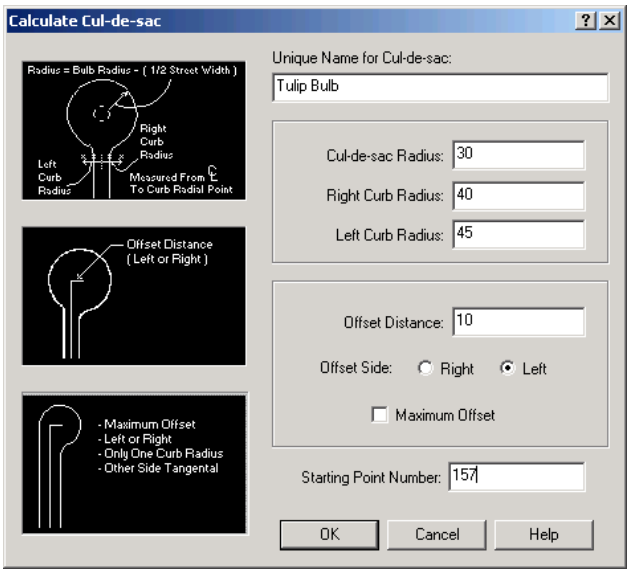
After the program has been run, the intersection will look like this:



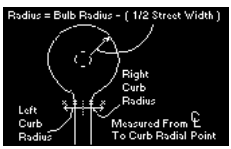
The intersection programs perform literally hundreds of calculations. Intersections can consist of either line or arc segments for the joining streets. You are prompted for the radius of each corner of the intersection. The radius is always measured from the radial point to the centerline of the street.

Cul-de-sacs

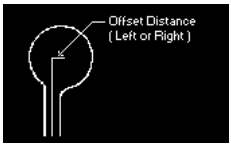
MicroSurvey will compute the three standard types of cul-de-sacs: Normal, Offset, or Right/Left. The program supplies a dialog box for the input of the measurements of the cul-de-sac. A Cul-de-sac can only be placed on the end of a Street. A normal line, arc or polyline, not defined as a Street, will not work.



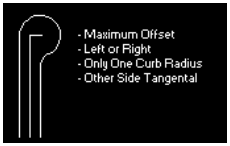
On this dialog box, there are only a few items that must be entered. The **Cul-de-sac radius** is measured from a distance starting at ½ the road width to the edge of the Cul-de-sac Bulb. The **Right and Left curb radii** are measured from the radial point to the centerline of the street.



The offset option is the amount the bulb is shifted right or left from the centerline.



If you select the **Maximum Offset** toggle, then the offset distance will be computed automatically.



When you pick the **OK** button, you will be prompted to pick a line or curve from the street you want the Cul-de-sac placed on. When you have selected a piece of the street, press **Enter**. The program will prompt which end of the street you want to calculate the cul-de-sac. You can answer Y or N to the prompt. The program will cycle until you say **Y**. The Cul-de-sac will be computed.

Edit Cul-de-sacs

This option allows you to edit the cul-de-sac points. This is a very risky thing to do, because if you modify the cul-de-sac points, they may no longer define curves. This would be invalid. However, the Print, Show, and Delete options would have obvious uses.

Area Calculations

Fast Area by Selecting Objects

This program allows you to quickly calculate and label areas on lots that are made up of individual lines and curves. This program will not work if the lot has been posted at the corners. The lines and curves that define the lot must meet or cross precisely at their end points for this function to calculate correctly

When asked to Select Objects using Window or Crossing: you can select the lines and curves that make up the lot using any normal selection method. This method does NOT use point numbers to define the shape of the area.

See the on-line help for more details.

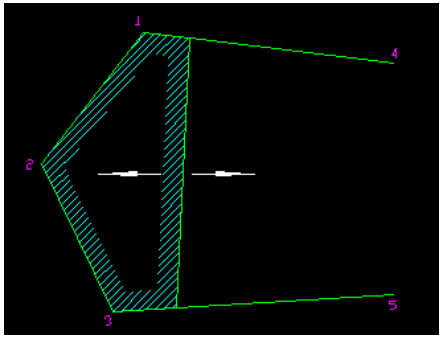
Calculate Area by Number

This option calculates the area of a lot whose boundary is any combination of lines and arcs. The segments are defined by entering MicroSurvey point numbers for the endpoints, and for the centers of the arcs.

See the on-line help for more details.

Sliding Bearing

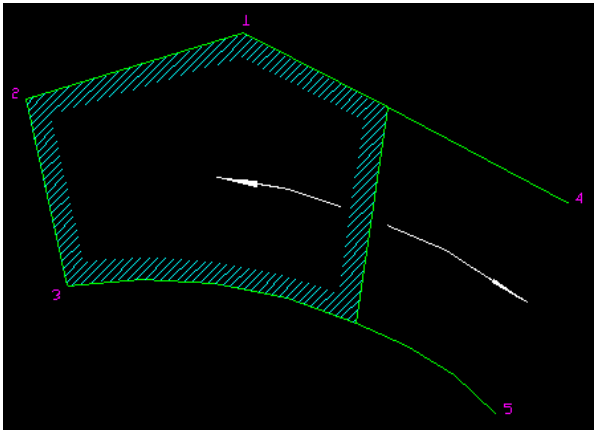
This option designs a lot to a specified area by sliding a line of fixed bearing. You select points that define the fixed sides of the lot; lot line directions from the endpoints; and the bearing of the sliding line.



See the on-line help for more details..

Radial Lot Line

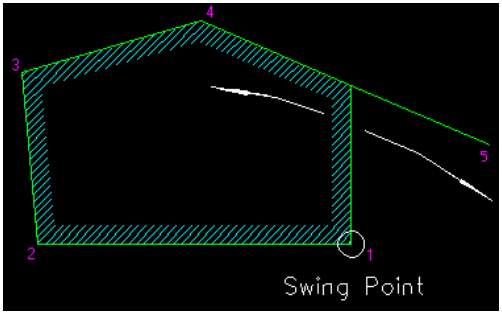
This option designs a fixed-area lot by positioning a radial line from a bounding curve. You define the fixed sides of the lot, including one arc, by selecting their endpoints; and you specify the direction of the variable- length side that will be terminated by the radial line.



See the on-line help for more details.

Swing Into a Line

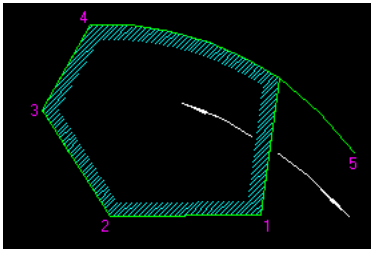
This option designs a fixed-area lot by swinging a line from a point into a line. You designate points that define the fixed sides of the lot and the direction of the variable-length side.



See the on-line help for more details.

Swing Into a Curve

This option designs a fixed-area lot by swinging a line from a point into a curve. Operation is much like the Swing into a Line command, except that the variable-length segment is a curve.



See the on-line help for more details.

Drawing Utilities

Arithmetic Calculator

This invokes a command line calculator. You are prompted for the number of decimal places to display, then prompted to enter your arithmetic expression. The answer is displayed on the command line. Press **Esc** to exit the calculator.

See the on-line help for more details.

Cut Out Drawing Detail

This program allows you to select a multi-sided polygon, rectangle, or circle for re-scaling to a different size. There are different parameters for this program so you can scale by a specific factor. Shown below is an example of how this command can be used to copy some information into a window. A scale of 2.0 was used in this example. The text was added after the copy.

See the on-line help for more details.

Set Cross Hairs to a Line

This option rotates the crosshairs on the display screen to match a line. In conjunction with the Draw Multisided Building option described below, it is useful in drawing a building whose sides are not parallel to the coordinate axes.

See the on-line help for more details.

Set Crosshairs Horizontal

This option returns the crosshairs to their normal orientation after you have used Set Cross Hairs to a Line. It has no prompts.

Mapping Grid

This routine allows you to draw a grid overtop of your drawing. You have several options available on the dialog box that comes up. You can pick or enter the lower left and upper right limits of the grid area. You can control the spacing both vertically and horizontally, and then choose to draw lines or tic marks then label them either at the end or on the grid.

See the on-line help for more details.

Draw a Rectangle

This option draws a rectangle on the screen. It prompts for the lengths of two sides; a starting corner; and the direction of the first side. Since each value defaults to the last entered value, you can draw multiple sheds and other objects very quickly with this feature.

Draw Multisided Building

This option makes it easy to draw a multisided building when you have the side measurements.

See the on-line help for more details.

Lot/Condo Unit Numbering Routine

This routine enters sequential numbers on lots or condo units. You specify a starting number and the rotational angle of the text, and then mark a location for each sequential lot label.

See the on-line help for more details.

Replace Desc with Blocks

This option inserts a specified block in place of a specified word of text wherever it occurs in the drawing.

Replace Text

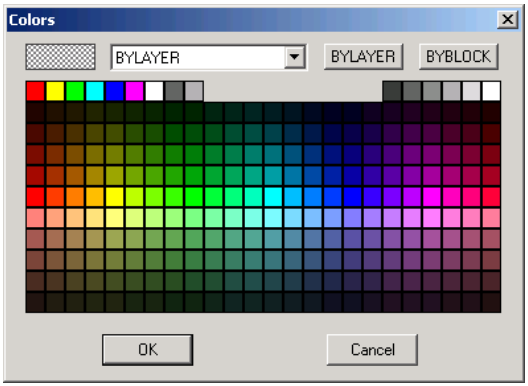
This useful routine will insert custom symbols in any description in the drawing regardless of how it got there. It should not be used to replace

the custom CAD block features, because it inserts the symbol at the text insertion point rather than the coordinate location. Trimming is not available in this option.

This option is similar to a "global search and replace" in a word processor: it replaces a string of text with another string wherever it occurs in a selected set of entities.

Change Colors of Objects

This option changes the colors of a selected set of objects. First a popup dialog box of colors appears. Select a color,



and you will be prompted to select objects. Select as many objects as desired with the usual object selection methods. The selected objects will be changed to the color you have chosen.

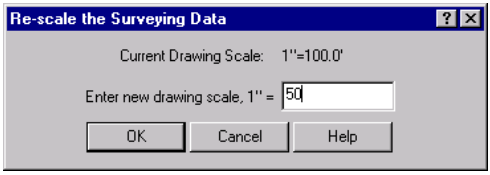
Re-scale Complete Drawing

This powerful option redraws the entire drawing at the scale factor you enter. All lettering sizes are re-computed. All point coordinates are re-stored. All MicroSurvey Lines, Symbols, Bearings, Distances, Point Numbers, Descriptions and Elevations are erased and redrawn. AutoMAP Blocks are also erased and redrawn at the new scale. Lines are re-trimmed to the correct distance back from the symbols. This routine checks the System Toggles before drawing anything so ensure the Toggles are set to draw what you desire.

NOTE: Any additional text or symbols (such as Title Blocks, Area Labels, Notes, Legends, Lot Numbers, North Arrows, etc) that you

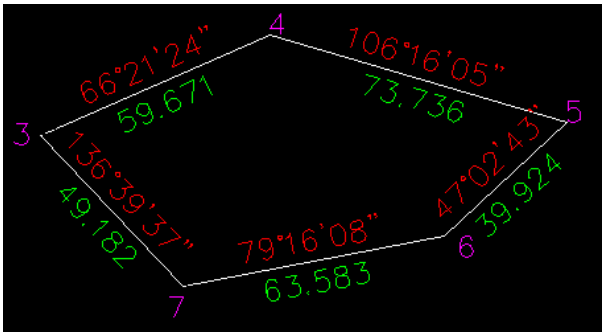
have added to the drawing but are not connected to the database will not be scaled. You must use the Modify → Scale command to adjust them to complete your drawing or erase them and add them again at the correct scale.

You will be shown what the current drawing scale is and be asked what scale you wish to change to.

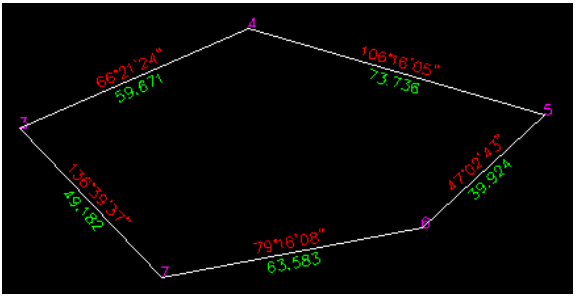


Then all of the Descriptions, Point Numbers, Elevations, Bearings, Distances and AutoMAP symbols will be rescaled based upon their defaults and the new drawing scale you have entered.

If there are discrepancies between the location of these entities in the drawing and the MicroSurvey databases, the points will be re-computed based on the location known in the database and not the drawing. Running Update Database from Drawing first should remove any discrepancies. The figures below show a drawing before and after a Re-scale operation.



Before re-scaling



After re-scaling

Re-scale with a Unit Change

An often frustrating experience happens when a customer tells you at the eleventh hour that he wanted the drawing in Feet, not metric or vice versa. We've outlined the basics below to help you to make the change with a minimum of effort.

Rescale from Feet to Metric

1. Make sure that all layers are on, unlocked and thawed.

2. Confirm that you have your CAD Drawing Toggles for points, point numbers, descriptions and elevations set as desired. Changes to these settings will be reflected in the rescaled drawing.

3. In the General Configurations Options dialog, change the Distances setting to metric and the Drawing Scale factor to the desired metric scale.

4. If you have a different Automap library set up for Metric drawings, go to the Automap library, select New Library, then Open the correct library for your new units and scale.

5. Scale entire drawing by 0.3048 using the SCALE command from the Modify menu. You will have to select the base point on the screen so set the Object Snap to point only. It is important to use a known base point so that you can rescale back if needed. If in doubt, create and use a point with coordinates 0,0,0. (Note: If your work includes elevations, the base point must have an elevation of 0.000. If your base point has an elevation, the elevations will be rescaled based upon the elevation of the base point) You will be

prompted with point protection for the first point being rescaled, toggle it off.

The SCALE command will update the text size of all bearings, distances, descriptions, elevations, point numbers and Automap symbols to correspond with the new drawing scale and units. These items will return to their default positions if they have been manually moved. If any default settings for these items have been changed, the rescaled drawing will reflect the current settings. Note also that CAD dimensioning will update but the text size/height will not.

6. CAD text and other items not controlled by the database now need to be rescaled. Calculate a factor by multiplying the feet scale by 12 and then dividing the result by the metric scale. For example, if your feet scale was 1"=50' and your new metric scale is 1:500, multiply 50 x 12 =600 then 600/500 = 1.2. You'll have to rescale these items by the calculated factor in pieces because their base points will be relative to the rescaled drawing. It is only their size that needs to be modified.

Don't forget to correct your scale bar and to turn point protection back on!

Rescale from Metric to Feet

1. Make sure that all layers are on, unlocked and thawed.

2. Confirm that you have your CAD Drawing Toggles for points, point numbers, descriptions and elevations set as desired. Changes to these settings will be reflected in the rescaled drawing.

3. In the General Configurations Options dialog, change the Distances setting to Feet. Change the Drawing Scale factor to the desired feet scale.

4. If you have a different Automap library set up for feet drawings, go to the Automap library, select New Library, then Open the correct library for your new units and scale.

5. Scale entire drawing by 3.28083989501 using the SCALE command from the Modify menu. You will have to select the base point on the screen so set the Object Snap to point only. It is important to use a known base point so that you can rescale back if

needed. If in doubt, create and use a point with coordinates 0,0,0. (Note: If your work includes elevations, the base point must have an elevation of 0.000. If your base point has an elevation, the elevations will be rescaled based upon the elevation of the base point) You will be prompted with point protection for the first point being rescaled, toggle it off.

The SCALE command will update the text size of all bearings, distances, descriptions, elevations, point numbers and Automap symbols to correspond with the new drawing scale. These items will return to their default positions if they have been manually moved. If any default settings for these items have been changed, the rescaled drawing will reflect the current settings. Note also that CAD dimensioning will update but the text size/height will not.

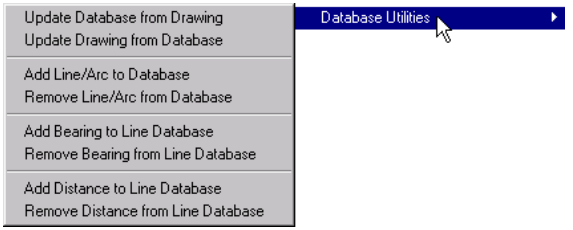
6. CAD text and other items not controlled by the database now need to be rescaled. Calculate a factor by dividing the metric scale by the feet scale multiplied by 12. For example, if your metric scale was 1:500 and your new feet scale is 1" = 50', 500 / 50 x 12 = 500/600 = 0.83333333. You'll have to rescale these items in pieces because their base points will be relative to the rescaled drawing. It is only their size that needs to be modified.

Don't forget to correct your scale bar and turn point protection back on.!

Following these steps should minimize the effort required for converting drawings between feet and meters. The concepts outlined will work for any unit conversion.

Database Utilities

These tools are provided in case your database and drawing become out of synchronization. These tools will help you organize the information so that the drawing and the database are the same.



Note: These are hot links in the on-line help. See the help for more information on these functions.

Update Database from Drawing

This selection resolves differences between your drawing and the external MicroSurvey databases. It searches all the entities in the drawing and compares their MicroSurvey handles with those in the databases. See Also: Update Drawing From Database

Update Drawing From Database

This program will walk through the database and replace all the coordinates on the screen. This is occasionally necessary if the database gets out of sync with the drawing. This will move any points, in the drawing, that are in the wrong place, based upon what the database indicates

See Also: Update Database from Drawing

Add Line / Arc To Database

This will reconnect a line or arc to the database and either use the points at the end of the line or arc, or it will generate new points if non exist.

Remove Line / Arc From Database

This will remove the line or arc from the database and it will also erase the line or arc from the drawing. To get the line back simply inverse between the two points in COGO.

Add Bearing To Line Database

This will reconnect a bearing to a line in the database. You will be asked for the bearing to connect and then the line to connect it to. Any future activity that affects the line will now have the bearing automatically updated.

Remove Bearing From Line Database

This will disconnect the bearing from the line. You will be asked which bearing to remove and then the line to remove it from. Any future updates of the line will not affect the bearing.

Add Distance To Line Database

This will reconnect a distance to a line in the database. You will be asked for the distance to connect and then the line to connect it to. Any future activity that affects the line will now have the distance automatically updated.

Remove Distance From Line Database

This will disconnect the distance from the line. You will be asked which distance to remove and then the line to remove it from. Any future updates of the line will not affect the distance.

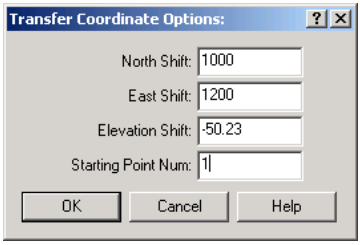
Other Program Ties

This selection pulls down a submenu of options for reading in data from other programs or exchanging data between your MicroSurvey data files and the data files used by the older DOS based MicroSurvey 10/386,11/ 386 and 12/386 programs. This feature lets you hook into the older data files from other programs such as Rapid Transit, Softdesk, Eagle Point, Carlson, AGEO and our own MicroSurvey projects, and read the coordinates directly into the MicroSurvey database.

See the on-line help for more details.

Transfer Points Between Jobs

This option transfers a range of points from an existing MicroSurvey project file into the project on which you are working. The standard File selection dialog box will appear to allow you to pick the file. When you have chosen a valid MicroSurvey job, the following dialog box will appear:



You can transfer the points into the current job with any Northing, Easting, and Elevation shift you choose. The points will transfer into the current job and they will be *renumbered* starting from the number entered in the dialog.

When you pick OK, the program will ask:

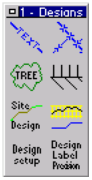
Enter Range of points to transfer:

Enter a range of numbers you wish to transfer from the other job. This prompt will repeat until all the point ranges have been transferred. Press ESC to exit.

Tip: If you want the exact point numbering of the "parent" job, specify a starting point number of 0 (zero). You will still have the option of specifying a range of points to import to your current job but they will not be renumbered.

Design Menu

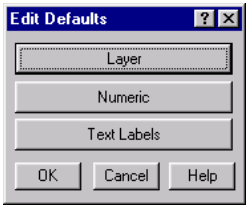
Design Toolbox



Design Defaults

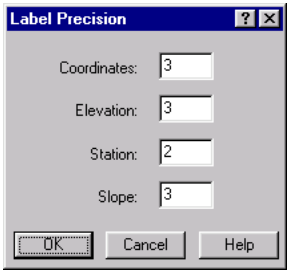
This selection lets you control the default settings for design layers, numeric values and text for labels. A unique set of defaults will be automatically loaded depending on whether you are working in metric or in feet.

Note: These two defaults files are independent of one another so changing feet settings will not affect the metric settings and vice versa. The defaults are saved within the MicroSurvey, not with the drawing.

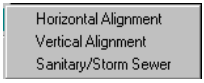


Design Label Precision

Specify the number of decimal places used for the drawing labels used for Design Menu routines.

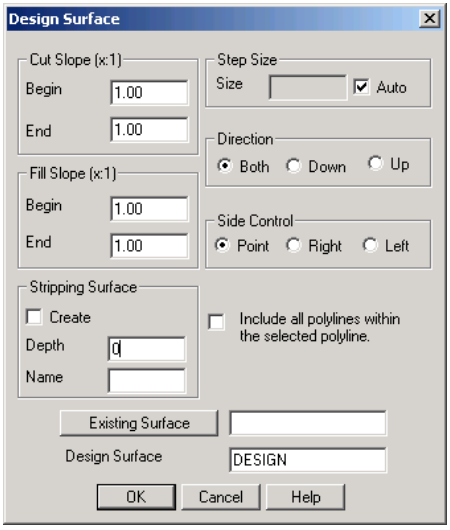


Design Palettes



Auto Site Design

Overview on how to Design A Surface



After filling out the dialog box and picking OK, you will be asked to select between drawing a polyline or selecting an existing polyline. Either way, the polyline can only consist of straight-line segments. No arcs are permitted. The polyline must also be drawn in 2D at whatever

elevation is correct for this job. **Note:** The data extraction, filter by entity option MUST be toggled off for this routine to work properly. See the Modeling - Data Extraction configuration section for details.

`Define design polyline. Draw/<Select>:`

Draw Option

Enter design surface elevation <0.000>: 100
Enter the desired elevation and begin drawing the polyline. You do not need to close the polyline, just press ENTER after the last point and the program will close it automatically.

Select Option
The select option allows you to pick an existing polyline and use it instead of drawing a new one. No curves are permitted in the polylines.

AutoRoute

Quickly design a road plan/profile by defining a horizontal alignment centerline, extracting existing ground data and design a finished grade profile. Create a new design surface and plot cross sections.

See the on-line help for more details.

AutoRoute Horizontal

Use this function to locate the POT's, (Point On Tangent/angle point)(Red Circles) on your drawing. The location may be determined by picking with the cross hairs, entering Northing/Easting coordinates or by standard angle/distance reference (@dist<angle).
Prompts are as follows:

`Enter alignment name:` Type in the name of the alignment.

`From point:` Enter xy co-ordinates or pick a spot on the drawing.

`To point:` Type A to enter a vector angle, D to enter a deflection from the previous POT, type in the xy or pick a point on the drawing. This prompt will repeat until you press <esc> or right click with your mouse.

A circular attribute block is placed at the POT (Red Circles) and tangent lines connect them. All subsequent alignment geometry information is retained as attributes by these POT blocks. Use the edit tools provided for modifying the alignment.

Edit Alignment

Use this function to access tools for editing an existing alignment. Here you have five functions at your disposal that allow you to add move or delete alignment POTs (point of tangent) as well as create circular or spiral circular compound curves.

The options available upon execution of the Edit Alignment function are:

Add POT
Delete POT
Move POT
Add **Curve** to POT
Add **S**piral-**C**urve-**S**piral combination at POT

See the on-line help for more details.

AutoRoute Vertical

Create Cross Section Template

The graphic template editor allows you to view an image of the cross section as you enter and edit values. It consists of Left and Right side radio buttons. The individual template legs and values are edited one at a time. The template legs are numbered from the centerline out to the edges of the template. Use the scroll arrows to move from one leg to the next. Enter desired leg width and elevation differences in the edit boxes.

See the on-line help for more details.

Profile Alignment

This routine is used to generate a vertical profile of an alignment in your drawing based upon a chosen surface. It will generate and label the centerline profile and place it in your drawing where you choose.

See the on-line help for more details.

Design Vertical Alignment

Once the existing ground has been profiled you can begin designing the finished grade. You do this by placing Vertical Point of Intersection (VPI) symbols at grade breaks on the profile. Once you Pick a starting point on the grid then move the cross hairs around the profile grid the distance and direction from the previous VPI are displayed on the status line at the bottom left of the screen. When you pick a point on the screen a dialog box comes up displaying the values for the Station, Elevation and Grade. You can alter the values in the edit box to fine-tune the actual locations. You will pick as many of these VPI's as required to design the full length of the alignment with all the breaks in grade marked.

See the on-line help for more details.

Edit Vertical Alignment

The edit options for a vertical alignment are Add/Curve/Delete/Move:

ADD:
CURVE:
DELETE:
MOVE:

See the on-line help for more details.

Query/Label VPI

Use the function to display current values at a selected VPI. Station and elevation are displayed, and if a curve is present its data will also be displayed.

Read .VRT File

Draw a design profile from a saved text file with the extension VRT. You will be asked to select a VRT file from the hard drive

Write .VRT File

Saves the design profile to a text file with a file name that matches the Profile Grid Name as specified in the grid dialog with the extension VRT.

AutoRoute Cross Sections

Create New Design Surface

Create a new design surface based on the horizontal alignment, vertical alignment, surface and cross section template. All these elements must exist in the drawing. Points from the existing surface are added to the drawing. The cross section is applied to the horizontal alignment using the APSEC function. Points within the polylines that define the slope intersection of the template and the existing surface are moved to a different layer and frozen. The remaining surface points and the new polylines created by the template are extracted to a new surface and renamed. A TIN is drawn of the new surface.

See the on-line help for more details.

Extract Cross Sections from Alignment

This program will draw a series of cross sections along your alignment. The program has many options for cross section spacing. When you have complete computing the cross sections, you can extract them to a file for plotting.

See the on-line help for more details.

Across Full Surface

Compute Cross sections on an entire surface quickly and easily with this command. The program will prompt you for direction and spacing of the cross sections. When done it will plot the cross sections for you.

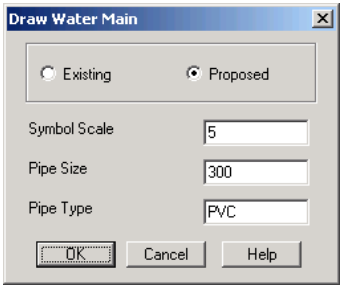
See the on-line help for more details.

Sewer/Storm Mains

See the on-line help for more details.

Water Mains

Draw lines and arcs and insert pipefittings and valves for a water main. Labels are drawing parallel to the pipes. Both Existing and Proposed lines can be drawn. The Existing pipelines and symbols are drawn on the layer EXISTING_WATER where the Proposed pipe lines and symbols are drawn on the layer PROPOSED_WATER.



Example:

From point:
To point:
To point Arc/Fitting: Fitting (the Water Symbols Palette is brought up to choose from)
Select fitting: wgvm

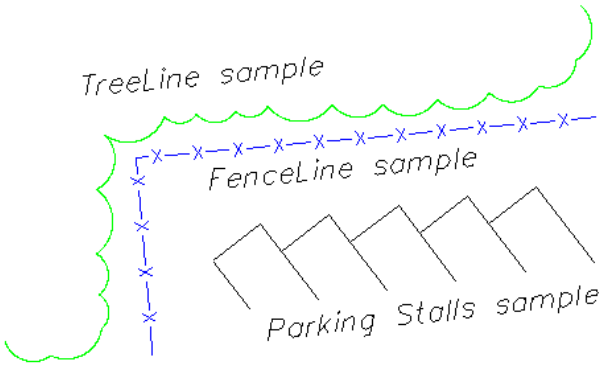
Sample Output:

Text Line

Draw a line with text inserted at user intervals. You choose the Text to be placed on the line as well as the spacing. This routine does not follow arcs. The lines between each piece of text are separate and can be manipulated individually. The text and the lines are both placed upon the layer TEXT_LINE by default but can be changed in the Design Defaults under the Layer option.

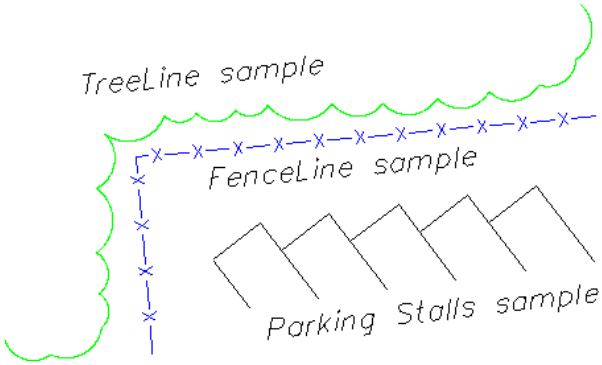
Fence Line

Draw lines with a letter X inserted at intervals to represent a fence line, and does not follow arcs. The letter X and the line segments are both placed on the layer FENCE_LINE, by default but can be changed in the Design Defaults under the Layer option.



Tree Line

Draw a polyline along the edge of a treed or forested area. Arcs of varying sizes, governed by the current text size variable, to form a scalloped line, replace the polyline. The arcs come in on the layer TREE_LINE by default but can be changed in the Design Defaults under the Layer option.



Parking Stall Layout

This routine allows you to create parking stalls. You can enter the exact size of the stall as well as the rotation angle desired for each stall. You also have control to design it as a single line of stalls or a double, back to back, line of stalls.

See the on-line help for more details.

Symbol Palettes

Display icon palettes of symbols for channelization, traffic signals, surface features, water fittings, survey markers, gas/telephone/power, and North Point and Scale Bar symbols.

Each palette has a button to pick to set the symbol scale. If you do not set the scale then upon the first insertion of any of the symbols, you will be asked for the scale. Once inserted, you will then be asked for the rotation of the symbol.

These symbols are from the American Public Works Association libraries (all except the North Point and Scale Bars palette).

Sewer/Storm Mains Submenu

Layout Network

Draw sanitary or storm sewer manhole structures and pipes in plan view. You can either insert individual structures or connect to existing structures and lay out a series of connected structures.

See the on-line help for more details.

Edit structures

You have the options to Add, Delete, Change Elevation, Change Grade or Move a structure.

Add
Delete
Move
Elevation
Grade

Sta/Offset/Rim

Add station, offset and rim elevations to manholes. Select from the dialog box to apply stationing based on the sewer alignment, a road centerline alignment, keyboard entry or none. Also select whether to retrieve the rim elevation from a surface, by keyboard entry or none.

See the on-line help for more details.

Tag Manholes

Tag manholes with station, offset and elevations. Once the structures have been stationed and all elevations have been assigned to rim and inverts (using the STA/Offset/Rim command) you may use this function to tag and display that information on your drawing. An example is shown below.

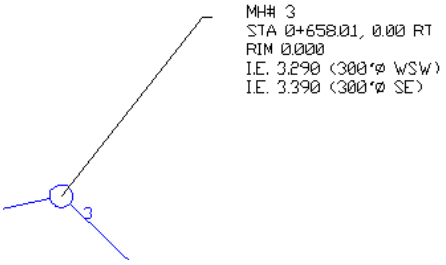


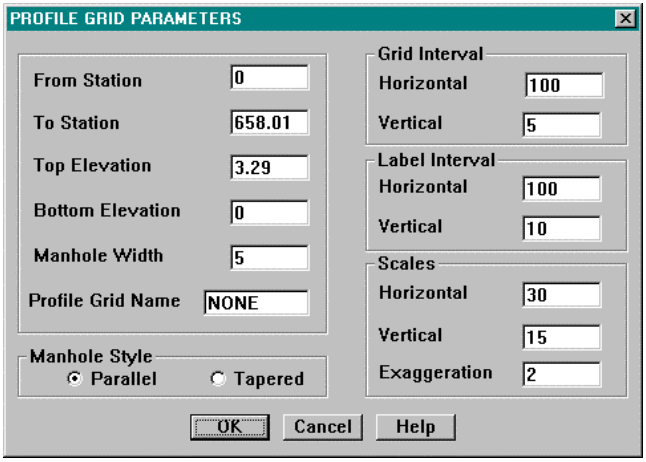
Table of Structures

Make a table of pipe runs between manholes. From and To manhole numbers are listed along with invert elevations, pipe lengths, pipe size and direction.

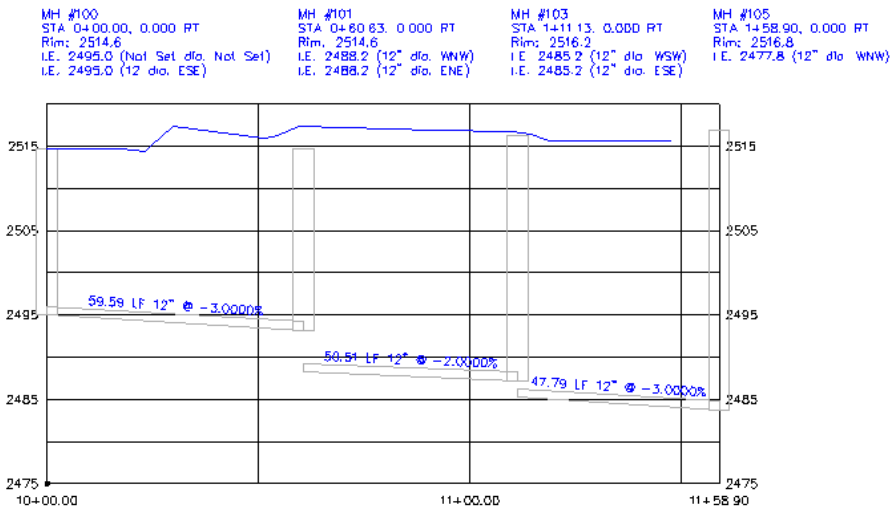
FROM	INVERT	TO	INVERT	LENGTH	SIZE	DIRECTION
3	3.390	4	6.680	317.04	300	316
2	0.000	3	3.290	329.05	300	12
1	0.000	2	0.000	329.00	300	342

Auto Profile

Draw sewer profile from plan view data. Enter the beginning and ending structure to profile. Edit the PROFILE GRID PARAMETERS dialog box. It displays data retrieved from the structures. You may want to modify the upper and lower limits of the grid as well as the intervals for lines and labels.



Once you specify where to locate the lower left corner of the profile grid, the data is read from the plan view of the sewer layout and drawn in profile. You will also have the option of extracting the existing ground elevation from the surface in memory and drawing it on the profile.

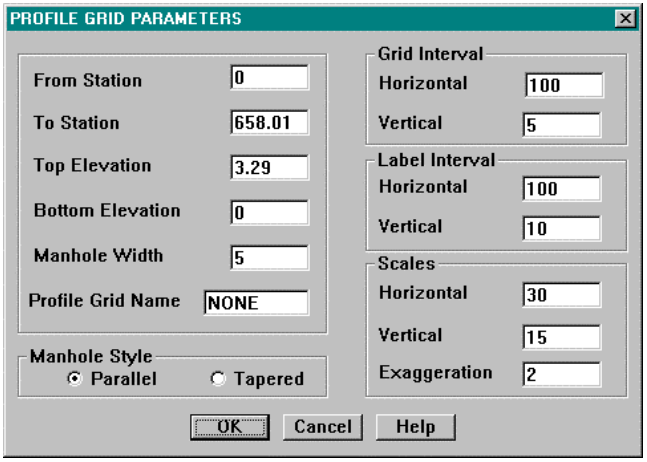


Manual Profile

Draw profile from manually entered data. All profile grid, manhole station, offset, rim elevation, and inverts are entered from the keyboard.

Example:

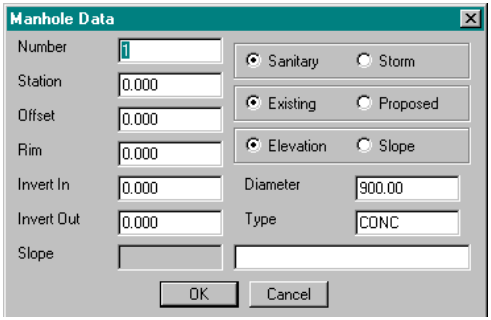
Draw new profile or use existing Existing/<New>: New

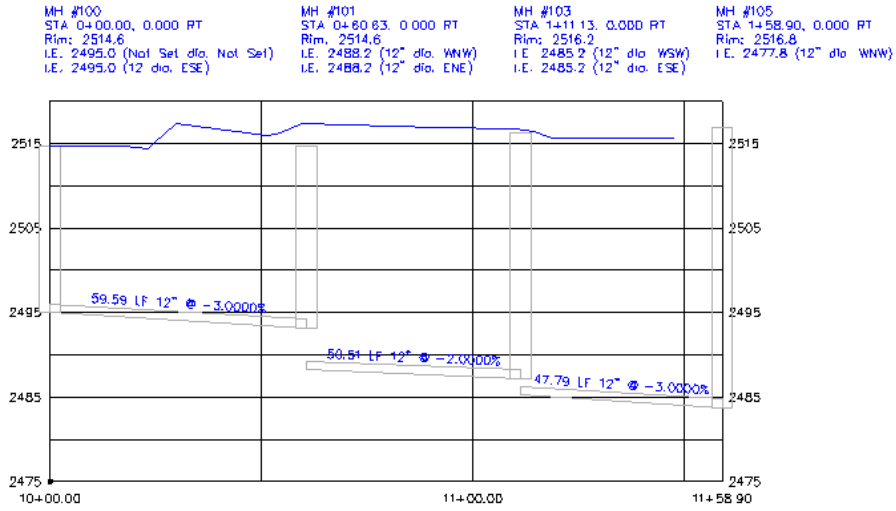


Begin grid from the Right/<Left>: **L**

Pick location for lower corner of grid: <pick a spot on screen>

Place manhole First/Next/Exist/Done <First>:

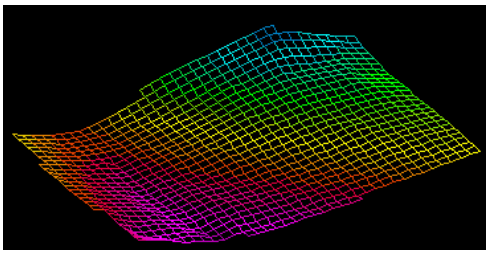



MH #100
STA 0+00.00, 0.000 RT
Rim: 2514.6
I.E. 2495.0 (Not Set dia. Not Set)
I.E. 2495.0 (12 dia. ESE)
MH #101
STA 0+60.63, 0.000 RT
Rim: 2514.6
I.E. 2488.2 (12" dia. WNW)
I.E. 2488.2 (12" dia. ENE)
MH #103
STA 1+11.13, 0.000 RT
Rim: 2516.2
I.E. 2485.2 (12" dia. WSW)
I.E. 2485.2 (12" dia. ESE)
MH #105
STA 1+58.90, 0.000 RT
Rim: 2516.8
I.E. 2477.8 (12" dia. WNW)
59.59 LF 12" @ -3.0000%
50.51 LF 12" @ -2.0000%
47.79 LF 12" @ -3.0000%
2515
2505
2495
2485
2475
10+00.00
11+00.00
11+58.90

Modeling Menu

What's a surface?

MicroSurvey creates and manipulates surfaces without the necessity of drawing it into your CAD drawing. A MicroSurvey surface is the mathematical description of a surface that exactly honors all input data points. MicroSurvey surfaces are a single-valued function of independent variables X and Y. This means that a surface only has one Z value for any given (X, Y), and so does not model overhanging surfaces or exactly vertical surfaces.



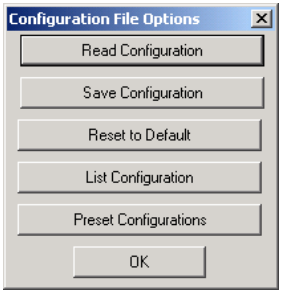
A Surface may represent anything. Existing or proposed topography, thickness maps, geologic structure maps, concentration distribution, slope maps, pressure gradient maps may all be represented as MicroSurvey surfaces. Surfaces may intersect. Overhanging surfaces may be modeled in multiple patches. MicroSurvey has no limit on the number of points in a surface or the number of surfaces simultaneously used. The ultimate limitation is available space on your hard disk drive. Surfaces contain one or more parts such as points, break lines, triangulated irregular networks (TIN), grids or triangulated grids (TGRD).

A surface is not a CAD drawing entity, rather it is a mathematical description held in surface memory. Representations of a surface, such as points, contours, grids or TINs may be drawn into your CAD drawing as point, line, polyline, 3D face, polyface mesh or mesh entities. It is important to keep the distinction between surfaces (which reside in surface memory) and drawn CAD entities representing parts of surfaces (which reside in the CAD drawing database).

All drawing entities created by MicroSurvey are placed in their proper position in 3D model space.

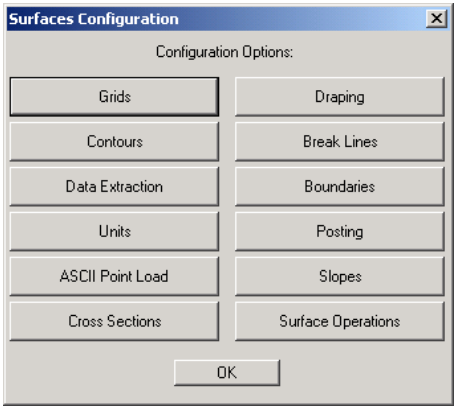
Configuration File Options dialog

This dialog allows you to access any of the configuration file operations including loading pre-set example configurations. Click on the button of interest to jump to that topic.



Configuration Settings dialog

This dialog allows you to access any of the configuration dialogs.



Configuration Settings

Configuration File Options

Read Modeling Configuration File

Reads option settings from a previously saved configuration file, and makes them active for the current surface and drawing. When invoked from a dialog box a standard file dialog will be invoked, otherwise a command line prompt will ask for a file name.
When a drawing is loaded with the Open command, the system will automatically attempt to load options from the default file. If <drawingname.qcf> is not present MicroSurvey will look for a QS.QCF file in along the search path. If that is not found, MicroSurvey will use its internal defaults.

Write Configuration File

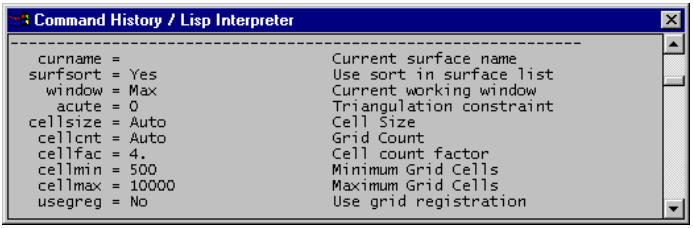
Saves current configuration settings to a named file. When invoked from a dialog box a standard file dialog will be invoked, otherwise a command line prompt will ask for a file name. Enter a file name without an extension (.qcf will be appended automatically), or press enter to accept the default of <drawingname.qcf>.

Reset Configuration

Resets MicroSurvey to use its internal defaults. Loading the Configuration file QS.qcf is generally a better idea, because resetting uses fairly severe settings (like no color).

List Configuration

Lists the current MicroSurvey option settings to the text screen. The display format is as follows:



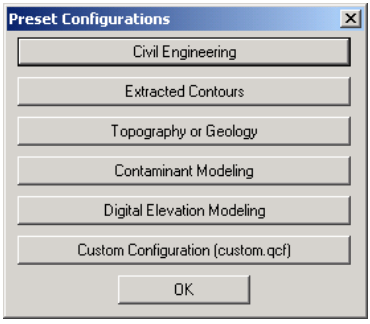
Approximately 130 entries will be listed. The configuration file is an ASCII text file with the extension .QCF. Although parameters are normally set using the dialog boxes, any keyword, which appears in the left side of the .QCF file, may be set manually using the QSOPT keyboard command.

Command: QSOPT

Keyword: enter keyword

After supplying a keyword, such as cellsize, you will be prompted for a value, along with a brief description of the meaning of the option. Enter the value as requested. The new setting will be in effect for the rest of the drawing session unless changed.

Preset Configurations



MicroSurvey includes a number of sample configurations that may be loaded as a starting point. Please review the settings for each loaded to configuration before using them so you understand how they have modified the program.

Configuration Files

Most aspects of MicroSurvey may be configured to suit your specific application. This topic describes all of the commands on the Surfaces Configuration. The configuration of all Surfacing option settings may be saved to a named disk file and read back at any time. You may keep as may different configuration files as you wish. Several example prototype configuration files are included. Using configuration files will speed your work by avoiding having to re-establish options settings.

Surface configuration files are ASCII text files with the extension .QCF. Configuration files are read automatically when MicroSurvey is loaded or you open a drawing. When you open a drawing configuration files will be searched for in the following order:

1. <drawingname>.QCF
2. QS.QCF

If a configuration file with the same name as the drawing exists it is loaded; if not, QS.QCF is loaded if found; if neither is found, MicroSurvey uses its internal default settings.

Saving a configuration file with the same name as the current drawing will cause the configuration to be automatically reloaded the next time the drawing is opened. All MicroSurvey settings will be restored automatically.

You may create a default custom configuration by saving your desired settings to QS.QCF in the directory in which MicroSurvey is installed. After doing so, any drawing without a custom configuration file will use the settings in the QS.QCF file.

Configuration file contents

Configuration files are ASCII text files that contain the current configuration of all global modeling variables.

Grid Configuration Options

Grid Configuration dialog box

The Configure Grid dialog box settings control how grids are constructed for both grid and triangulated grid surface parts.

See the on-line help for more details.

Contour Configuration Options

Contour Configuration dialog box

The basis for contours (TIN, Grid or TGRD), contour interval, and levels of automatic contouring are controlled by the Configure contours dialog.

See the on-line help for more details.

Data Extraction Configuration Options

Extract Configuration dialog box

Configure Extract dialog box allows you to filter which entities you extract; densify lines and polylines during extraction; determine whether spline or frame points are extracted from polylines which have been smoothed; and limit the maximum number of points extracted.

See the on-line help for more details.

Units Configuration Options

Unit Configuration dialog box

Different disciplines use different unit conventions for slopes, areas and volumes. MicroSurvey allows you to specify how to display measurement of slopes, areas and volumes.

See the on-line help for more details.

Cross Section Configuration Options

Cross-section Configuration dialog box

The Configure Section dialogs control the scaling, labeling and layers used when building 2D profiles and cross sections using the cross-section command.

See the on-line help for more details.

Draping Configuration Options

Drape Configuration dialog box

The Drape command projects an entity vertically onto a surface (TIN, Grid or TGRD) in memory so that the entity registers at the local Z elevation of that surface. Drape either modifies the z coordinates of selected objects (points, text, blocks), or converts them to 3D polylines (lines, 2D polylines) which follow the shape of the surface as closely as possible for a given step size.

See the on-line help for more details.

Break Line Configuration Options

Break Line Configuration dialog box

Extract Breaks (QSBX) involves segmentation of straight line segments and approximations of curves for accurate modeling.

See the on-line help for more details.

Boundary Configuration Options

Boundary Configuration dialog box

The Configure Boundary dialog controls the criteria for determining when a TIN or TGRD face is displayed within a boundary.

See the on-line help for more details.

ASCII Point Load Configuration Options

Read ASCII Configuration dialog box

The Read ASCII Points command supports alternate column order and scaling of the values in your input file. You may scale x, y, or z independently during loading using the settings within the Configure ASCII Load dialog box.

See the on-line help for more details.

Surface Operation Configuration Options

Surface Operations Configuration dialog box

The sort order of the surface list displayed by the Surface Operations dialog box and the internal computation sequence of all surface operations is specified in the following dialog box.

See the on-line help for more details.

Delete surface points

Delete Surface Points allows you to select a drawing entity and delete the corresponding points from the <.> surface in surface memory. Note: This ONLY works on points on the <.> surface. If you need to delete points from a named surface, copy it to the <.> surface first, delete the points, and then copy it back to the named surface.

Points command

Displays the points in the current surface on screen.

See the on-line help for more details.

Breaks command

Shows or draws the break line data from the current surface.

See the on-line help for more details.

TIN command

Generates and/or displays a triangulated irregular network (TIN) for the points and/or breaks in the current surface.

See the on-line help for more details.

Triangulated Grid (TGRD) Command

Running Triangulated Grid generates a surface model based on a triangulated grid (TGRD), using points and break lines from the specified surface. The TGRD is a triangulated model incorporating grid nodes and densified break lines as vertices of a complex TIN.

See the on-line help for more details.

Grid command

The Grid command displays a grid model of the current surface. If a grid already exists in the current surface, this command simply displays

the grid. If a grid does not exist, one is created based upon the current settings of the Configure Grid dialog box. If the standard continuous curvature method is selected, a TIN and derivatives will also be created as needed.

During a grid calculation the status bar will report progress of any required TIN, derivative and the grid calculations.

A grid is computed only within the area of the smallest rectangle containing all the points. The window definition is normally handled automatically, but may be manually defined via the Surface Option -> Window command.

If a grid already exists in the current surface and you wish to create a new grid reflecting different cell parameters or grid methods, you must first clear the old grid and derivatives parts from the current surface using Surface Operations dialog or use one of the following surface operations which clear and regenerate the grid in one step: window, cellsize, cellcount and cellfactor.

See the on-line help for more details.

Contour command

The contour command generates and/or displays contours of the current surface.

See the on-line help for more details.

Contour Interval

The contour interval may be set via the Configure Contour dialog box, or directly using the Contour Interval menu command. Command line versions will prompt:

Contour Interval/Auto <Auto>: enter value

Enter the desired contour interval from the keyboard. If you respond with Auto you will be prompted for the number of levels to use. The number of levels represents the number of intervals the Z range of the data points is divided into while automatically choosing a contour interval.

The Auto default is 20 levels.

Extract from Drawing

Extracting Surface Data from Drawing Entities

There are three commands which create surface data in the <.> surface by extracting x,y,z information from CAD entities in your drawing.

Extract to Surface creates a NEW <.> surface of points, deleting any pre-existing data in the <.> surface.

Merge Extract incrementally adds points to the existing data in the <.> surface.

Extract Breaks incrementally adds break lines to the existing data in the <.> surface.

Objects may be selected with the normal CAD selection methods. Pressing enter will select all visible entities on screen. Use the **Return to select all visible** prompt with caution as you may extract elevation information from unintended entities (such as TEXTs or INSERTs). If you want to add points to an existing surface, use Merge Extract.

See the on-line help for more details.

Extract to Surface command

Extracts X, Y, Z points from CAD drawing entities and loads them into a surface. The < . > contents will be destroyed when running this command.

Merge Extract command

Merge extract functions exactly like **Extract to surface** with one major difference: Merge Extract incrementally adds the extracted points to the results < . > surface, as opposed to Extract to surface which deletes the existing < . > surface and creates a new one.

Extract Breaks command

The **Extract Breaks** command extracts drawing entities from the drawing.

See the on-line help for more details.

What happens when you add break lines

Break lines represent 3D continuous traces in space (think of them as a 3D polyline) which

1. Define the surface elevation
2. Allow slopes to be different on either side of the break line

When MicroSurvey extracts or loads break lines, it must find any crossing break lines and insure that they cross at the same elevation and then re-triangulate the surface and check that no triangle crosses a break line. If break lines cross at different elevations, a new point is added at the crossing at the average elevation. Then it compares edges of the triangles of the existing TIN to the break lines to see if any triangle edges cross break lines. If this occurs MicroSurvey adds points at those crossings and re-triangulates the TIN. This process is called auto-densification.

You will see a sequence like
Auto densification...
xxxxx triangles built
xxx additional points added to current surface
Auto densification...
xxxxx triangles built

Each time an auto-densify cycle completes, it compares the TIN to see if a triangle edge crosses a break line and repeats the cycle if needed. Normally this converges in a few iterations to a solution where no triangle edge crosses a break line.

The settings of Configure Breaks and the Autocycles variable affect how fast convergence occurs. If you are experiencing many cycles of auto-densification see these two topics for adjustments that will correct the situation. Normally only pathological geometry and massive numbers of break lines require intervention.

Import Data

Importing surface data

Data may be directly loaded from ASCII text files from spreadsheets, databases, total stations or digital elevation models into surface memory without having to place the points or breaks into the drawing.

See the on-line help for more details.

Read ASCII Points command

The Read ASCII Points command loads one surface at a time from an ASCII file of free-form x,y,z data.

Read ASCII Table command

The Read ASCII Table command is designed to load single or multiple surfaces from an column oriented ASCII file in a one pass. It can read delimited or fixed field data structures such as those exported from spreadsheets, database managers or total stations.

See the on-line help for more details.

Read ASCII Breaks command

Read ASCII Breaks allows you to read break line data (representing 3D polylines) from a generic ASCII file format consisting of X, Y, Z triplets.

See the on-line help for more details.

Read ASCII Boundaries command

2D polyline data representing boundaries may be read from an ASCII file.

See the on-line help for more details.

Read QSB

Invokes the standard file dialog box to read surfaces from a MicroSurvey binary QSB file that has been previously saved. QSB files are created by the **Write QSB command**.

See the on-line help for more details.

Export Data

Data Export commands

Surface or drawing entity data may be exported directly to several different text and graphic formats.

You may also draw the desired surfaces parts into the drawing and use the DXF and DWG save facilities from the CAD program to write those formats.

Export ASCII from memory command

Points, breaks or grid nodes may be exported from surface memory directly to a comma-delimited ASCII file. The menu selection is found under Export Data.

See the on-line help for more details.

Write QSB

Writes the selected surfaces to a binary QSB disk file. See the on-line help for more details.

Write ASCII Boundaries command

The current boundary definition (created with Set boundary) is written to an ASCII file. See the on-line help for more details.

Export ASCII from drawing entities command

This command extracts points from CAD drawing entities and writes them to an ASCII file as space delimited x,y,z triplets, one per line. See the on-line help for more details.

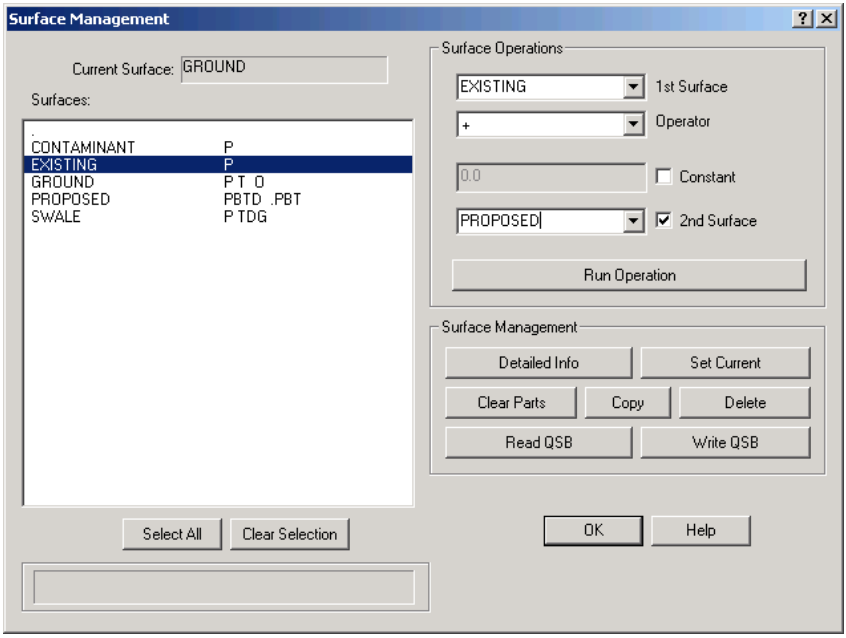
Surface Operations

Surface Operations Overview

Surface Operations allow you to perform mathematical calculations between surfaces. Surfaces may be copied, renamed, deleted, and read from or written to disk. Individual parts of one or more surfaces may be selectively cleared. Surface Operations allows inspection of detailed surface statistics for any surface. The Surface Operations command from the menu invokes the Surface Operations dialog box from which all surface math and manipulation is performed.

Surface Operations dialog box

Surface management and surface math are accomplished by invoking Surface Operations from the menu.



The surface operations dialog has three main divisions. The surface list is in the upper left quadrant; the mathematical controls are in the upper right quadrant with the surface management buttons beneath them.

See the on-line help for more details.

Surface Options

Four commonly used surface commands are clustered for convenience under the Surface Options menu. They may also be accessed via the surface operations dialog or the configuration dialogs.

Current Surface command

Sets the current surface to the surface name specified. This surface name will be offered as the default name in any subsequent Surface: prompts.

Surface Options Window command

The Window command restricts where grid and TGRD nodes are created. See the on-line help for more details.

Cell Size

The Cell Size settings control the X and Y dimensions of an *individual* grid cell. See the on-line help for more details.

Cell Count

This allows you to specify an exact number of grid cells in the X and Y directions. See the on-line help for more details.

Color Options

The colors for contours and surfaces are controlled separately. Color sequences may be remapped into more useable ascending sequence using the Remap colors options.

Contour Colors dialog box

Contours may be drawn in specific color sequences or simply drawn with color BYLAYER. See the on-line help for more details.

Surface colors

MicroSurvey objects representing surfaces (TIN, TGRD or Grid) may be colored based on surface properties such as elevation, slope, visibility, and lighting or by the elevation of a second surface. The Surface colors dialog box controls all aspects of surface coloration. Both the show and draw options support these color options, with the exception that some CAD entities such as meshes and polylines can only be displayed in a single color.

See the on-line help for more details.

Surface Colors dialog box

The Surface Colors dialog box allows you to pick a coloration method from the choices down the left side of the box.

See the on-line help for more details.

Set SHOW Color

Selects the color assignment method used when points, TIN, grid or contours are shown on screen with the Show option when Surface Color options are set to none. See the on-line help for more details.

Remap Colors dialog box

The Remap Colors dialog box allows you to interactively re-map CAD colors into a different color number sequence. See the on-line help for more details.

Boundary Options

Boundary command

You may limit the area in which Points, TINs, TGRDs, Grids, Contours, Posted or draped objects are displayed by specifying one or more closed polylines as boundaries with the Set Boundary command. See the on-line help for more details.

View Options

Surface Zoom

Command: SZOOM
Surface zoom allows you to immediately zoom so the view covers an area centered over and slightly larger than the selected surface. See the on-line help for more details.

Surface Plan View

Command: SPLAN
Surface plan view changes to a plan view centered over the selected surface. See the on-line help for more details.

Annotate

Post from memory

Command: POST

Posts values into the drawing directly from surface memory based upon the current settings in the Configure post dialog box. See the on-line help for more details.

Configure Post

Command: SETPOST

The Configure Post dialog box controls text height, rotation, justification and position (offset) of posted values displayed by the Post from memory command. See the on-line help for more details.

Post Entries

Command: DPOST

Post entities labels selected drawing entities with their z values. This command does not operate on points in surface memory, only selected drawing entities, such as POINT or INSERT entities. The number of digits displayed to the right of the decimal point is controlled by the linear units settings. See the on-line help for more details.

Edit Contour Vertex

Command: EDITCONT

Manually editing the 2D polyline entities representing contours has no effect on surfaces in surface memory. Changes made to contour polylines are reflected in surface memory only if the altered contours are extracted with Extract to surface and a new surface created.

Editing contours doesn't change surface memory! See the on-line help for more details.

Label contours command

This command Labels **drawn** contour lines with their z values at user-selected locations. This command will not work on "shown" contours. **Note:** Contours must be labeled prior to Smoothing. See the on-line help for more details.

Common contour labeling problems

If the Label contours command is not prompting you for text height, you may have a fixed text (rather than variable) height set. Running the Font command and setting text height to 0.0 (variable) will bring back the text height prompt.

If label placement is irregular, you may have object snap set to snap to a particular entity element. Set object snap mode to None to correct the problem.

The label command relies on the z value of the contour polyline to determine the elevation. Many editing commands can change the elevation of a contour (move, stretch, polyline edit, etc.). If you edit a contour polyline, then subsequently label it, you may get a different value than the original elevation.

If you have a very small text height set and pick a label location such that the text doesn't intersect the contour, in very rare cases the contour polyline will disappear. If this occurs, use the Undo command to recover the contour and re-pick the label location.

Be sure to label all contours prior to smoothing them to get proper labels. If you do this in the reverse order then the labels may not break the contour and the label will not follow the contour direction.

Smooth Contours command

Applies a smoothing algorithm to contour polylines and draped objects. This may improve the appearance of a contours generated using a sparse grid. Re-computing the grid to a more reasonable cell size is a better approach to smooth contours when contouring on the grid. Smoothing contours greatly increases file size, so use only where required. See the on-line help for more details.

Index Contours

Command: INDEX

Highlights major contours by widening them and/or moving them to another drawing layer. Contours lying at integer multiples of the index intervals are changed to the selected width and layer.

See the on-line help for more details.

Hachure Contours

Command: TICK

Draws evenly spaced, locally perpendicular tick marks (hachuring) on selected polylines.

See the on-line help for more details.

Design Tools

Drape command

Drape modifies CAD drawing entities to conform to their z values to the specified surface. Conceptually you are moving the entity vertically up or down until it lies in the surface. The Configure Drape dialog box controls the specific drape parameters.

See the on-line help for more details.

Profile Drape line (Flatten command)

Flatten creates an 2D elevation profile of a 3D polyline. Typically the 3D polyline being flattened has been draped so it lies within the surface.

See the on-line help for more details.

Intersect Slope

Intersect slope projects a user-specified slope from a 2D or 3D polyline until it intersects the current surface and draws a new 3D polyline where the projected slope and current surface intersect. Additional lines may be created by Intersect slope, if needed; to produce smooth radial corners or to properly represent V-shaped areas where two fill surfaces merge. The lines and 3D polylines created by Intersect slope will subsequently be used as break lines.

See the on-line help for more details.

Extrapolate

Extrapolate uses local triangulation and surface gradients to adjust the Z values of CAD drawing entities that lie adjacent to, but not overlying a surface. Points, lines, polylines and circles are the only entities modified. See the on-line help for more details.

Volumetrics

Volume Concepts

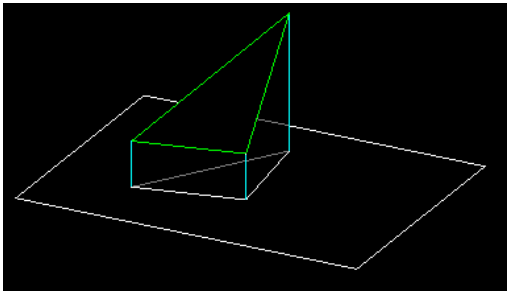
See the on-line help for more details.

Fast, accurate volumes are very important in many surface-modeling applications. Within Modeling, volumes may be computed directly from surfaces residing in surface memory using the **Surface volume** and **Area volume** commands or computed from a drawn TIN, TGRD or Grid using the **Volume by entity** command.

None of these volume functions use the current boundary that may have been set with the Set Boundary command. Use the Area Volume command to determine volumes beneath arbitrary areas.

TIN based volumetrics
MicroSurvey Modeling calculates volumes of a surface by summing the volume underneath each face of the surface within the area specified. A face may represent the either the triangles of a TIN or Triangulated Grid, or the rectangular grid cells of a Grid.

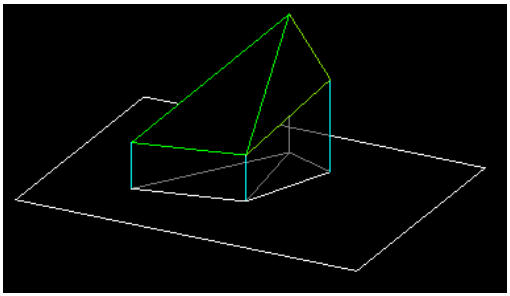
Volume under a triangle
For any surface with a TIN, calculating a volume consists of calculating the volume under each triangle in the desired area and summing the result. Remember that regular TINs and Triangulated Grids are just different types of TINs, both are composed of triangles. First let's look at one triangle of a TIN shown and determine its volume.



Volume under one triangle

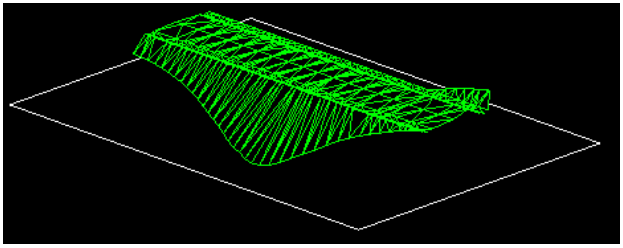
The volume under the green triangle is measured relative to the zero (XY) plane shown in white. The Z value of the surface used in a volume command represents thickness. If you use the Surface volume or Area volume commands, you may also calculate the volume between two surfaces or the volume between a surface and a constant. In these two cases, MicroSurvey Modeling calculates the thickness surface and places it in the results <.> surface. A Z value of zero in this surface represents zero thickness. All volume calculation is then performed on this thickness surface.

For the simple case of two triangles, the volume is the sum of the volumes under two triangles.



Volume under two triangles of a TIN

Volume under a surface
In the general case of calculating the volume under a surface consisting of many triangles, the total volume is just the sum of the individual volumes of the component triangles within the area to be calculated.



Volume under part of a surface

If the resultant faces extend both above and below zero datum, those faces above the zero plane are reported as positive volumes and those faces below the zero plane are reported as negative volumes. If a single face penetrates through the zero plane separate positive and negative portions of the face are computed and summed.
If you have subtracted an existing topographic surface from a proposed topographic surface, areas of fill will have positive thickness values and areas of cut will have negative thickness values. When volumes are then calculated, positive (fill) values are calculated as positive volumes and negative (cut) values are calculated as negative volumes. Reversing the order of the surfaces in the calculation will reverse the sign (+/-) of the resulting volumes.

Entire surface volumes
You rarely will want the volume under an entire surface unless you have very carefully controlled the edge conditions. Instead you will

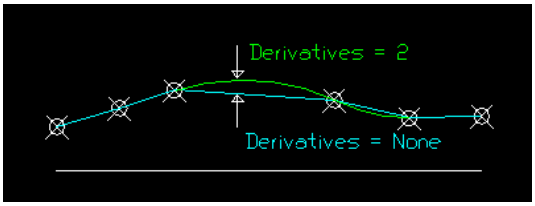
commonly use the Area Volume command to compute the volume beneath a specific area. If you want the volume under an entire TIN, you may use Surface Volume to return the entire volume. Alternatively, you may simply draw the TIN and use the Volume by entity command and select the drawn TIN. If the TIN was drawn as a polyface mesh, select the one polyface entity. If the TIN was drawn as individual 3D faces, select all of the 3D faces. It is easier to use the Surface volume command that returns the same result, but does not draw any CAD drawing entities.

Partial surface volume

To determine the volume under a sub-area of a surface, we first must insure that the edge of the triangles of the TIN follows the outline of the sub-area boundaries. The Area Volume command does this for you by internally draping the closed polylines defining the sub-areas onto the thickness surface and extracting them as break lines. This forces the triangle edges to follow the area defining polylines. The reason for doing this is that each triangle needs to be entirely in one sub-area, never straddling the dividing line between two sub-areas.

Triangle and grid cell volumes

Grid cells, TGRD triangles or TIN triangles may be used to compute a volume under a surface. They can yield slightly different results because triangles are treated as flat faces, whereas the grid or TGRD represent uniform sampling of a smoothed curved surface that passes through all the control points if you have Derivatives set to 2.



Cross section view

Volumes on the TIN (blue) are slightly different than on the grid or TGRD (green) which follow surface curvature when Derivatives are set to 2 in the Grid Configuration dialog box.

All of the different volume commands will produce identical results when run on the same surface parts. Volumes run on a TIN, TGRD and Grid of the same surface will yield different results, because of different amounts of curvature information carried by the different

surface parts. TGRDs and Grids may reflect surface curvature whereas regular TINs do not. Always visually examine a surface prior to calculating its volume.

Volume calculation from surface memory
Volumes may be computed directly from surfaces residing in surface memory using the Surface volume or Area volume command. None of these volume functions use the current boundary that may have been set with the Set Boundary command.

See the on-line help for more details.

Common volume calculation mistakes

The most common user mistakes in calculating volumes relate to boundary conditions. The following guidelines should be reviewed:

The thickness surface must be defined under the area to be calculated. If the difference between two surfaces is used, both original surfaces must be defined under the area to be calculated.

Use Area Volumes and one or more area polygons to constrain the surface to honor the exact boundary. Area polygons used with Area Volumes should not overlap or be nested.

Inspect the <.> surface visually by contouring it on the TIN in plan view or viewing the TIN, Grid or TGRD from a perspective view prior to calculating the volume. Is it reasonable?

Calculate the volume of the appropriate surface part based on the guidelines the Volumetric Concepts topic.

If you are comparing resulting two volumes calculated from two different surfaces, they must be computed under exactly the same area to have any meaning.

If you are using a volume conversion factor in the Configure Units dialog box, a mistake in entering the conversion factor will be reflected in all volumes reported.

What Area Volume does internally

The area volume command computes the volume between two surfaces for the selected area polygons (closed 2D polylines) as follows:

1. It uses the current setting for Maximize Surface Operations (default is bi-directional drape).
2. It uses the surface part specified in the volume dialog box. (When in doubt, use Planar TIN.)
3. It subtracts the two surfaces by draping the first surface points on the second surface, then the second surface points on the first surface, then subtracting the two values it now has for any point in either surface.

4. The resulting thickness surface only exists where both input surfaces overlapped.
5. The area polygon(s) are then internally draped on this thickness surface.
6. The draped area polygons are extracted as breaks into the surface. This forces each triangle to be entirely in one area or another, never partially in an area.
7. The entire TIN is then processed triangle by triangle to compute the volume under each triangle and accumulate the volumetric sum for each area.

The process is quite mechanical and calculates volumes based on a TIN representing thickness which is left in the <.> surface and should be examined each time. If this TIN is correct, the volumes will be correct.

Volume by Entity

Volume by entity calculates the volume under drawing entities that have faces, like drawn polyface meshes representing TIN, grid or TGRD surface parts. Unlike Surface volume and Area volume which operate on surfaces in memory, Volume by entity only operates on drawing entities such as meshes, polyface meshes and 3D faces drawn with the TIN, TGRD or Grid commands.

See the on-line help for more details.

Utilities

Modeling Elevation Utilities

See the on-line help for more details.

Modeling Utilities

See the on-line help for more details.

Modeling Polyline Utilities

See the on-line help for more details.

Modeling Polyface Utilities

See the on-line help for more details.

Modeling General Utilities

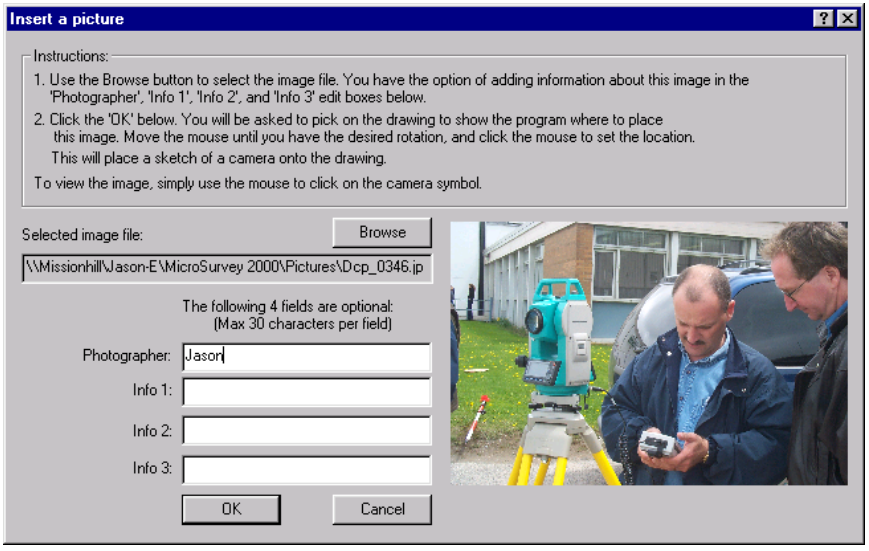
See the on-line help for more details.

Raster Menu

See the on-line help for more details on most of these functions

Insert a Photograph

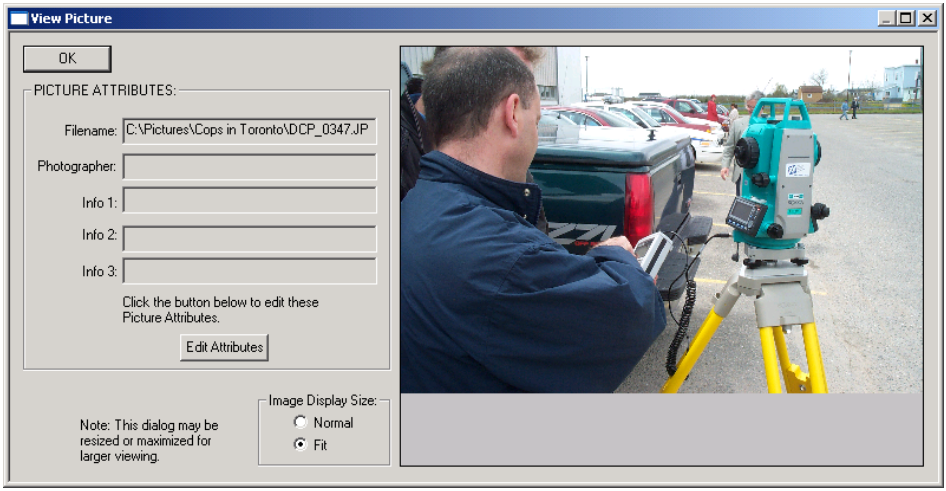
This routine allows you to insert the information for any photos taken at the job site.



You can browse to find the picture of your choice. You can also add attributes or comments to be stored with the photo.

You can set the insertion point based upon where you were standing when you took the picture, and then drag out in the direction you were facing. You will see the name and location of the picture inserted into the drawing.

You can then go back later and simply pick on the information and the picture will be loaded into a viewer automatically.



You can use the scroll bars to view different portions of the photo. You can also resize the dialog box and the photo will be adjusted accordingly. The Image Display Size option allows you to fit the image into the dialog to make it easier to view.

You are also permitted to edit the attributes that were added when the photo was first inserted.

See the on-line help for more details.

Load Raster Image File

This item allows you to insert a scanned, raster image that has been stored in the RLC, BMP or TIFF format, into your current drawing thus allowing you to see the raster image and the vector information at the same time.

There are many different formats that a scanned image can be stored in such as BMP, GIF, CALS, TIFF, etc., and RLC.

At this time the program will support RLC, BMP and TIFF formats. There are many conversion programs on the market that may assist you in changing from another format to one of these formats.

Note: You must use the RLC format if you wish to use the Raster Transformation function.

When you start the routine you will be shown the following dialog box to allow you to select the raster image you wish to insert into the drawing.

Note: You can currently only bring in two raster images at the same time.

Once selected you will be asked where to Insert the image, the Scale to insert it as, the Color of the image, the DPI and the rotation factor.

These questions are asked on the **Properties** dialog box.

Once you have entered the correct information, your raster image will then appear on screen.

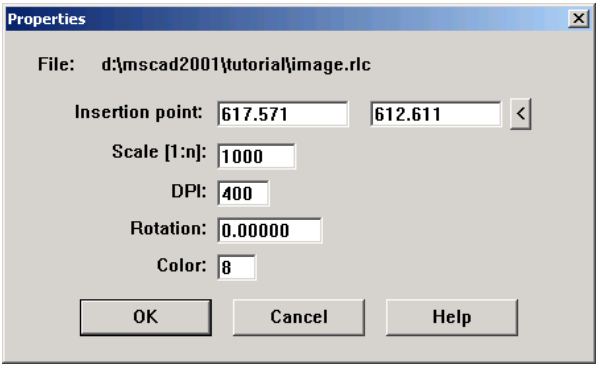
Command: RLOAD

Remove Raster Image

This option will remove the selected image from the drawing and optionally allow you to save any changes to the properties of that image.

Edit Raster Image Properties

Brings a dialog box up and allows you to change any of the fields. The changes are only done in the current drawing and will not affect the original image on the hard drive.



Insertion point: Location to insert the image on your drawing.

Scale [1:]: Sets the relative scale factor to match your drawing.

Scales the raster image larger or smaller.

DPI: Dots Per Inch. Affects the resolution of the Image.

Rotation: Affects the orientation of the image in reference to the drawing.

Positive rotation is counterclockwise, negative rotation is clockwise.

Color: Sets the color the image will take on when inserted into a drawing.

The whole image is the same color.

Each raster can be brought in with it's own color.

Command: RPROP

Save Raster Image and Properties

This function allows you to save all of the changes to the image, done by the edit image properties item, back to your hard drive.

Zoom Raster Extents

This command allows you to zoom to the extents of the raster image(s). If you have lines drawn beyond the extents of the raster image(s), you will need to use the standard zoom and pan commands to augment this command. Lines beyond the extent of the raster image(s) will not be zoomed to with this command.

Move Raster Image

This command moves an raster image on your screen. If you have more than one raster image in the drawing, you will be asked to select the one to be moved.

Rotate Raster Image

You can rotate any raster image to match your vector drawing by using this command.

Mirror Raster Image

This command allows you to mirror image the image of your choice.

Copy Portion of Raster Image

This command lets you copy a portion of or the whole raster image on screen by putting a window around the portion to be copied. This is similar in methodology to the Zoom Window command.

Cut Raster with Polyline Boundary

If you have a portion of the raster that you do not want and you wish to remove it from the job you can use this command to do so. Note: this command only works with RLC images.

Options

Raster Frame On/Off

The rectangular frame border around the images can be turned off or on by using this command.

All frames are affected at the same time. When on, the frame shows the outer edge of the image and is in the same color as the image.

Command: RBORDER

List Info on Raster Images

Below is a sample of the information that is displayed:

```
(("C:\\RLC_Images\\MyImage.rlc" 7521 7672 (10.000000 20.000000)
250.000000 400.000000 0.000000 5))
```

If there are more than one raster images on screen then all of them will be listed at the same time, in the same format as above.

An explanation of the above format:

`"C:\\ RLC_Images\\MyImage.rlc"`	Name of Raster Image
`7521 7672`	Original Origin on Raster
`(10.000000 20.000000)`	Insertion point for Raster Image
`250.000000`	Scale Factor of Raster Image
`400.000000`	Dots Per Inch
`0.000000`	Rotation Factor
`5`	Color of Image on Screen

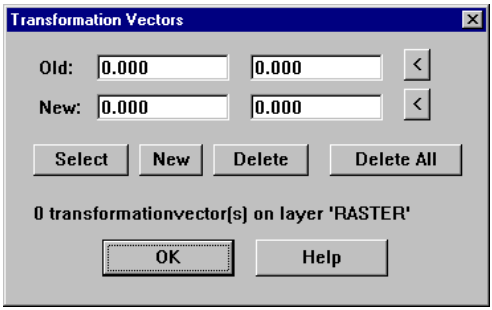
Command: (rinfo)

Transformation Vectors

Note: this command only works with RLC images.

This dialog box allows you to enter and edit the transformation vectors required to adjust the raster image to match the vector drawing.

The vectors you define will be drawn on the layer RASTER and you can actually draw the vectors before starting this command and it will detect them for you.



Old Vector Coordinate

This is displaying the current starting location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

New Vector Coordinate

This is displaying the current ending location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

Select Button

This button will let you select which vector you are currently editing by allowing you to pick that vector on screen.

New Button

This button will create a new vector from the coordinates displayed in the OLD and NEW fields above. It will duplicate a vector if you pick this button while old values are in the fields above but you can still

manipulate those vectors by using the two pick buttons to the right of the NEW and OLD coordinates displayed.

Delete Button

This button will delete the currently selected vector, both from the routine and the screen.

Delete All Button

This button will delete all the vectors on the layer RASTER.

Raster Layer Display

This location displays the number of vectors detected on the layer RASTER**.**

Note:

Do NOT use this layer for anything else or you will confuse the program.

Pick Button for New Coordinate

This button allows you to pick a new location for the vector to end. This is the ending location of the transformation for this vector. You can pick any where on the screen as the original coordinate for the vector.

Pick Button for Old Coordinate

This button allows you to pick a new location for the vector to start. This is the starting location of the transformation for this vector. In other words the amount of transformation will be determined by the start and end of the vector. The amount of transformation is the delta X and delta Y of this vector. You can pick any where on the screen as the starting (old) coordinate for the vector.

Command: RVEKTORS

4-Point-Transformation

Note: this command only works with RLC images.

This simple transformation allows you to stretch the raster image to match your vector line work.

You are required to enter 4 vector translations in the edit/define transformation vectors menu.

Then when you select this option the stretching of the raster will be done. Make sure you enter exactly 4 transformation vectors as this routine will not run with more or less than four!

$x' = a0 + a1 * x + a2 * y + a3 * x * y$

$y' = a4 + a5 * x + a6 * y + a7 * x * y$

Once the transformation is completed you will be prompted for the properties to confirm the insertion point color, etc.

Command: RTRANS

Help Menu

MicroSurvey Training Movies

We have put together many short movie clips that show you how to run certain commands and how to link these commands together to complete a task.

The movies can be found under the Help menu. Simply click on any movie from the pull-down menu to start it.

PRESS the SPACE BAR to pause or stop any movie clip.
The movies were recorded at a video resolution of **800x600** so to view them correctly your video resolution should be the same or higher.

We now have almost 6 hours of extremely useful examples to watch. Most of these are only a few minutes long. There should be something for everyone to help you learn more about the program.

Keep in mind that some of the options, menus and dialog boxes may have undergone a few minor changes since the movie clips were generated. None of the changes should stop you from following the steps required to produce a final result as shown in the movie clips!

Tutorials

Several tutorials are included in this help system. The fastest one to introduce you to MSCAD is the Data Collector Tutorial.

Data Collector and Contours Tutorial

The Data Collector Tutorial covers loading a data collector file, annotating the points, elevations and descriptions, drawing 3D break lines based on the descriptions and building and viewing a 3D surface model. Concluding with a full contour plan, labeled and smoothed.

Manually Entered Traverse Tutorial

The Manually Entered Traverse Tutorial covers field note entry, coordinating the traverse, computing closure, traverse balancing and listing the result.

Property Line (COGO) Tutorial

The Property Line Tutorial covers calculating property boundaries, and locating buildings on three lots using COGO.

Earthwork Volume Tutorial

The Earthwork Volume Tutorial covers importing of a data collector file, editing the traverse file, using AutoMAP and the Graphic Editor to manipulate points and draw linework, creation of surfaces in Modeling, calculation of a volume between the surfaces and finally creating cross sections across the entire job to confirm the volume via the hand method people are used to using.

Simple Road Design Tutorial

The Simple Road Design Tutorial starts with a contour plan and a few points. You create a working surface and then enter in a horizontal alignment and add curves and spirals. A profile is generated from this and then a design vertical alignment is created. After making a cross section template, we pass this template along the alignment and create a new surface. Once this surface is generated we then extract cross sections from along the final alignment. These cross sections are added to the drawing for final presentation.

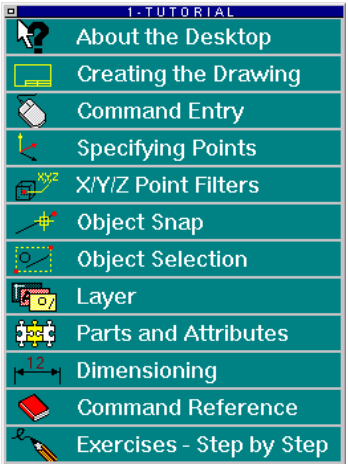
Lot Design and House Placement Tutorial

The Lot Design and House Placement Tutorial steps you through entering and defining a lot from lines and curves, designing a house, defining the house and saving it into the house library and then placing the house onto the lot. Then place stakes and generate staking reports.

CAD Tutorial

For users who are keen to learn more about the MicroSurvey CAD engine, look over the CAD Tutorial. Keep in mind that these topics relate to the CAD functions only and do not make any direct connection with MicroSurvey specific commands.

Clicking the CAD Tutorial button from the Help menu will bring up the following palette menu.



CAD Exercises

The exercise part of Tutor contains drafting exercises illustrating the basic commands of MicroSurvey.
It provides a continuous course of instruction with learning sequences structured by degree of difficulty.

How to do an exercise

To organize the steps of an exercise - starting with an exercise selection up to ending with the evaluation of the drawing - a palette as illustrated below is provided:

This palette is displayed automatically if you choose, within the tutorial palette, the item *Exercises - Step by Step* or enter the command EXERCISE.

The symbols in this palette have the following meaning:

 Dialog to select a new exercise

 Exercise task and assistance for the selected exercise in a dialog box

 Evaluation of the user's drawing

 Task and assistance for the current exercise in the Help window set always on top of the desktop

 Parameter setting for the evaluation program

Exercise Assistance

The following types of assistance are provided for each exercise:

- Graphically illustrated task
- Description of the task
- Sample solution presented in a demo slide show
- Help for the commands that you are practicing throughout the drawing

You can call these features to support you any time when you are doing an exercise.

When you have finished the exercise drawing, you can start the *Evaluation Program*, which tests the drawn entities on validity. This control program recognizes faulty, missing and superfluous elements.

Any faulty or superfluous entities will be highlighted.

You can configure the evaluation program. For example, you can disable the checking of certain object properties (like layers).

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