XVIII. SurvCADD Module

SurvCADD file Extensions and layer names:

SurvCADD uses certain files and extensions in the inner workings of the program. Some of these files are Windows standard such as txt, asc and dxf. Others are unique to SurvCADD; crd, ctr, grd, lay, pro, sct and mxs. During this portion of the training you will become familiar with each type of file and its use.

The first set of files or the Windows standard files are generally used as input into SurvCADD. SurvCADD will allow you to bring information in by text files with txt, asc, dat, cr5 and csv extensions. The text extensions are an ASCII files and are used for formatted data such as a data collector or xyz file. The original file can come from a data collector, spreadsheet, database or user generated text editor.

The **dxf** file extension (**D**igital **X**fer **F**ile) is used for file import/export to transfer line work from one application to another. For example you can export and AutoCAD/SurvCADD drawing to dxf format and import it into Corel Draw or ArcView. Conversely you can import a dxf created by another application into AutoCAD.

The **crd** extension is used for coordinate files. These are binary files created by SurvCADD to hold point and coordinate data. Unlike the txt or ASCII files, these can only be edited inside of SurvCADD with the coordinate file utilities.

The **ctr** extension is a default extension created by the contour menu. This extension is not necessary for SurvCADD to operate and can be changed either in the contour menu or in the configure SurvCADD menu.

The **grd** extension is used by the gridding module. All grid files must have this extension for proper SurvCADD operation. This extension is automatically added to a grid file when it is created. The grd file is an ASCII file but should not be edited except by grid file utilities within SurvCADD.

The **lay** is used by the Save Layer State command and allows the user to store a unique view of a drawing with various layers frozen and thawed.

The **pro** extension is used by the profile module. When a profile is created the pro extension is automatically used.

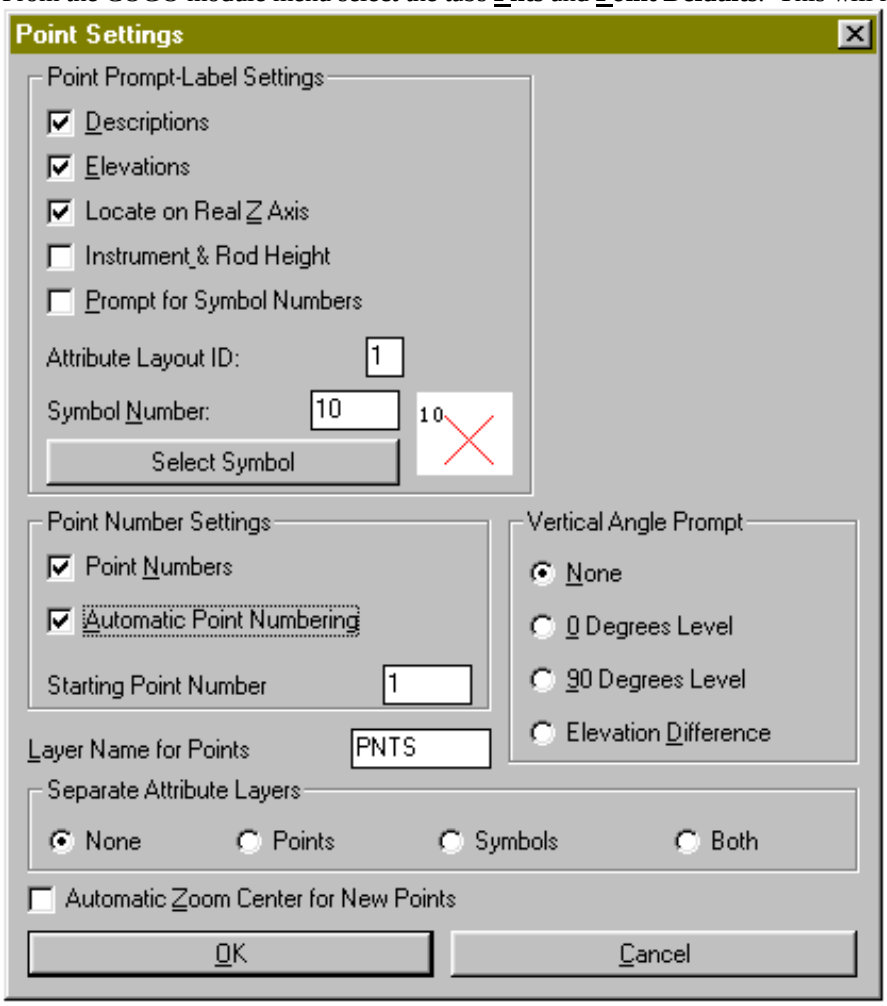
The **sct** extension is used by the section module. This extension is automatically added to section files.

The **mxs** extension is used to designate a section centerline. The extension is automatically used when a section centerline is created.

SurvCADD Points

This section of the course deals with basic SurvCADD commands and an understanding of SurvCADD's basic unit which is a SurvCADD point. These points are different from AutoCAD points in that a SurvCADD point is a block with many attributes not just the xyz of an AutoCAD point. The SurvCADD point stores not only the xyz location of the point but also and description, point number, elevation text and point symbol.

From the COGO module menu select the tabs **Pnts** and **Point Defaults**. This will result in the following pop up.



The settings in this window allow you to control the appearance and data stored in the SurvCADD point. The first five boxes control the information gathered other pop up windows. Generally descriptions, elevations, and locate on real z-axis should be checked. This is the basic information that needs to be stored in the point. The Instrument & rod height are used for field note reduction, which will not be covered in this section. Prompt for Symbol Numbers allows the operator to change the symbol type on the fly otherwise the default point is used. Attribute layout ID is used to control the location of the text around the symbol, for more info see reference manual.
The next window and button allow the operator to change the symbol for the point. The default is 4 or a circle. Select the button Select Symbol and change the symbol to 10 or a cross.

The Point Number Settings area allows the operator to choose whether to number the points manually, automatically or not at all and to choose with what number to start.

The section labeled Vertical Angle Prompt will be ignored at this time for more information see the help tutorial.

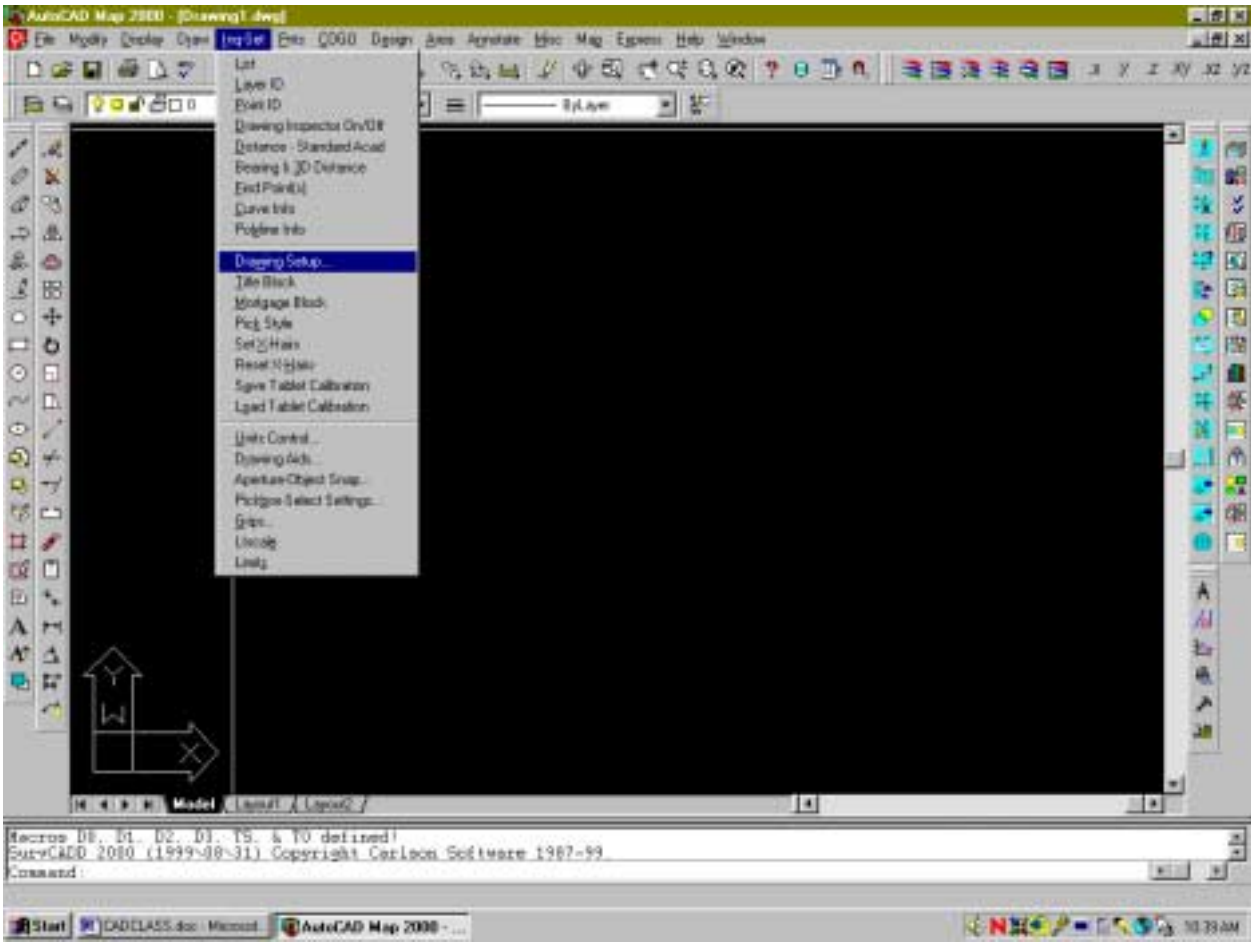
The points can be assigned to different layers, the default layer is PNTS. For this exercise we will allow the default of PNTS to remain.

This setup window also allows you to place the symbol and the point on separate layers. This allows you to turn off the dot of the pint in favor of the symbol. For now leave the default of none.

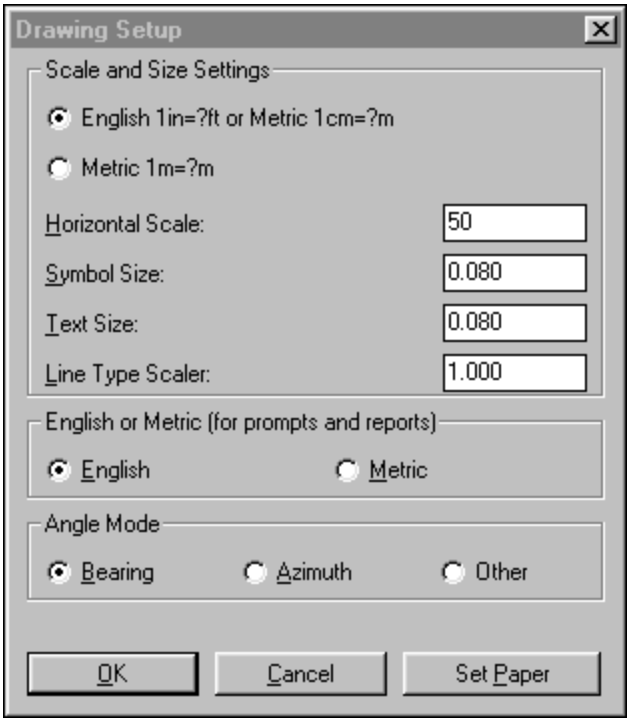
The last setup on the box is to automatically zoom to the center of the point, again for this exercise we will let the default of not checked be our choice.

When you are finished you setup box should look like the one above.

Drawing Setup



To assist the user in selecting text sizes and line scales, SurvCADD has a unique command called Drawing Setup. The command can be access from any SurvCADD menu under **Inq Set**. After selecting the command the following dialog box will appear.



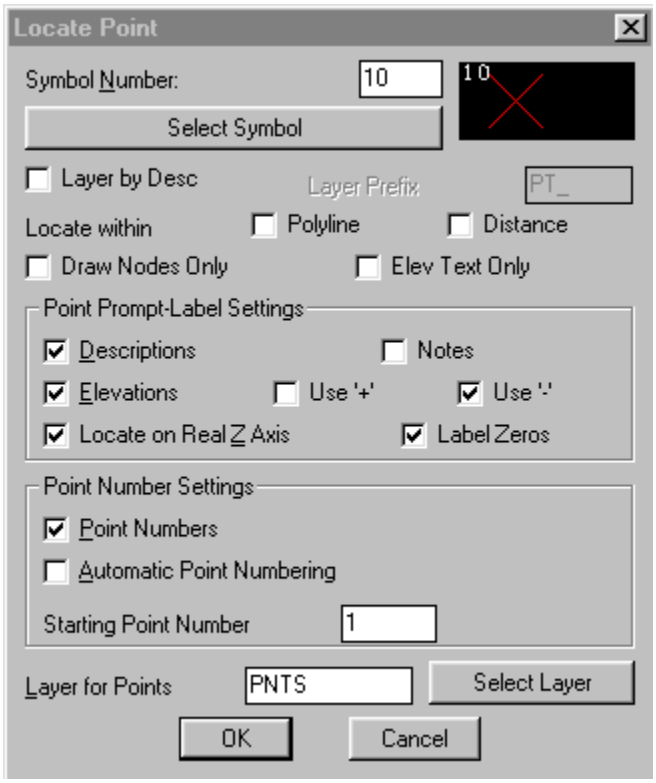
What this box allows the user to do is to choose a map scale and the size the text and symbols will be in the plotted version of the map. SurvCADD will adjust the appropriate variables so the user does not have to. The symbol and text sizes are in plotted inches.

Inputting Points into SurvCADD

There are several methods to get xyz data points into SurvCADD. We will explore two methods in this section. First we will input the points by hand within the SurvCADD module. We have already set up the point defaults.

Exercise A

From the COGO module select Pnts ➔ Draw/Locate Points and the following pop up will ask for information. This looks very similar to the Points Default menu and items like symbol can be changed in this screen. For this exercise we will take the defaults as shown so select OK and continue.



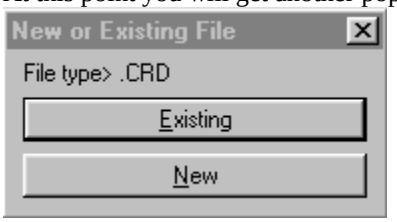
The following command is shown on the command line.

Enter coords/All/<Pick point or point numbers>:

For this exercise you will type in E for Enter coords.

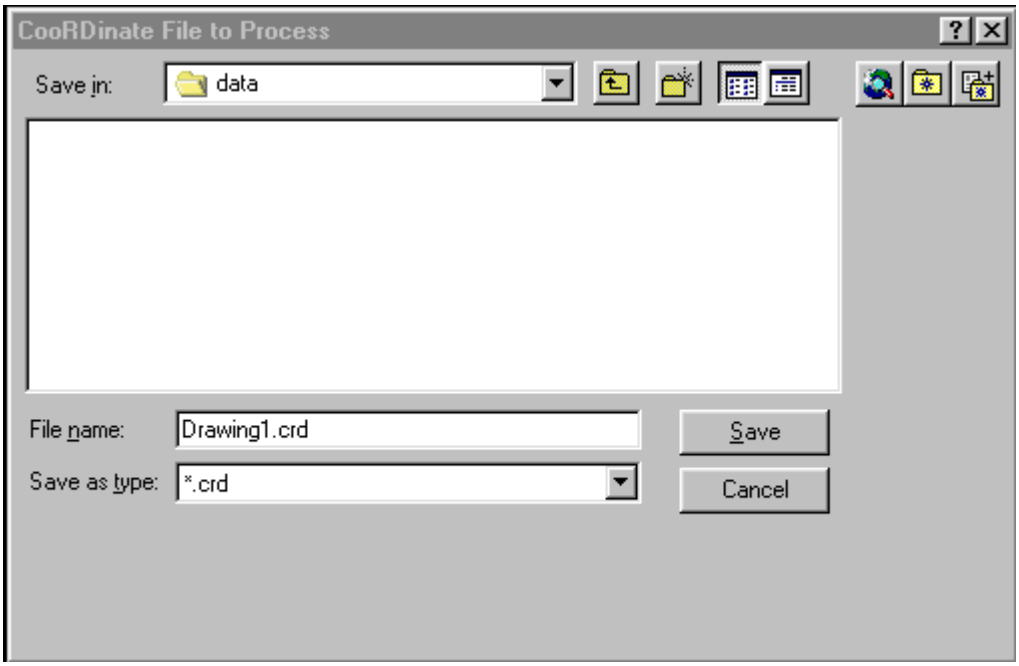
You will then be asked for a North (y). After you type in a number corresponding to the north coord (10000) and hit enter you will be prompted for the East (x). After typing the number for the East coord (10000) and hitting enter you will be prompted for the elevation. After typing the number for the elevation (400) and hitting enter you will be asked for a point description. This can be any alpha/numeric (ex. Shot, hub, top, etc). After typing the description (start) you will be prompted for a point number. The default number will appear, to use this number simply hit enter.

At this point you will get another pop up asking about what coordinate file to use.



For this exercise choose the New button.

You will then see a pop up asking for a file name. The default name is the drawing name with a crd file extension.



For the exercise choose the default by selecting the Save button.

You should see a point on the screen and the command will prompt for another Northing. At this point continue to enter the following points:

North	east	elev	description
10101	9000	395.45	shot
10203	8545	377.56	base
10250	8903	388.79	shot
10054	10234	412.99	top

When you are finished typing hit an enter or esc to end the command. You should have five points on the screen. You may have to zoom extents to see them all.

The second method of entering points is by converting a txt file to crd file.

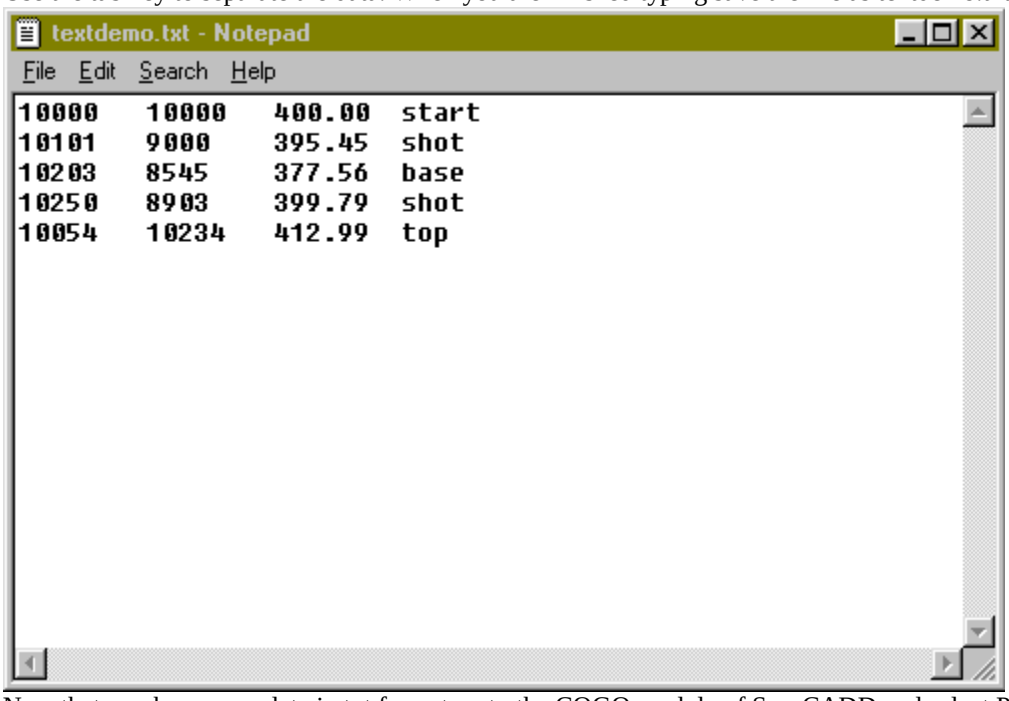
Exercise B

Txt file to Crd file

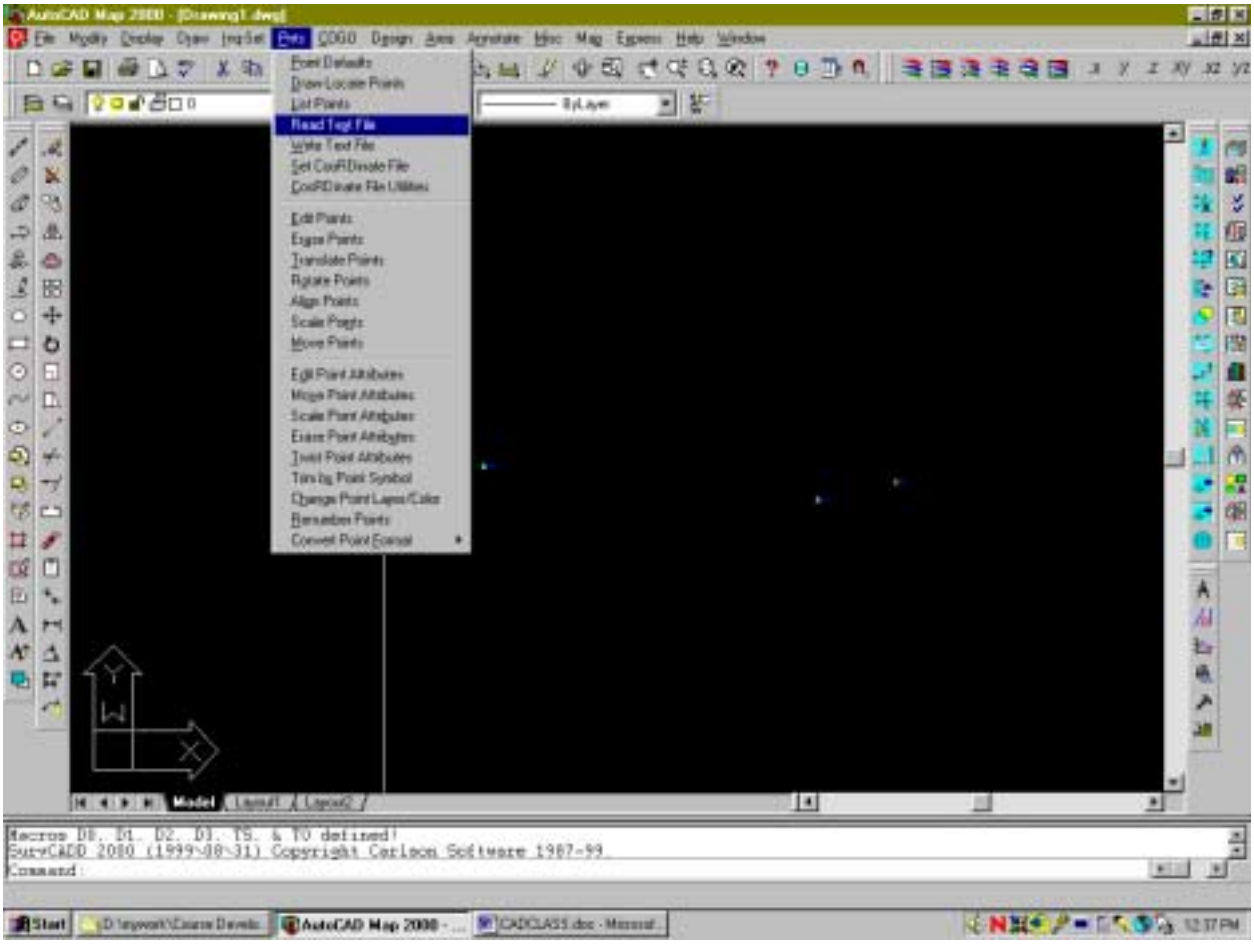
For example you have survey point data in hard copy format as follows:

North	east	elev	description
10000	10000	400.00	start
10101	9000	395.45	shot
10203	8545	377.56	base
10250	8903	388.79	shot
10054	10234	412.99	top

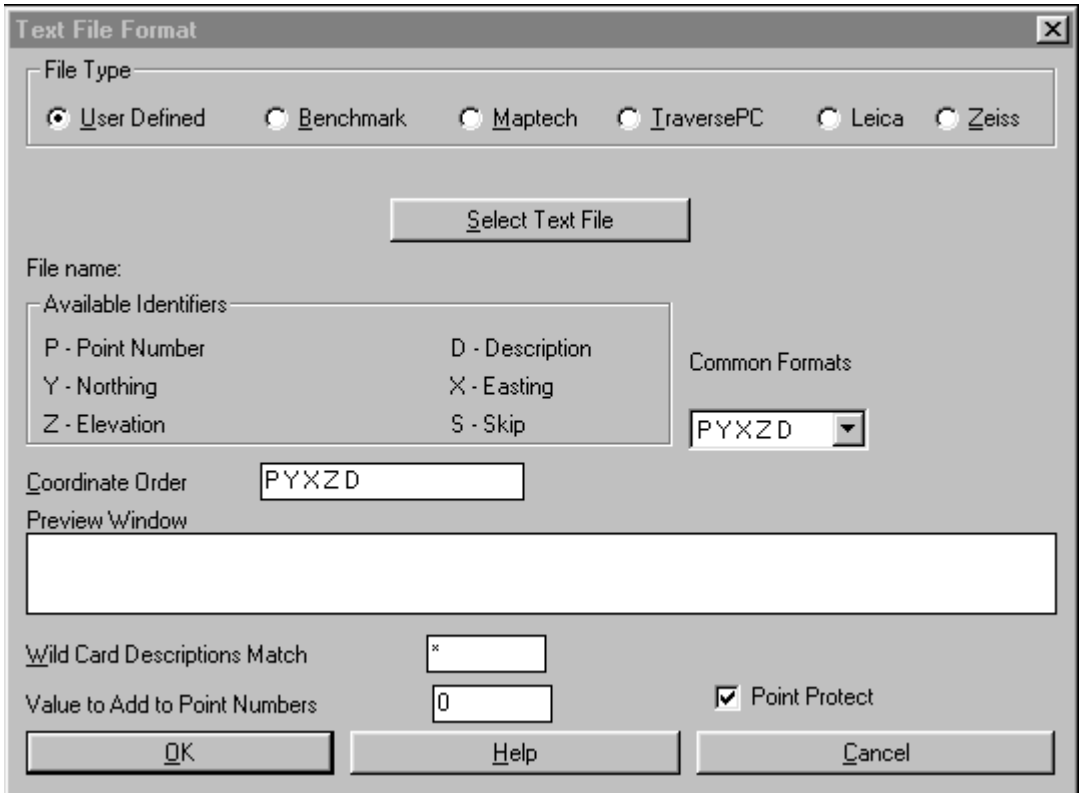
Start up a text editor such as Notepad and type in the above text, don=t type in the header only the numbers and text. Use the tab key to separate the data. When you are finished typing save the file as textdemo.txt.



Now that you have your data in txt format go to the COGO module of SurvCADD and select Pnts and Read text file.



You will be prompted for a crd file, select new and give the file a name of textdemo.crd and hit OK. You will be prompted with the following menu.



At the top there is a box called File Type. There are several options that allow files from different sources. At this point we will be using User Defined. Next you will select the button labeled Select Text File and you need to select the file textdemo.txt and select the open button.

Next you will need to choose the format for your data. Remember this data is in Northing, easting, Elev, Description, so go to the common formats button and choose Y X Z D as the format. Y stands for Northing, X stands for Easting, Z stands for elevation and D stands for description.

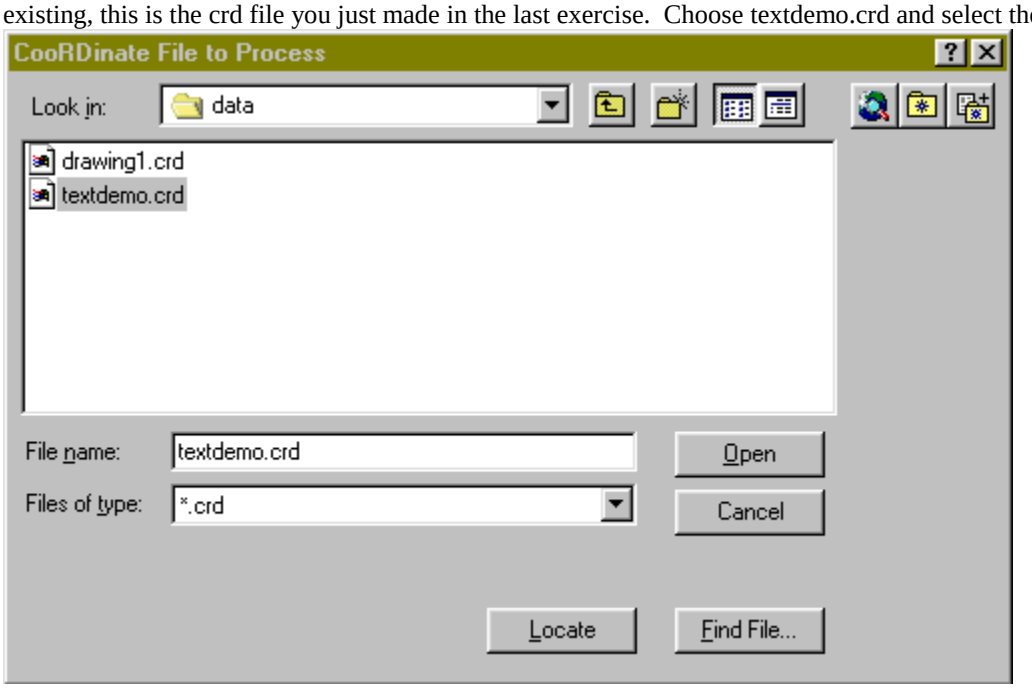
When you are finished your screen should look like the one above. User Defined should be dotted, File name should be textdemo.txt, and Format should be Y X Z D. Choose OK and the command should finish with done. You have now changed a txt format to crd format.

Next you will need to change the Crd file to SurvCADD points.

Crd file to SurvCADD Points

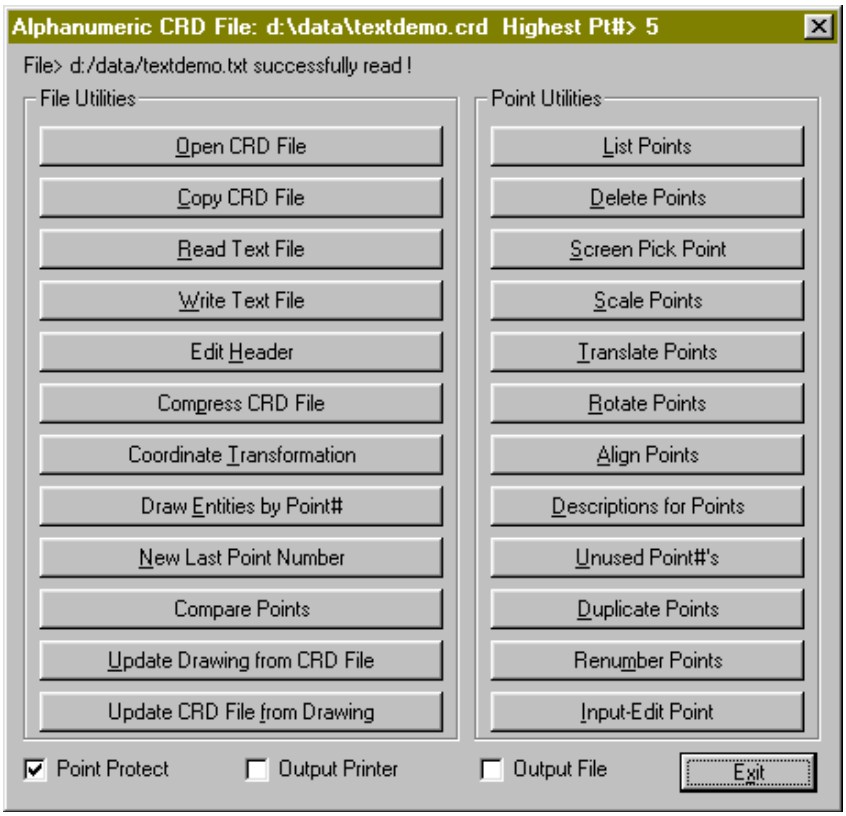
Exercise C

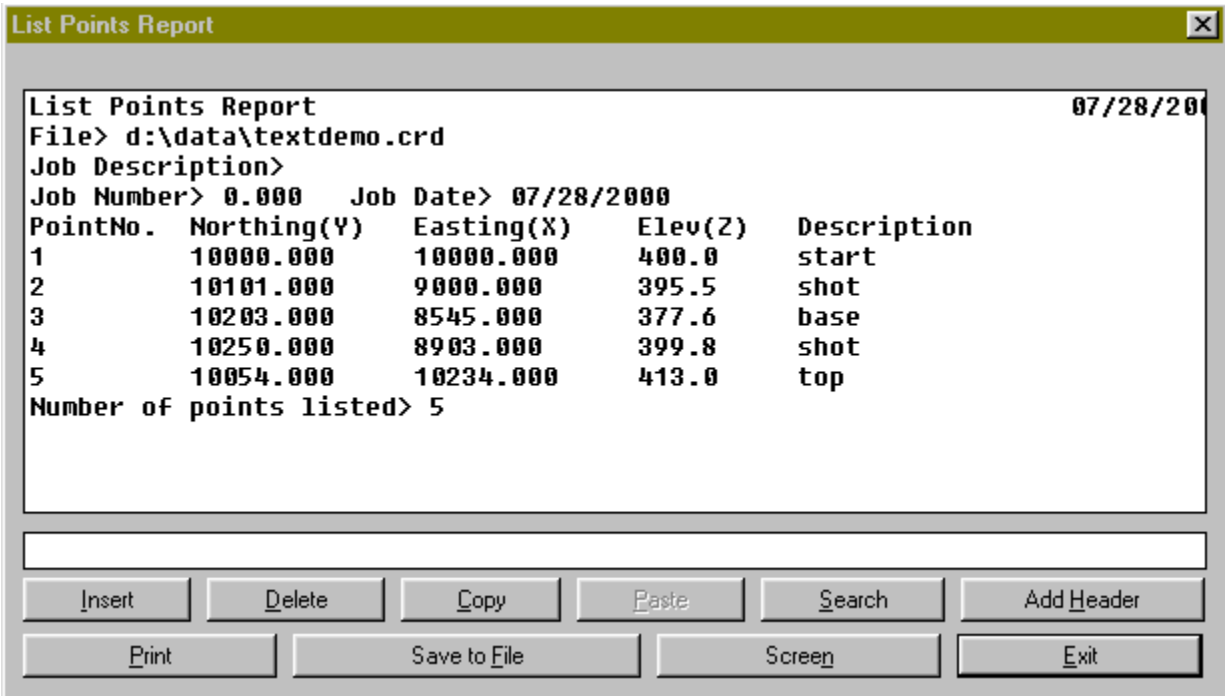
From the COGO module choose Pnts and CooRDinate File Utilities. You will be prompted for a crd file. Choose existing, this is the crd file you just made in the last exercise. Choose textdemo.crd and select the Open button.



The following menu will pop up:

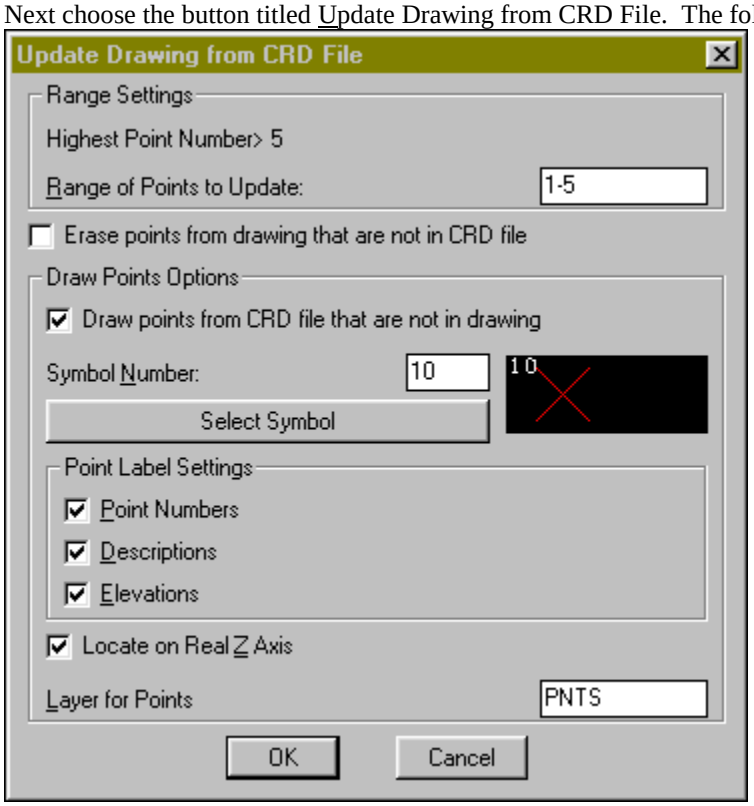
You will see on the top that it is reading from the testdemo.crd file and that there are five points. Choose the List Points button and you will see a pop up requesting info. Select the OK and take the defaults. You will then see the points as they exist in to crd file.





Choose the Exit button after you are finished looking at the file.

Next choose the button titled Update Drawing from CRD File. The following menu will appear:



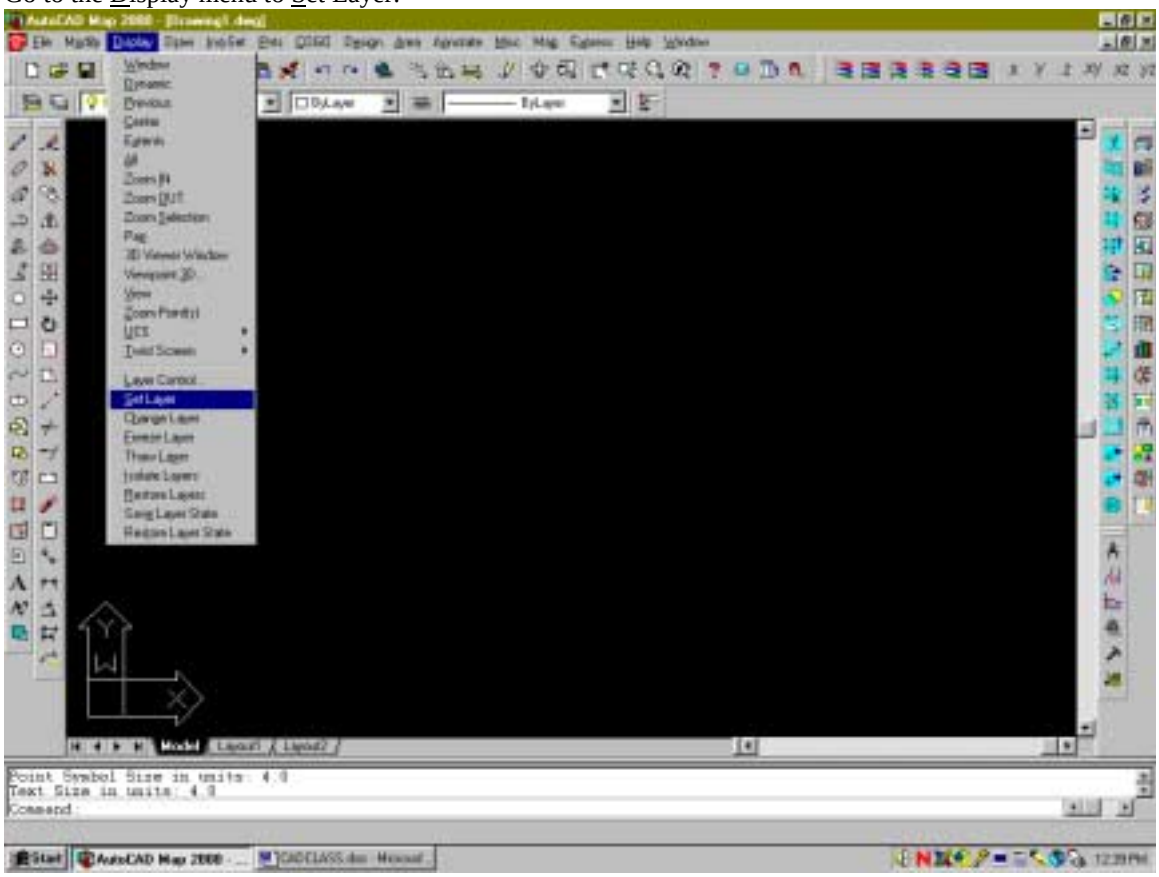
The Point range is 1-5, which is all five points. Check the box where it says Draw points from CRD file that are not in drawing. Select OK and the points will be placed in the drawing.

Layer Management

SurvCADD has some unique layer management tools. These command tools allow the user to manage existing layers without having to use the Layer command or use the Layer dialog.

Exercise

Go to the Display menu to Set Layer.



This command allows the user to change the current layer to the layer of the layer of the object chosen. For example the current layer is 0 and you want to add on to a ditch. You would choose the Set Layer command. You will be asked to "Pick entity with layer to set:" When you pick an entity on the ditch layer the current layer will be set to ditch.

The Change layer command allows you to change the layer of an entity. Choose the Change Layer command and you will be prompted to choose an entity(s). After you pick the entity(s) you will be prompted to either type in the layer name you want to change to or you can pick an entity that is already on that layer. Either way the entity(s) chosen will be changed to the new layer.

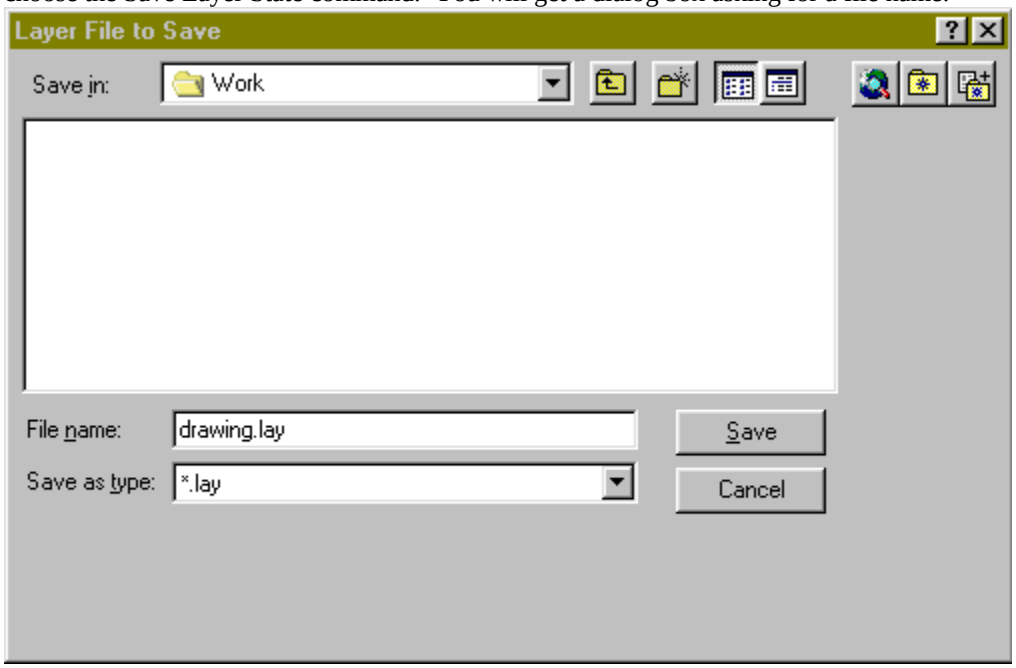
The Freeze Layer command allows the user to selectively freeze layers. You will be prompted to choose entities on the layers that you want to freeze. For example you want to freeze the contour layers so that you can view the other entities easier. You would choose the freeze layer command. Pick an entity(s) on the layer(s) that you want to freeze and hit enter. These layers will be frozen and not displayed.

The Thaw Layer command works in tandem with the freeze layer command. After a layer(s) is frozen, in order to bring back the layers frozen with the freeze command, the user picks the thaw layer command and the entities are unfrozen and displayed. This command only works after a freeze and only on the last freeze. It cannot be nested with previous freeze commands. It works as a toggle for rapid viewing and editing.

While freeze works only on layers chosen the Isolate Layer command works more globally in that it freezes all but the layers picked. For example you want to see only the ditch layer. You would choose the Isolate Layer command and pick an entity that exists on the layer ditch. You will be asked if you want to retain the POINT layers. The default is No. The only time you would take a Yes is if you wanted to see a layer associated with a SurvCADD point such as Pntelev. If you want to isolate the point elevations and choose No at the prompt to retain POINT layers then when the Layers are frozen the point block will also be frozen and even though Pntelev is thawed no elevations will be displayed. When you choose Yes at the prompt the Point block will not be frozen and the elevations will be displayed.

The Restore Layer command works with the Isolate Layer command such as the Thaw command works with the Freeze command. After an Isolate the Restore is used to redisplay the layers frozen by the isolate. Again as in the Thaw the Restore only works one time after the last isolate.

The Save Layer State command allows the user to store the current layer state to a file. This is useful if you need certain layers active for plotting. You freeze and/or thaw the layers you want to display and then choose the Save Layer State command. You will get a dialog box asking for a file name.

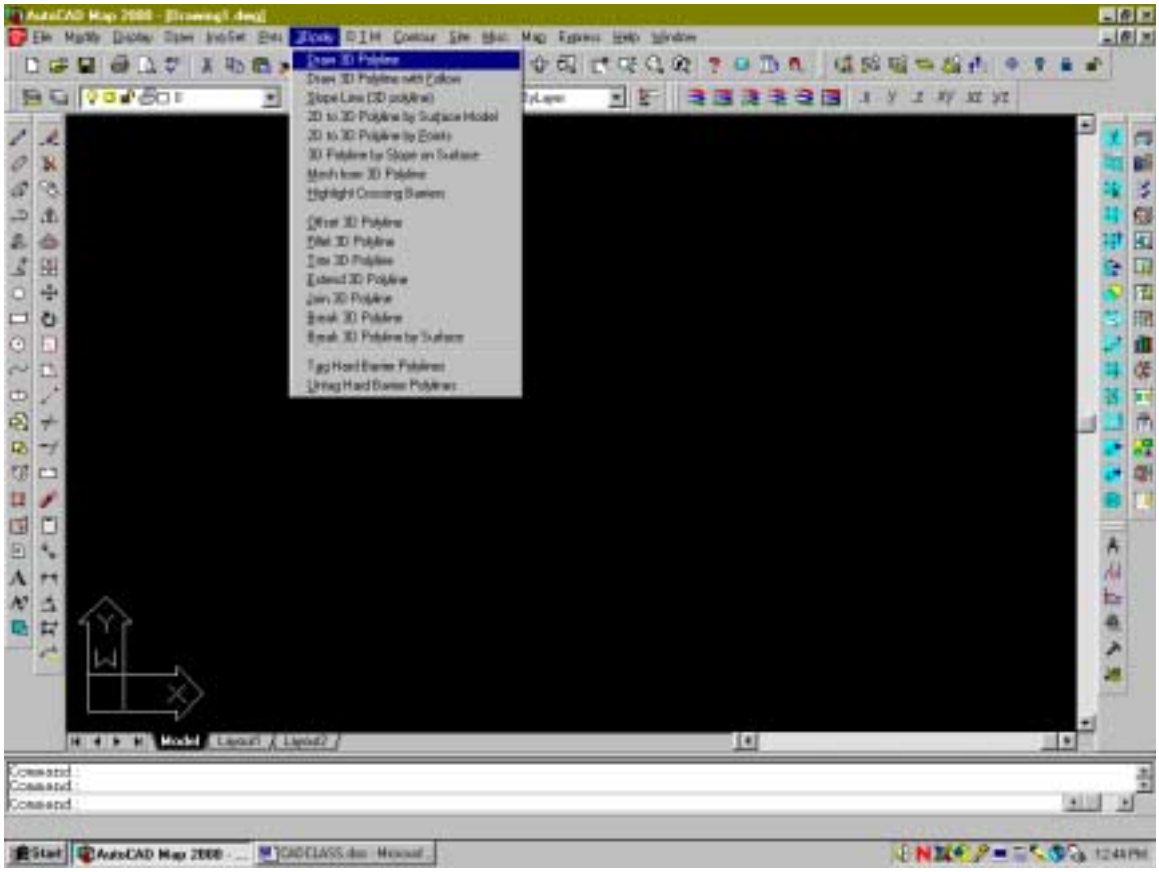


You will be prompted with the current drawing name with a lay file extension. You can either use that name or rename to describe what you are saving.

Restore Layer State. This command will restore saved layer states. In the previous command a layer state was saved. This command is used to restore the saved state.

3D POLY UTILITIES

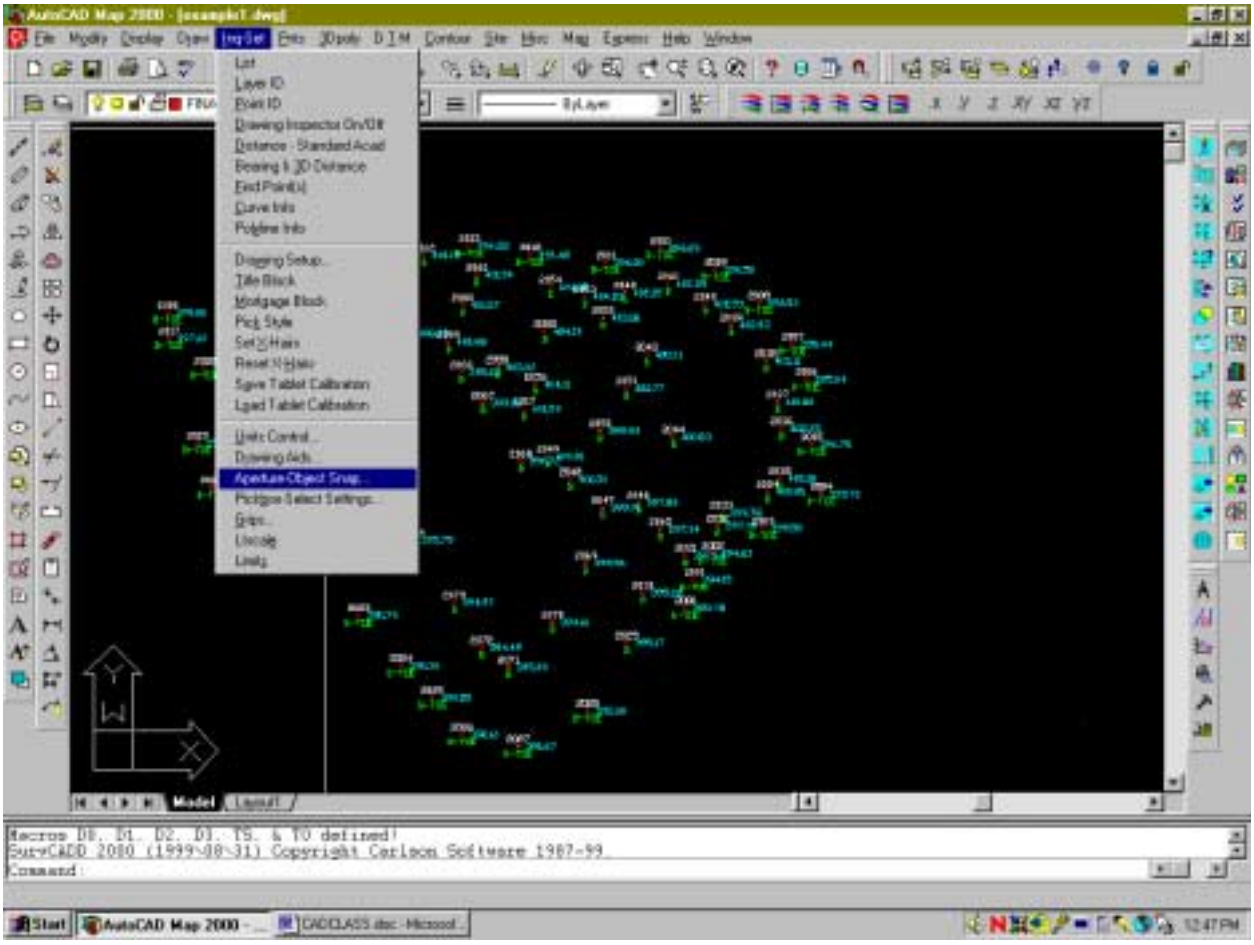
3DPOLY UTILITIES can be accessed from all menus under the modify tab, or under the DTM/Contour menu under the 3Dpoly tab. The only difference is that the ABreak 3Dpoly by surface@ command is only under the DTM/Contour menu. This tutorial will address the commands as they are found under the DTM/Contour menu.



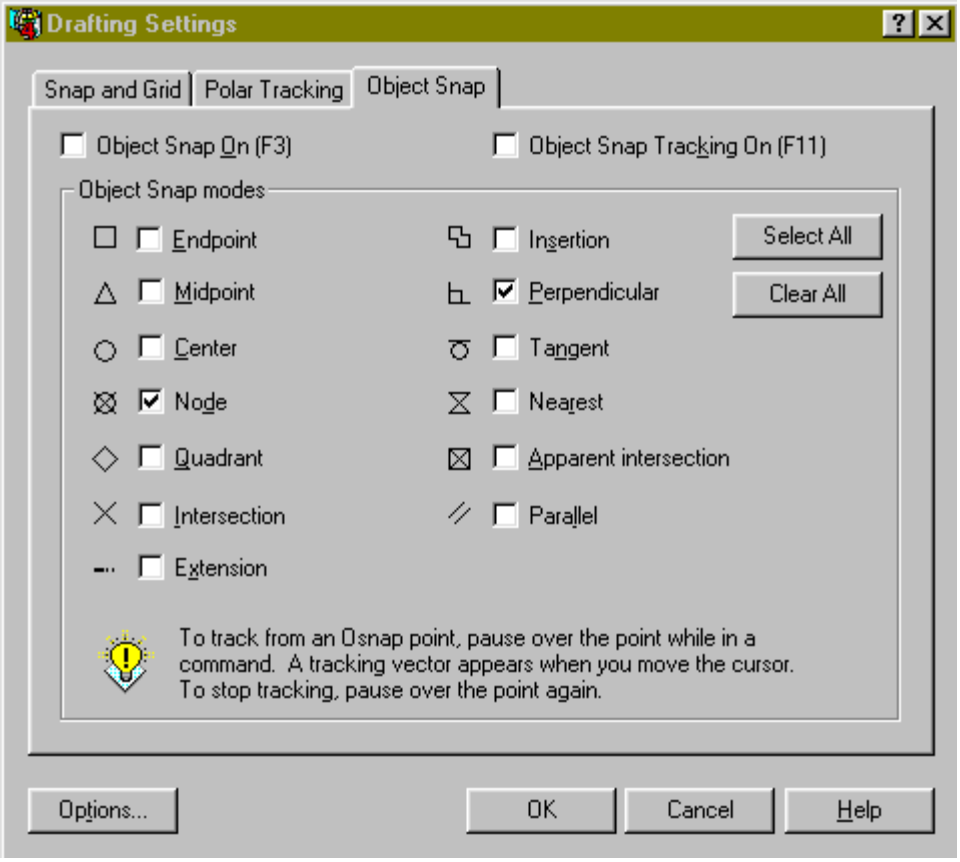
The above screen capture shows the menu with the various tab options. The first item is draw 3D polyline. 3D polyline are similar to the standard 2D polyline with two big exceptions, the first being that each vertex has its own elevation and the 3D polyline cannot have a width. 2D polylines are used for contours where the line has the same elevation through out. The 3D polyline is used for ditch and road centerlines, inclusion/exclusion perimeters, and section and profile lines. The first exercise is to build a 3D polyline using the first line item under the 3D Poly tab.

Exercise

In order to start, open the drawing titled example1.dwg found in the sc2000/work directory. In this exercise the user will draw a contour inclusion perimeter around the data points shown in the drawing. Start by changing the menu to DTM/Contour if it is not already up. Next go to the tab Inq-Set and go to the line item of Aperture-Object Snap and choose it.



A dialog box will appear and the user should choose the box marked node. This is the only box that should be checked. Choose OK and you are ready to proceed to the next step, which is to choose the tab marked 3Dpoly and pick the line Draw 3D Polyline.

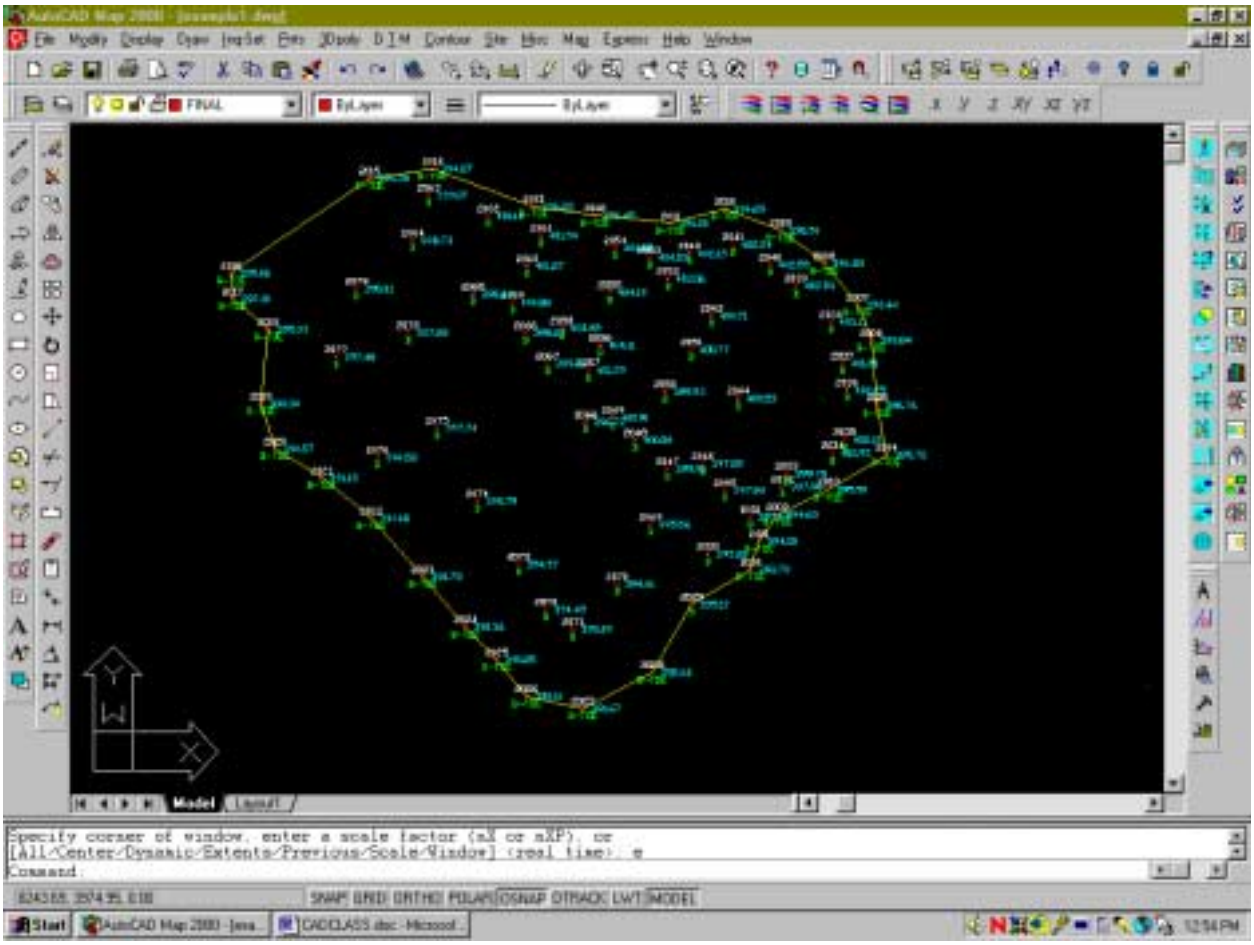


The first question asked at the command line is ALayer name for 3D polyline <Barrier>?@. Barrier is a default layer set up by SurvCADD. Generally 3D polys are built on this layer because SurvCADD will recognize these lines as Abarriers@ in contour generation. For this example hit an Enter to take the default choice.

The next question asked is APrompt for elevations (.XY filter) (Yes/<No>)?@. The question is asking if the user wants to supply the AZ@ or elevation every time that a point is selected. Generally when the user is building a 3D Poly the points in the model will supply the AZ@ value. That is why the object snap was set to node. When a node is picked SurvCADD will assign the X,Y and Z of the point to the poly vertex. At this point either hit Enter for the default value of No or type N then hit Enter.

The next question is AUse surface model from file (Yes/<No>)?@. During the draw sequence the user will be asked to pick a point from the screen or supply a SurvCADD point number. If the user is working totally from the screen the answer to the question is No. If however, the user wants to type in the number sequences the answer is Yes and the user will supply the CRD file that contains the point data. It has been the experience that selecting a snap filter such as node and picking points from the screen is the most efficient way of building this line. For the example the user will select No by hitting the Enter button.
The final question is AUndo/ <pick points or point number>:@. At this point the user will move the cursor until the cross hairs are over the first point. As you move near the first point you will notice that the node symbol is lit at the center of the cross hairs replacing the aperture box. When this happens the user can select the point with a left mouse click and the 3Dpoly is started. Continue around the periphery of the points selecting the nodes of the outside points. When you get back to the beginning instead of picking the first point again type in a c at the command line to close the polyline.

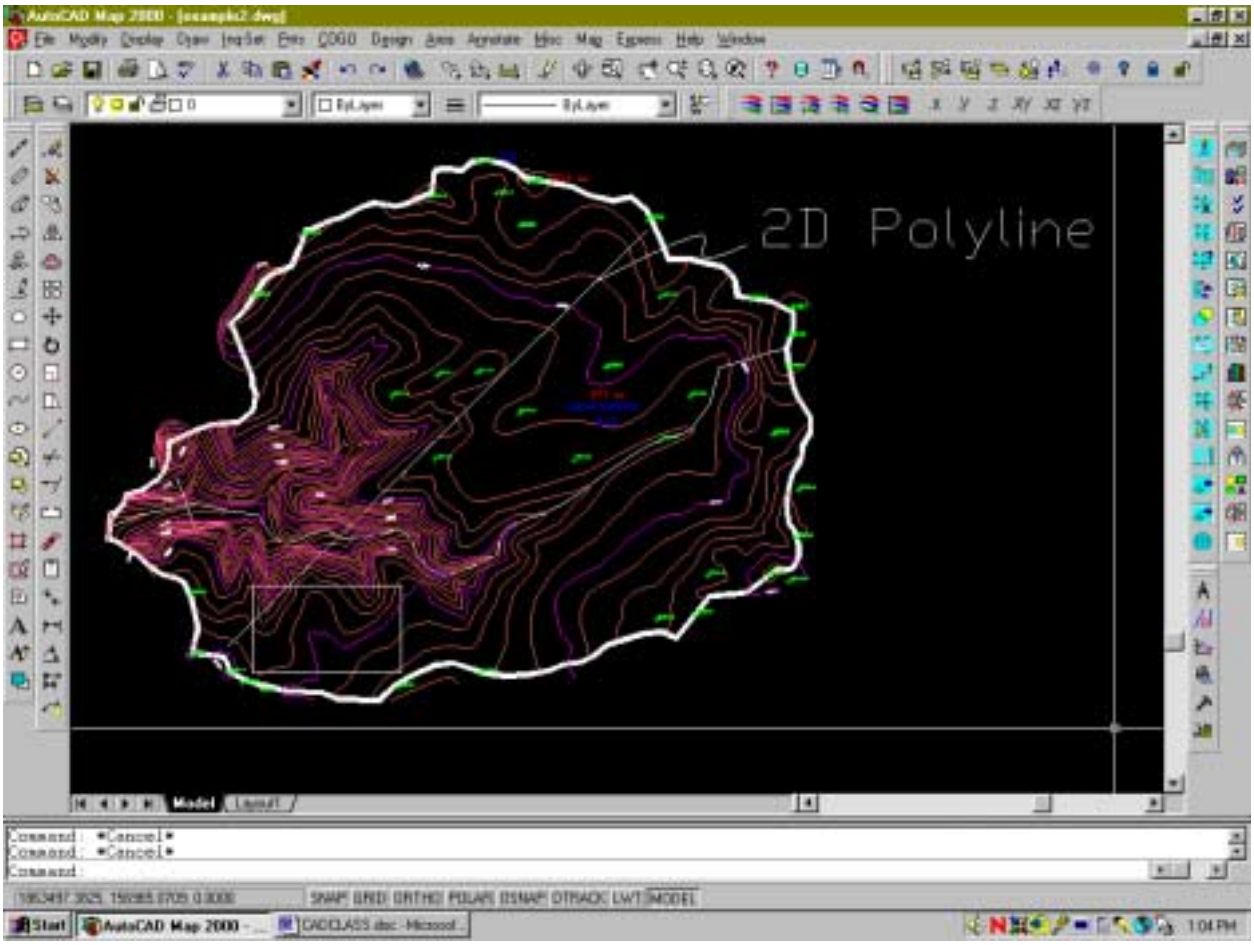
The user after closing the polyline will be asked one final question ADraw another 3D polyline (Yes/<No>)?@. Hit enter to take the default No and the command will exit. The users screen should look similar to the one below with a 3D Polyline around the points.



Valley Fill Exercise

For the next set of exercises the user needs to open the drawing SC2000/work/example2.dwg.

Exercise 2 will be to convert a 2D Polyline to a 3D polyline that conforms to the surface of the model. First draw a 2D polyline from the lower left of the drawing toward the upper right keeping the line inside of the topo lines as shown below.



Select 3Dpoly tab in the DTM/Contour and pick A2D to 3DPolyline by surface model@. This command will convert the 2D line that was drawn and make it into a 3D line conforming to the surface model contours by inserting 3D vertices at each contour crossing. The user will be asked a question at the command line ASource of surface model (File/<screen>)?@. If the user had previously built a grid file of this model that could be used to convert the line. In this example we will choose Ascreen@ and allow the computer to build the files it needs Aon the fly@ to do the conversion. So the user should hit enter to take the default Ascreen@.

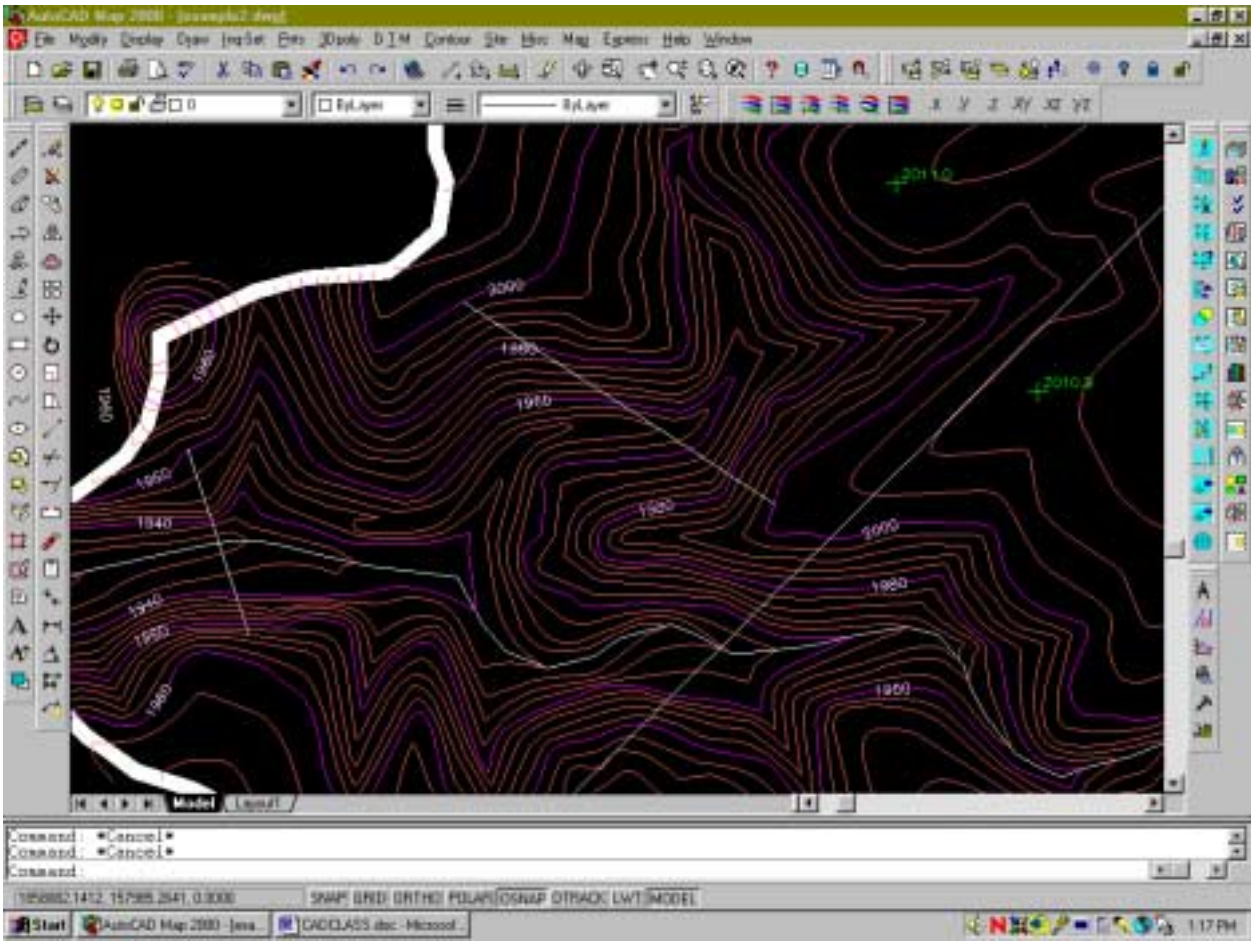
Next the question is asked to ASelect Polyline to convert Select objects:@ The user should pick the 2D poly, that was just drawn< with the left pick button. Since there is only one line hit enter after Aselect objects@.

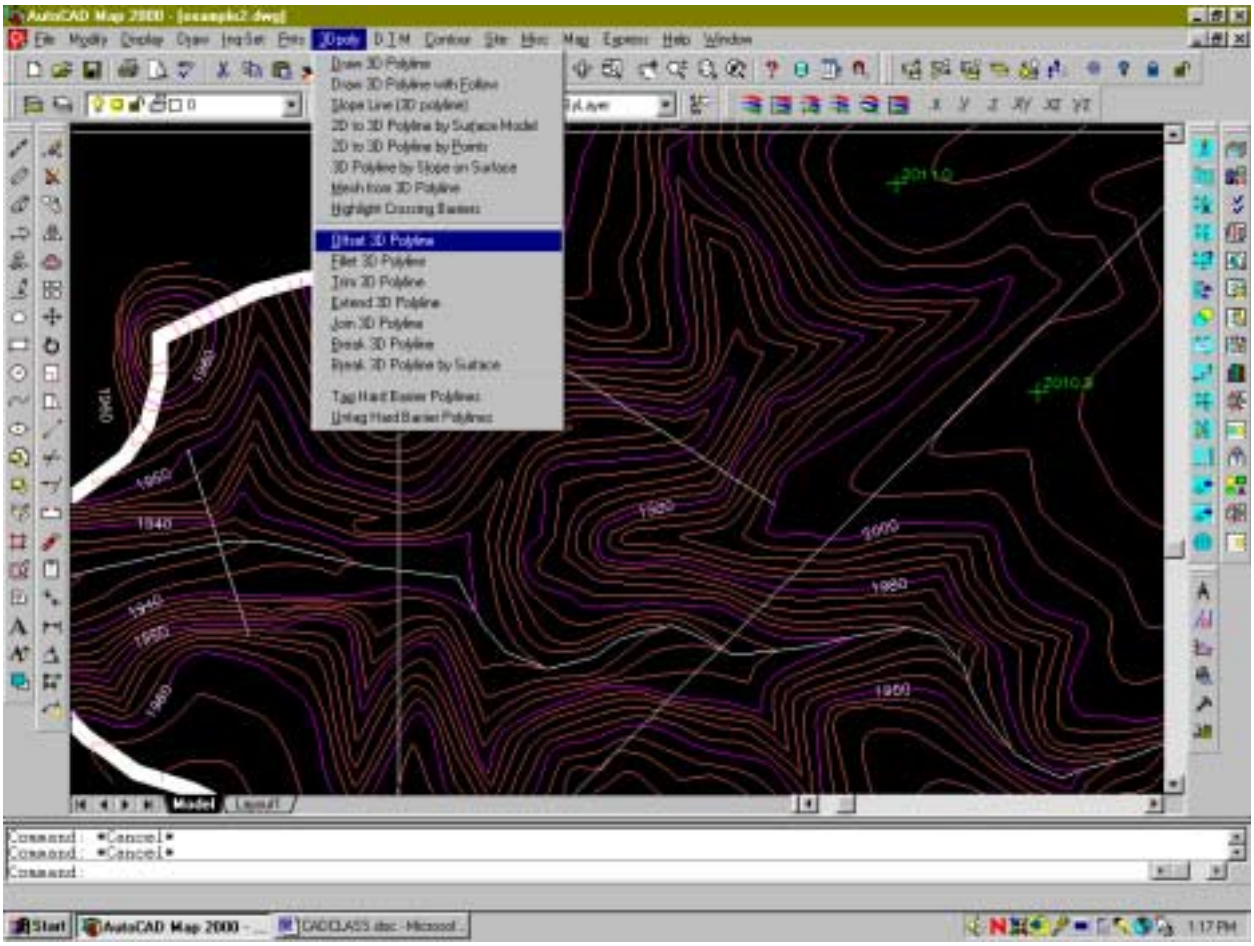
The command will prompt with another question ASelect 3D Faces, lines and polylines select objects:@ The user should select all of the contour and points on the model by using a crossing window. Any 2D lines with Azero@ elevation will be filtered out. Hit enter after selecting all surface lines.

The command will finish with AConverting polylines done@. If the user now does a Alist@ on the line you will see that the line now has 3D vertices corresponding to the surface. This command can be used with any 2D poly and can have more than 2 vertices.

3D Polylines can not be edited with the usual offset, fillet, trim, extend and break commands so there are commands provided to operate on these lines. The fillet, trim, extend and break commands work just like the 2D counterparts and will not be covered, however offset and break by surface will be demonstrated. The first on the commands is Offset 3D Polyline. This command allows the user to offset a 2D or 3D line horizontally like the standard offset command, it also extends the capability to offset in the vertical direction also.

The exercise is to build a hollow fill face on the model at the 2000' contour with a 5:1 outslope. First the user needs to draw 2D polyline on the model starting at the 2000' contour and extending across the valley to the 2000' contour on the other side, per the example. This 2D Poly will be the top of the valley fill.



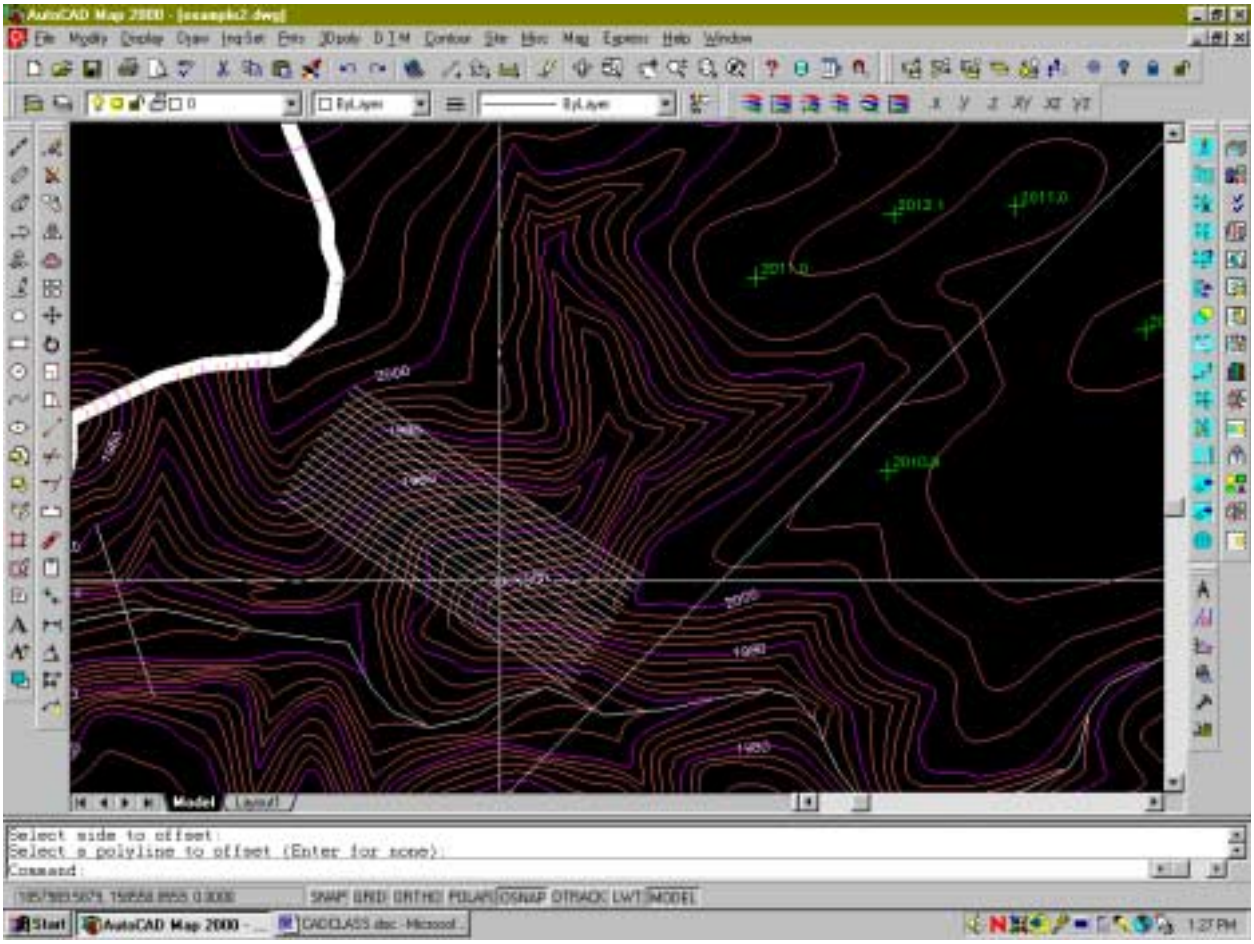


After picking the command the user will be asked the following question AEnter the offset method (<Interval>, Constant, Variable)?@. For this example the interval option will be used. The constant and variable will be left for advanced SurvCADD users. Briefly AInterval@ allows the line to be offset by an input horizontal amount and an input vertical amount. AConstant@ will offset the line by a set horizontal amount and change the AZ@ values to a constant amount specified by the user. AVariable@ allows the user to offset each segment of the polyline to be offset by different amount horizontally and vertically. For this exercise hit Enter for the default Interval.

The user will be prompted with AVertical/<Horizontal offset amount>:@. First the user will supply the offset amount. Since the existing surface contour are separated by 4' vertically we will use a horizontal offset of 20' so that the outslope will be 5:1. Type in 20 and hit enter.

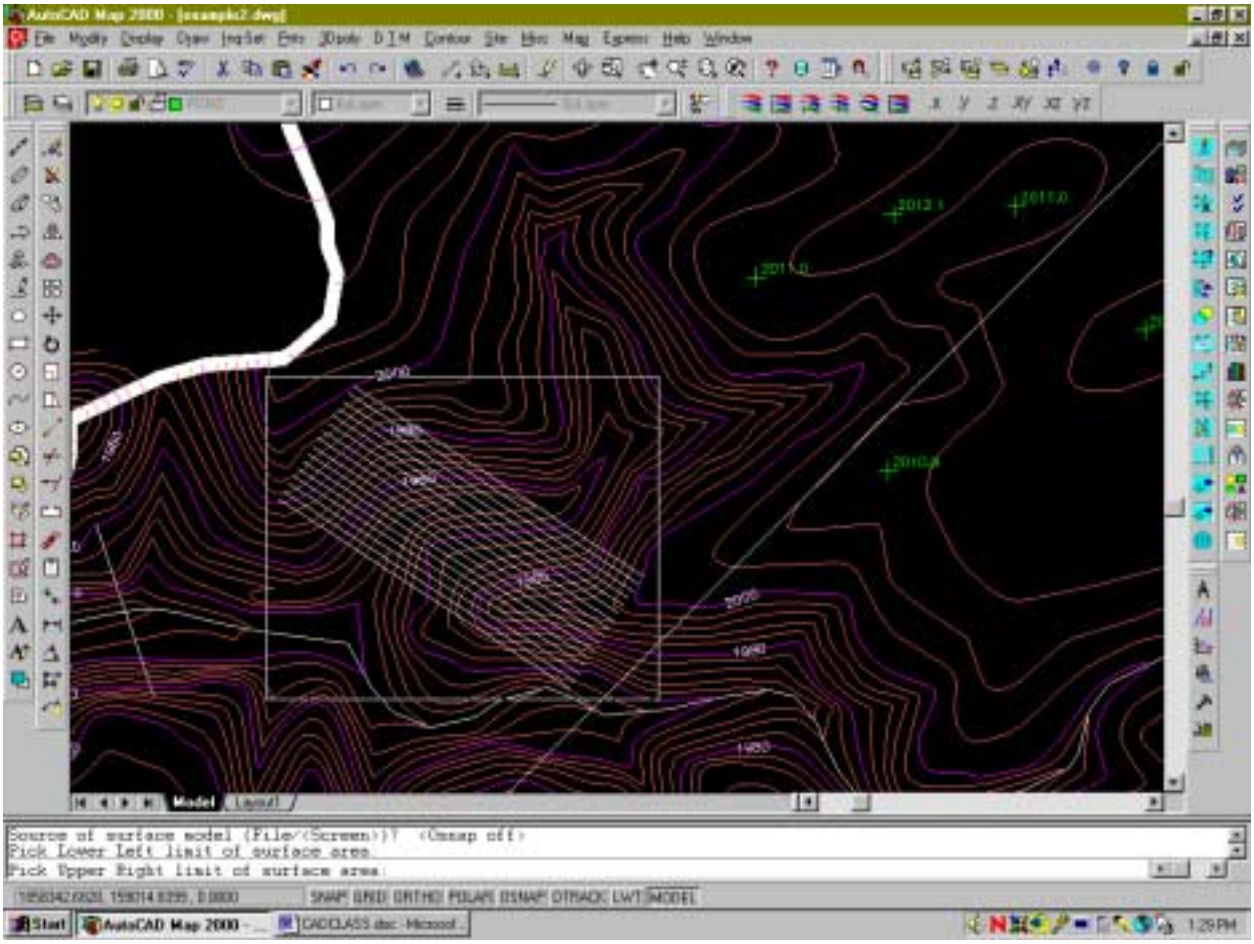
The next prompt is APercent/Ratio/Vertical offset amount <0>@. In this example we will use the vertical offset amount so the user will type in -4 and hit enter. Be sure to include the negative sign (-) because we are starting at the top of the fill and building down.

You will then pick the 2D top of valley line and pick a point down slope from the line and an offset line will appear and the user will be asked to select another line. At this point select the line that was just generated and pick downslope again and a third line should appear. Continue picking lines in this manner until you have 16 lines on the model as shown The user now has a 3D face of contour lines representing the face plane of the valley fill.

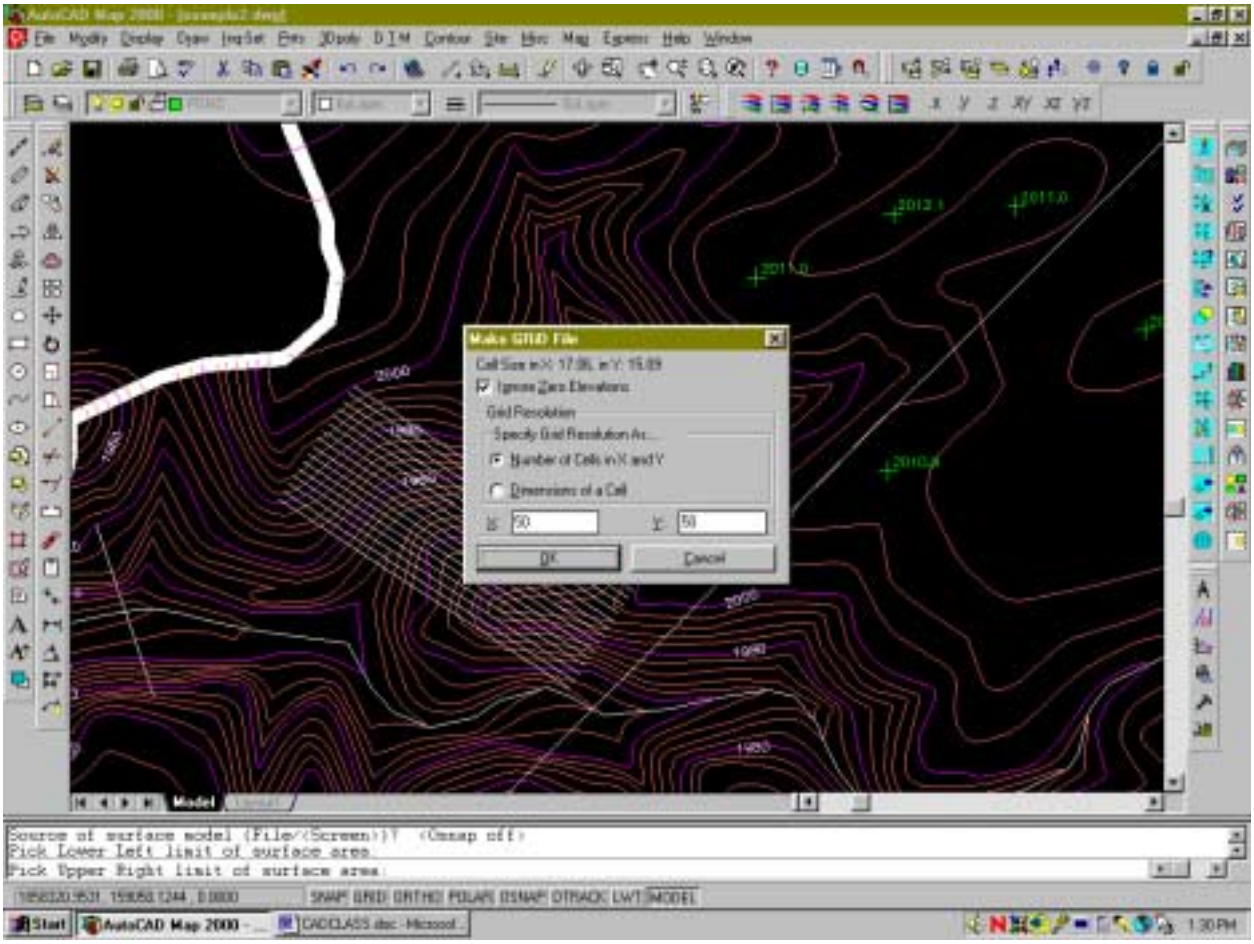


The command ABreak 3D Polyline by surface@ will be used to trim these contours. Pick the command from the pull down and follow the prompts. The first being ASource of surface model (File/<Screen>)?@. This command will allow the user to use a previously generated grid file or take the model from the screen. For this demo we will use the screen so hit enter at this prompt.

The user is now asked to pick the lower left area of the trim area. It is important that this area is large enough to include all lines that are to be trimmed and that the entire area be displayed on the screen. Using the curser place the crosshairs in a lower left area of the lines to be trimmed and pick with the left button. The command will prompt to pick an upper right corner. Using the cursor drag the box until all lines are within the box and pick with the left button again.



A dialog box will pop up asking additional information.

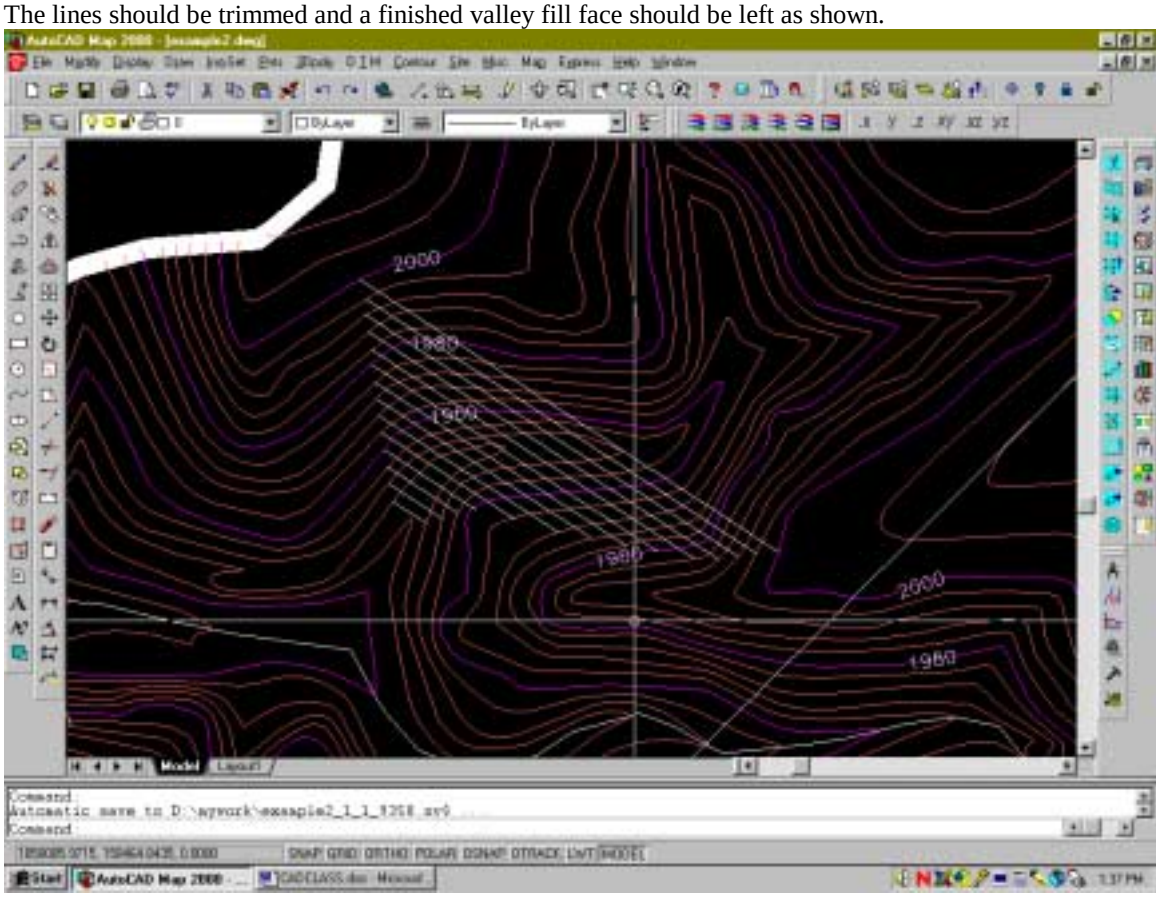

Make GRID File
Cell Size in X: 17.86, in Y: 15.69
Ignore Zero Elevations
Grid Resolution
Specify Grid Resolution As...
Number of Cells in X and Y
Dimensions of a Cell
X: 50
Y: 50
OK
Cancel
Model
Source of surface model (File/<Screen>)? <Osnap off>
Pick Lower Left limit of surface area:
Pick Upper Right limit of surface area:
SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL
Start
AutoCAD Map 2000 - ...
CADCLASS.doc - Microsof...
1:30 PM

The computer is building a grid file Aon the fly@ and wants to know how many cells or the size of the cell. Choose the box with Dimensions of the cell and type 10 into each box. This will build a fairly tight grid of cells 10 units on a side. The smaller the cell side the closer the trim lines will meet the model, but it will also take longer and on older machines may cause a crash. After putting 10 in each box pick OK.

Next you will be asked to ASelect polylines to clip select objects@. Using the cursor pick each offset face line that was built in the previous exercise. When you are finished hit enter to move to the next portion of the routine. Note: you can use your zoom button to get closer to pick lines. Remember to zoom out after you are done picking the lines.

The command will next prompt for the surface lines. Pick all of the surface entities excluding the valley face lines that are being trimmed. Hit enter after all have been picked.

After some calculations which may take up to 5 minutes depending on the speed of the machines the command will prompt with AErase polyline below surface (<Yes>/No)?@. If you choose No the program will prompt you as to what layer you want the various line segments placed on. For this example we know that any line below the model need to be erased so hit enter for Yes.
The lines should be trimmed and a finished valley fill face should be left as shown.



Since these lines were built at the contour interval they tie into the existing contours and can be used as finish regrade contours.

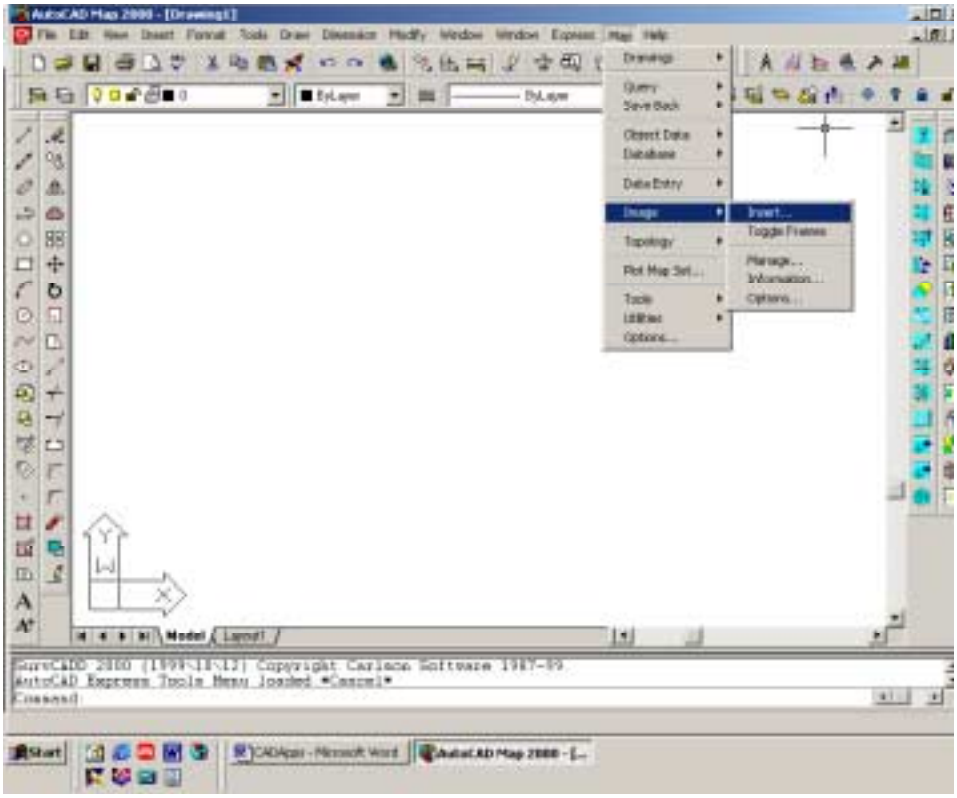
Save this drawing as sc2000\work\example21fill.dwg

Heads Up Digitizing

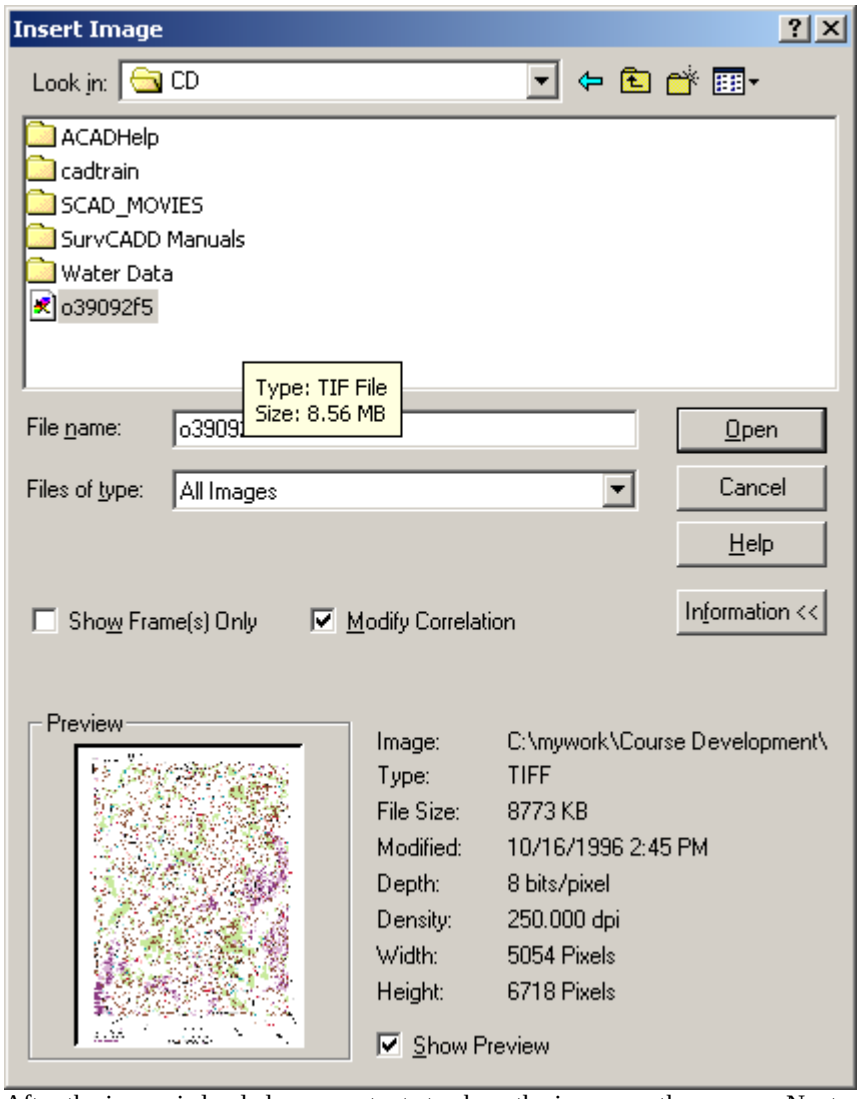
In order to facilitate the digitizing of hard copy maps into vector format, Heads Up Digitizing is utilized. Some preparation work is necessary before going into SurvCADD. The user will need a scanable copy of a map with at least two coordinates on the copy. These coordinates should be on opposite corners.

The copy can be scanned in several different formats such as jpg, tif, gif, rlc, etc.. The most efficient of these formats is jpg for picture images and rlc for line drawings. For this example we will consider a line drawing with contours. Scan the drawing at 300 dpi to keep the file size to something reasonable. After the file is scanned and saved on the computer, you are ready to start SurvCADD.

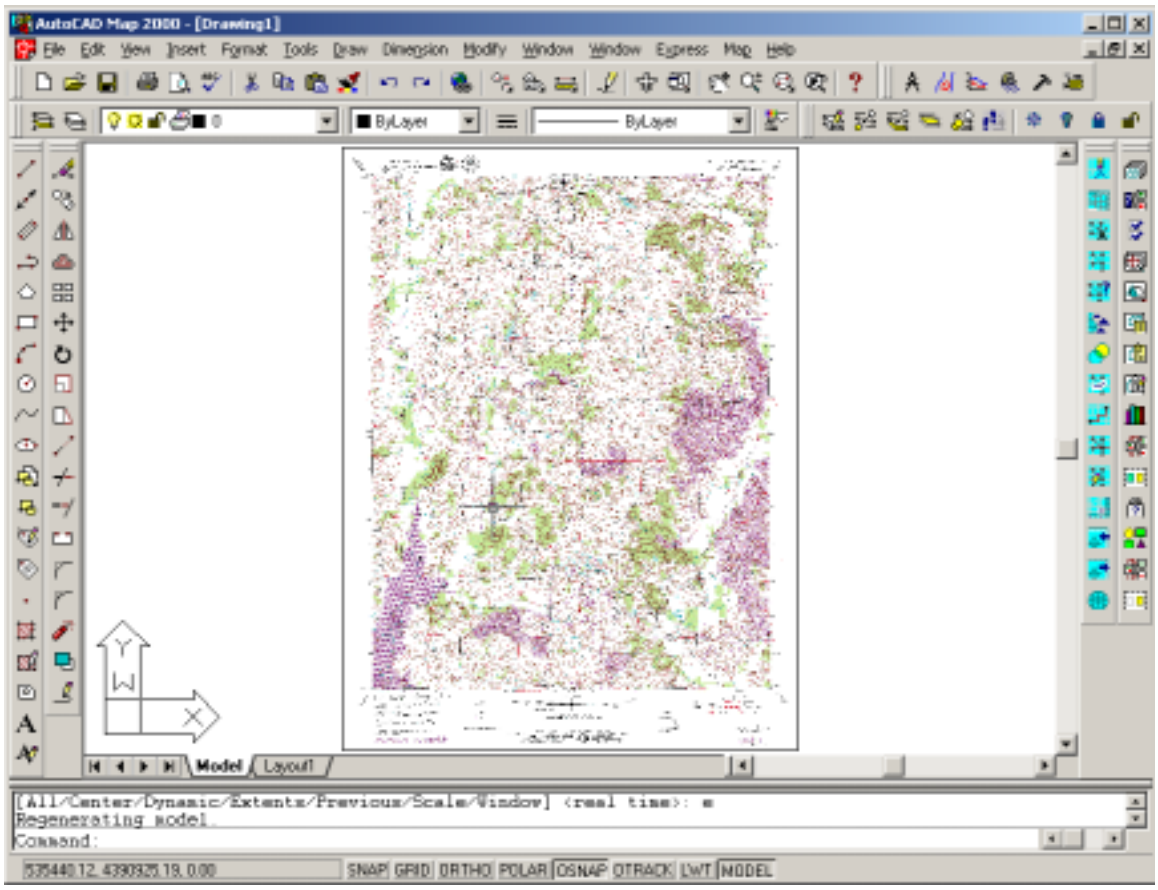
Go to the Map pulldown menu and pick Image then Insert from the menu items.



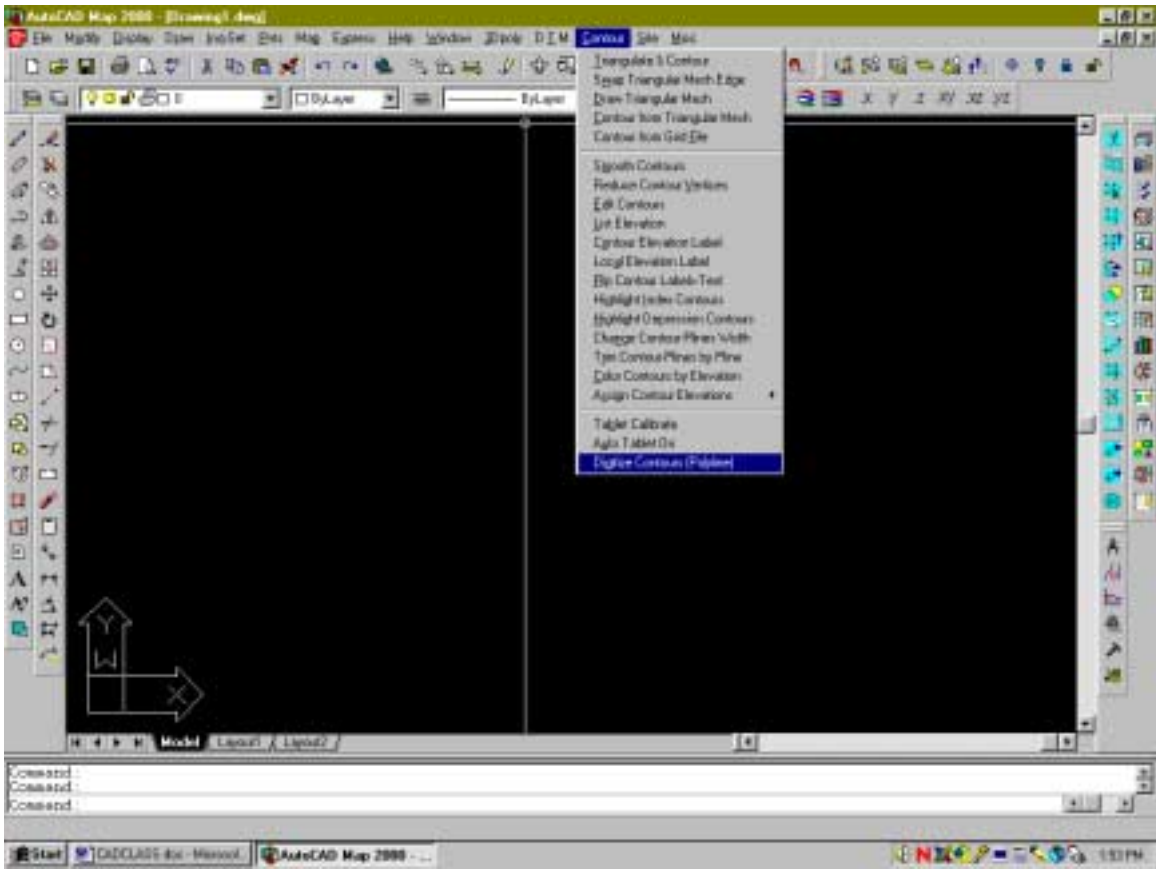
This dialog box will allow you to attach the scanned image of the contours. After picking attach and browsing for the file take the defaults to insert the image into a new drawing.



After the image is loaded zoom extents to show the image on the screen. Next you need scale the image to fit the coordinate system that is on the map.



You are now ready to go to the DTM-Contour module and pick Contour from the menu items.


Triangulate & Contour
Erase Triangular Mesh Edge
Draw Triangular Mesh
Contour from Triangular Mesh
Contour from Grid File
Smooth Contours
Reduce Contour Vertices
Edit Contours
List Elevation
Contour Elevation Label
Local Elevation Label
Flip Contour Labels Text
Highlight Index Contours
Highlight Depression Contours
Change Contour Plines Width
Type Contour Plines by Pline
Color Contours by Elevation
Assign Contour Elevations
Tablet Calibrate
Auto Tablet On
Digitize Contours (Polyline)
Model
Layout1
Layout2
Command:
Command:
Command:
Start
AutoCAD Map 2000 - ...

Since you are not digitizing from a tablet you do not need to calibrate the tablet or use Auto Tablet On. Go to the last command Digitize Contours (Polyline). At the command prompt you will be asked the following: Layer/Interval/Elevation <0.0>: First you can type in Layer and you will be prompted for a layer name for you contours. The default is CTR, hit enter for this value. You will then be asked for an interval value. This is a indexing value for contour digitizing. If you want to build the contours from the lowest to the highest using a 2' interval, you would choose 2. If you want to digitize from highest to lowest using a 5' interval you would type in -5. For this example use a value of 2. You are then prompted for a starting value for the elevation. Since we choose a positive value for the indexing number, you should enter the lowest elevation that you will contour.

You are now ready to digitize the contours. Place you cursor crosshairs on the first elevation line and pick. Continue picking following that contour line. If the line is closed then when you get back near the start type in a C to close the pline. If the line is not closed, when you get to the end hit escape.

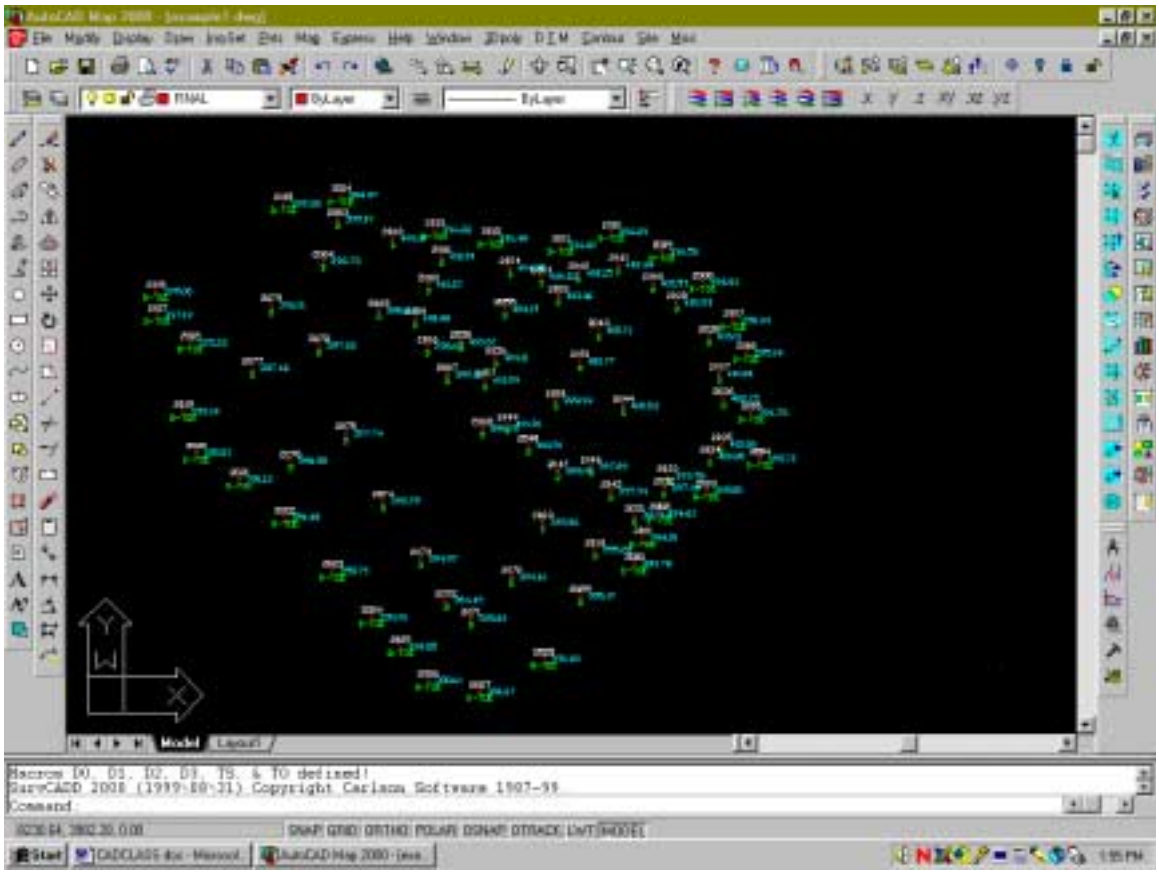
In order to digitize the next contour hit enter again to repeat the command and you will see that you are ready to digitize the next line and that the elevation has indexed itself to the next contour interval. You can choose this value by hitting enter or you can change it by typing in a new value.

Continue in this way until all have been finished.

BASIC CONTOURING

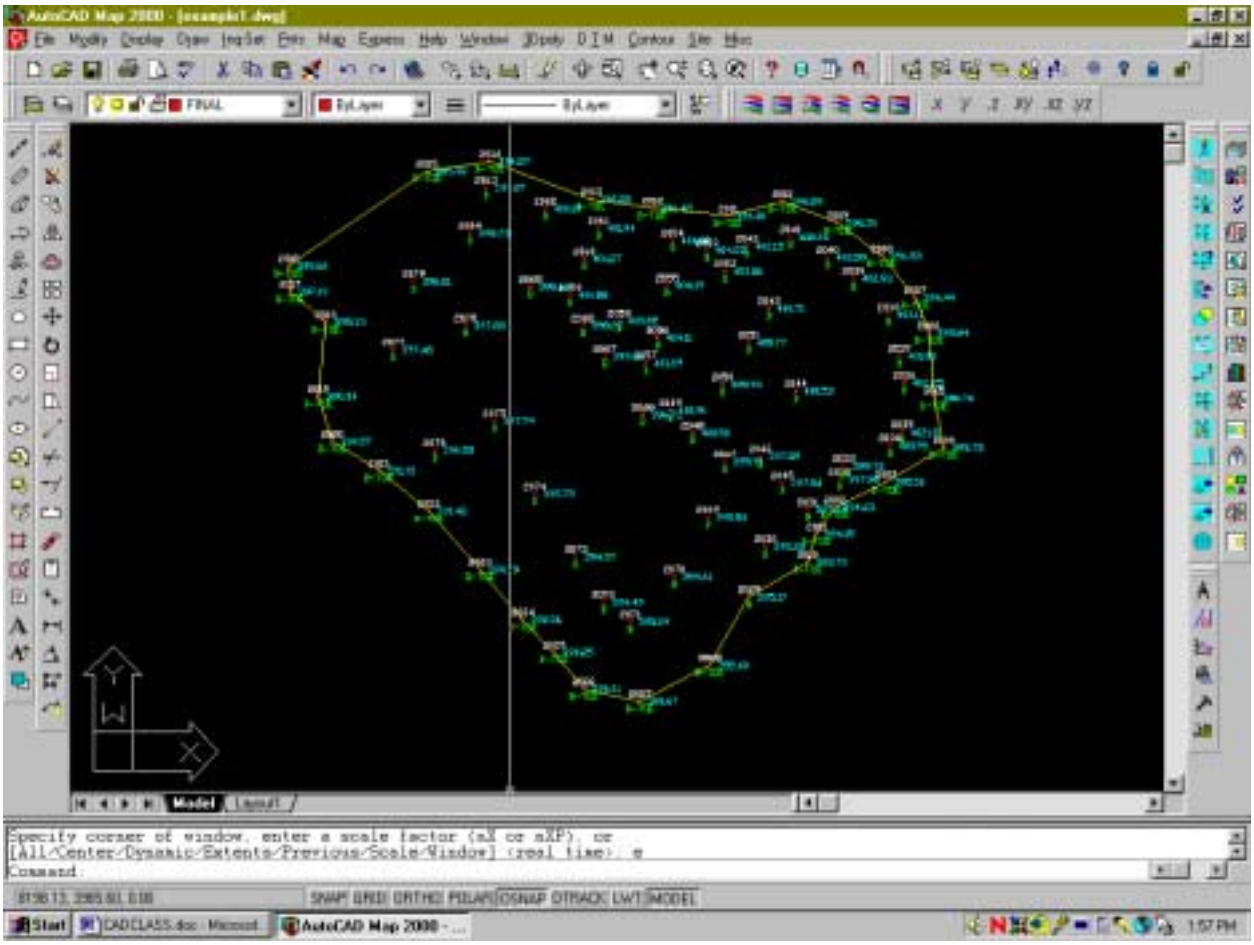
In this module the user will learn basic contouring. The file that will be used is sc2000/work/example1.dwg.

Exercise A



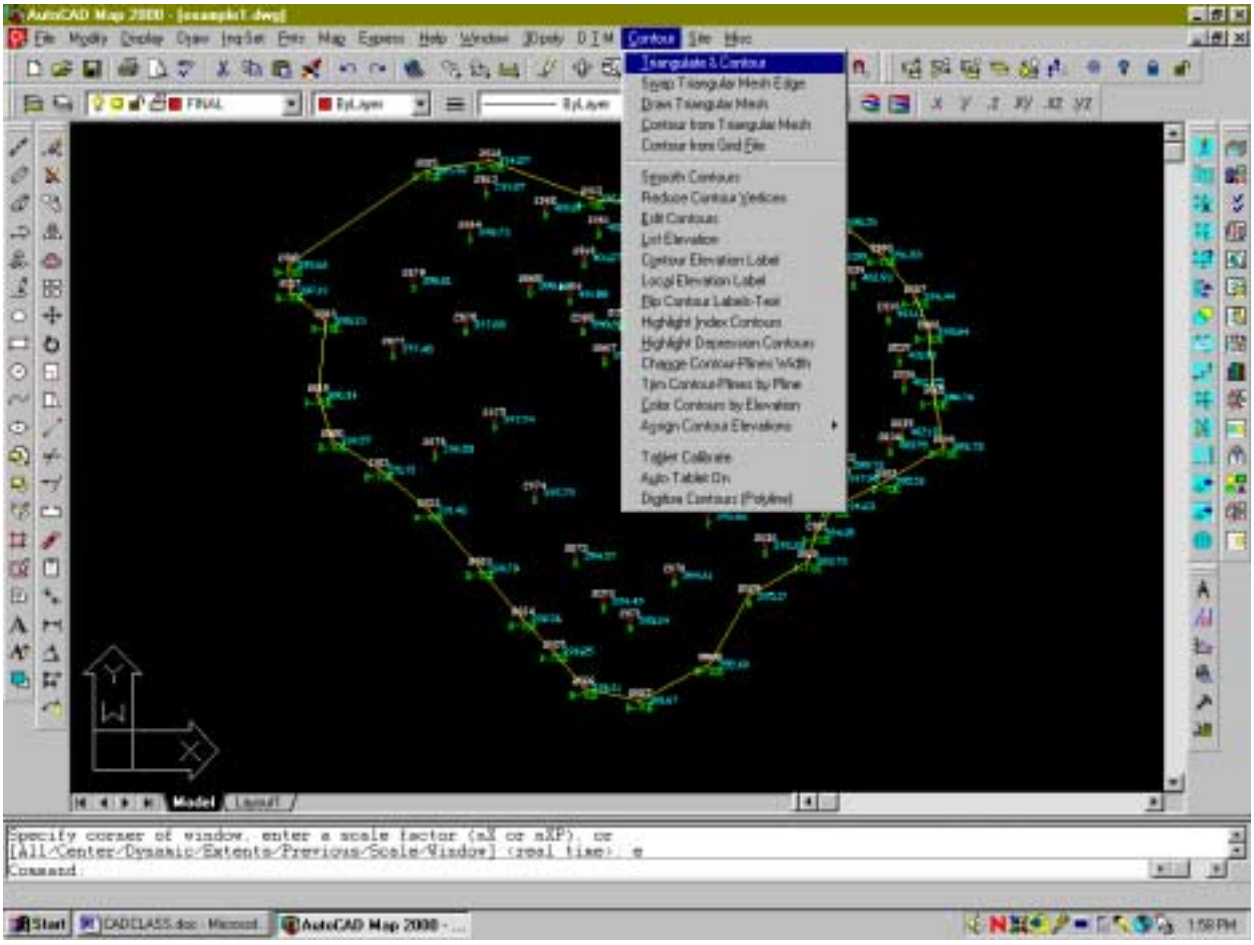
In order to contour this set of points and not get bogus contours beyond the extent of the points the user needs to generate an inclusion perimeter around the points. This perimeter can be either a 2D or 3D polyline and can be drawn freehand or follow the most outside points. It should be noted that if a 2D polyline is used it must be built with an elevation of zero; otherwise the contouring module will use it as data. Therefore it is better to draw a 3D polyline using the osnap of node turned on.

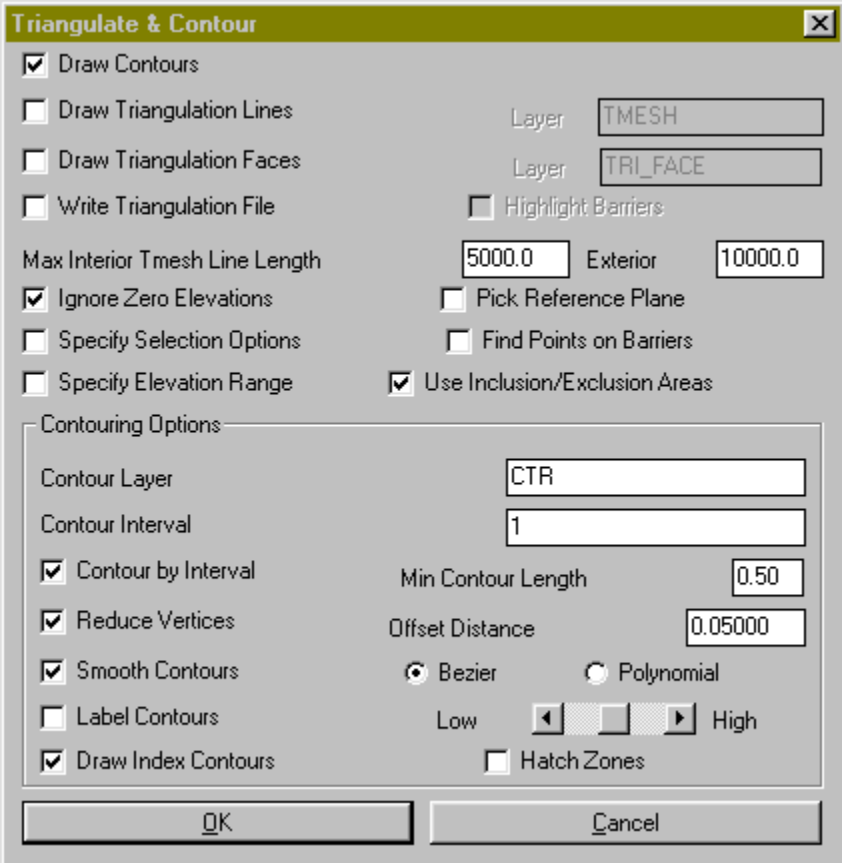
After opening the drawing go to Inq-Set and select the Aperture-object snap and choose the node box and pick OK. Next go to 3Dpoly and select Draw 3D Polyline. After taking the defaults to the three questions proceed to draw a 3D polyline around the points and close the line when done.



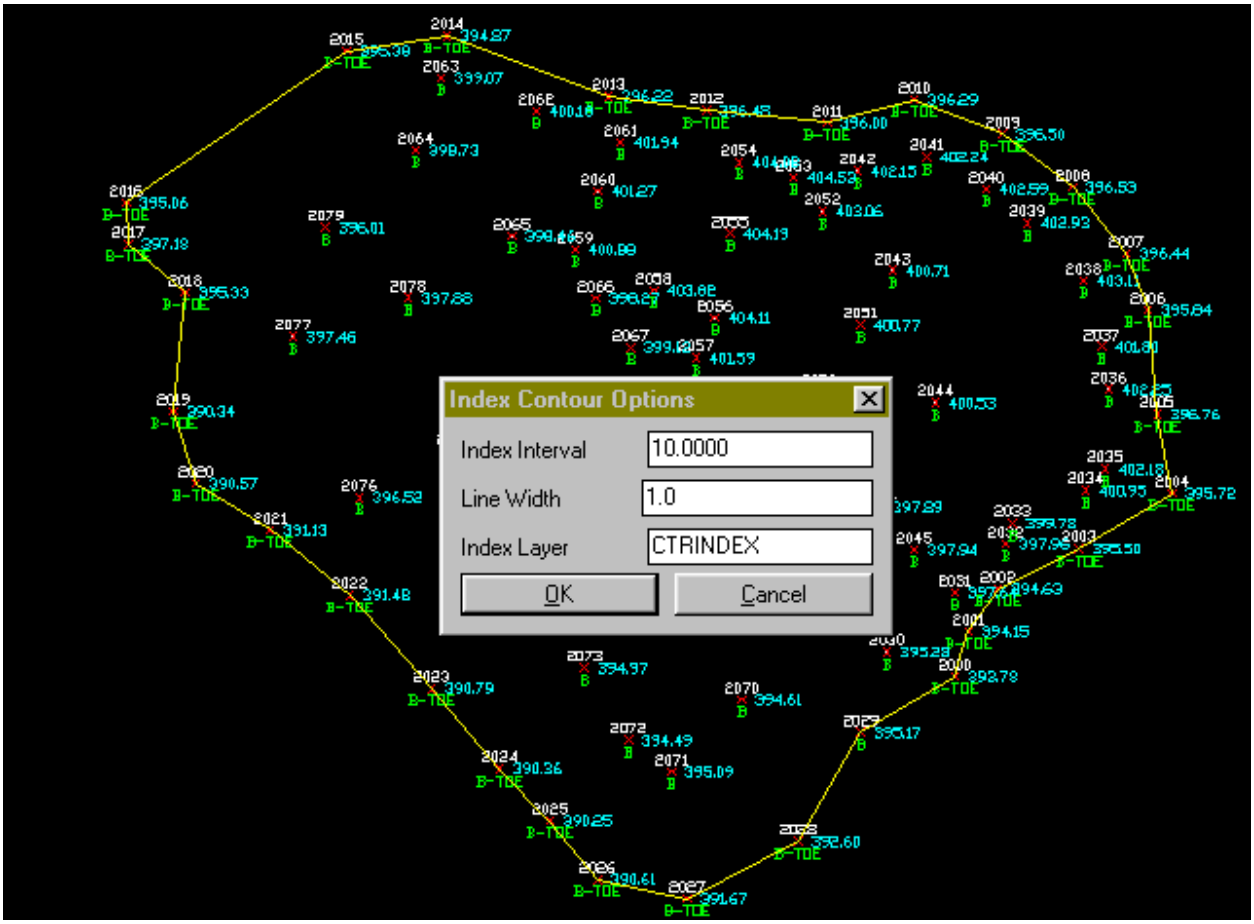
The drawing should look similar to the above. Next select the Contour tab and pick Triangulate and Contour.

Exercise B


Contour
Triangulate & Contour
Swap Triangular Mesh Edge
Draw Triangular Mesh
Contour from Triangular Mesh
Contour from Grid File
Smooth Contours
Reduce Contour Vertices
Edit Contours
List Elevation
Contour Elevation Label
Local Elevation Label
Flip Contour Labels Text
Highlight Index Contours
Highlight Depression Contours
Change Contour Plines Width
Trim Contour Plines by Pline
Color Contours by Elevation
Assign Contour Elevations
Tablet Calibrate
Auto Tablet On
Digitize Contours (Polyline)
Model
Layout1
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: w
Command:
Start
AutoCAD Map 2000 - ...



In the dialog box check the following boxes: Triangulate and contour, Ignore Zero Elevations, Inclusion/Exclusion perimeters, Contour by Interval, Reduce Vertices, Smooth Contours, Draw index Contours. The default layer name of CTR is OK and the defaults for the segment length and smoothing are OK. Change the contour interval to 1. The dialog box should be similar to the one above. Pick OK.



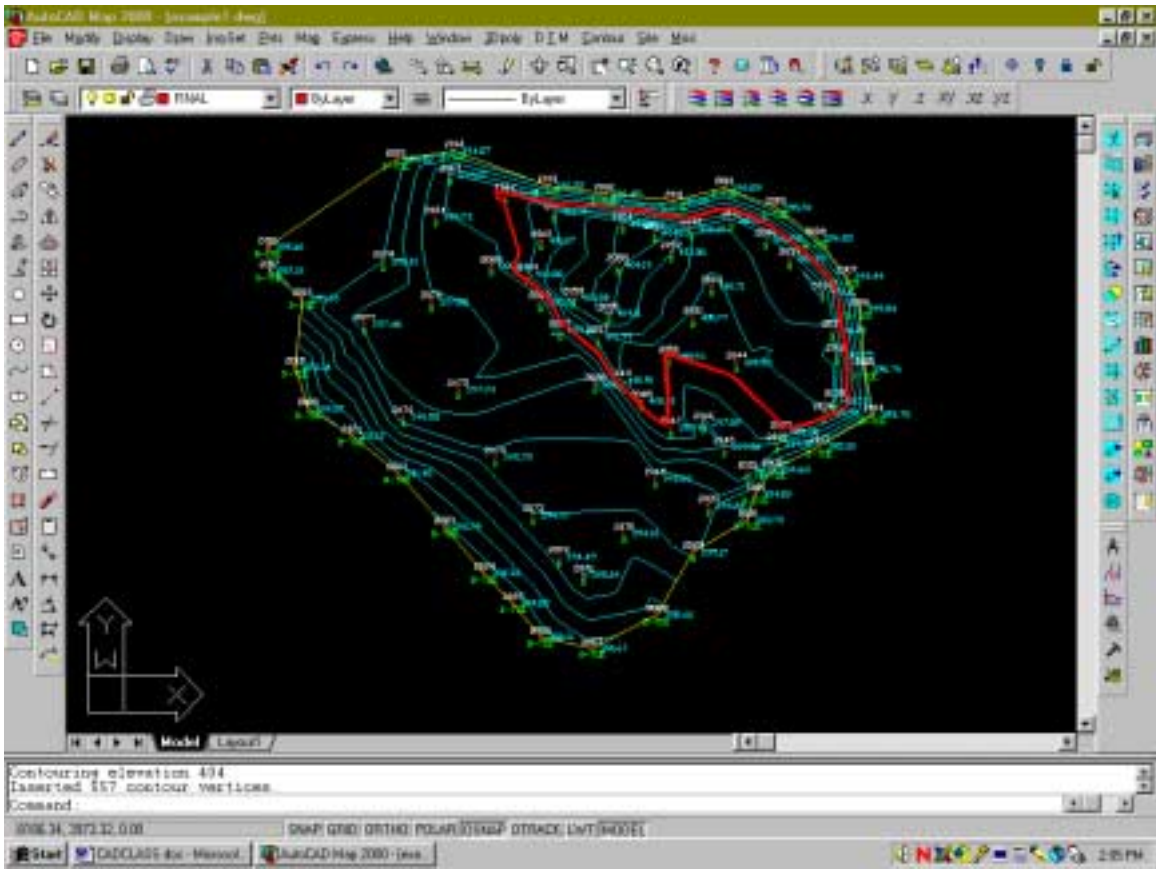
The next dialog is requesting information about the index contours, leave the defaults and pick OK.

The next prompt asks for the inclusion perimeter. Pick the perimeter line and hit enter. You will be prompted for an exclusion perimeter. Since there is none hit enter again.

At the next prompt select all of the lines and points on the screen as this is the data that you need to generate the contours. Hit enter after selecting all of the lines.

The following contours will be generated.

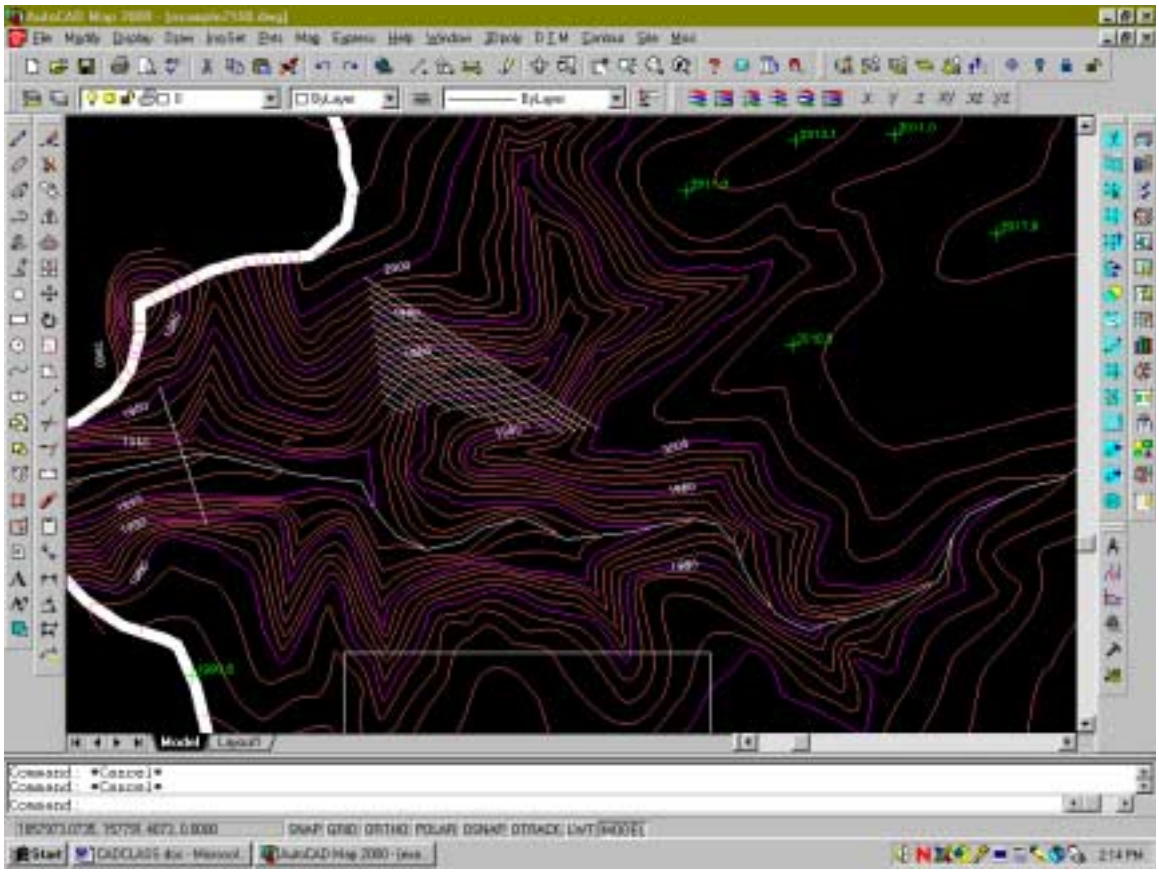
Save this example as sc2000/work/exampl1con.dwg.


Contouring elevation 404
Inserted 557 contour vertices
Command:
Model
Layout1

VOLUMES BY LAYERS

In this module the user will learn to calculate 2 surface volumes using different layers as a selection filter. From the previous session on 3D Poly Utilities we will use the hollow fill face that was developed and show the user how to calculate the volume of the fill.

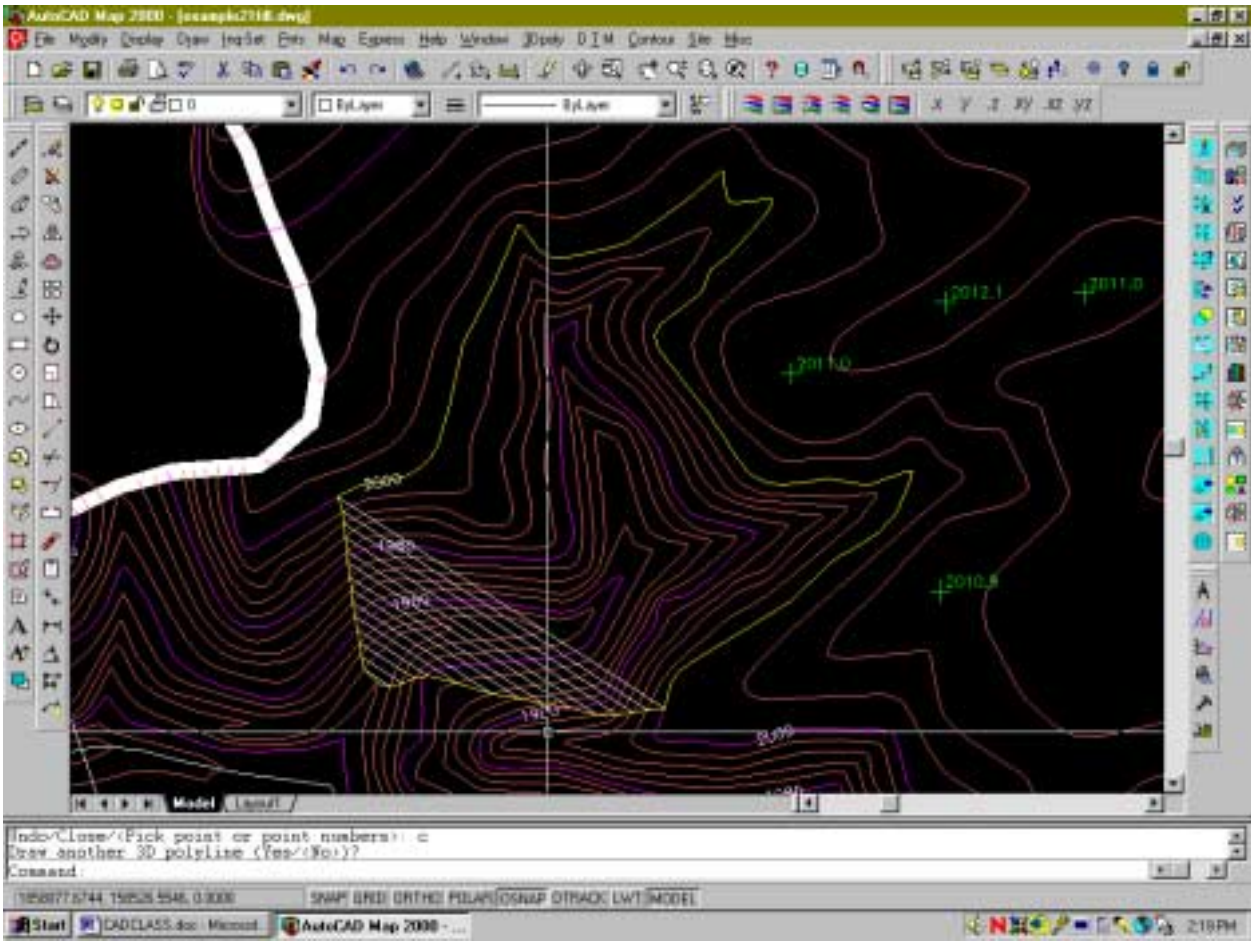
Exercise A



The first item needed is to build an inclusion perimeter around the regrade lines. This inclusion line needs to outline the fill face and continue back the hollow along the 2000' contour as this is where the fill will extend. This can be done with draw 3D poly using the osnap filter of END.

First the user will go to the DTM menu and pick Inq-Set, Aperture-Object Snap (OSNAP) and will select end as the OSNAP filter. And hit OK. Next the user will chose 3D Poly, draw 3D polyline. Take the defaults of layer name (Barrier) and no for the next two questions. When the user is asked to start picking points, start at the base of the fill being sure to pick the end points of the regrade lines. Move upward along the regrade/original grade interface with your perimeter line until you get to the 2000' contour.

At this point continue the line along the 2000' contour selecting the vertices of the line. The OSNAP will automatically highlight the vertices as you move along the line. Just select the vertex by using the left pick button. Go around the 2000' contour until you intersect the opposite side of the face regrade line. Proceed down the regrade/original grade interface until you come to the starting point. Before you select the original starting point type C for close. Your perimeter should look like this.

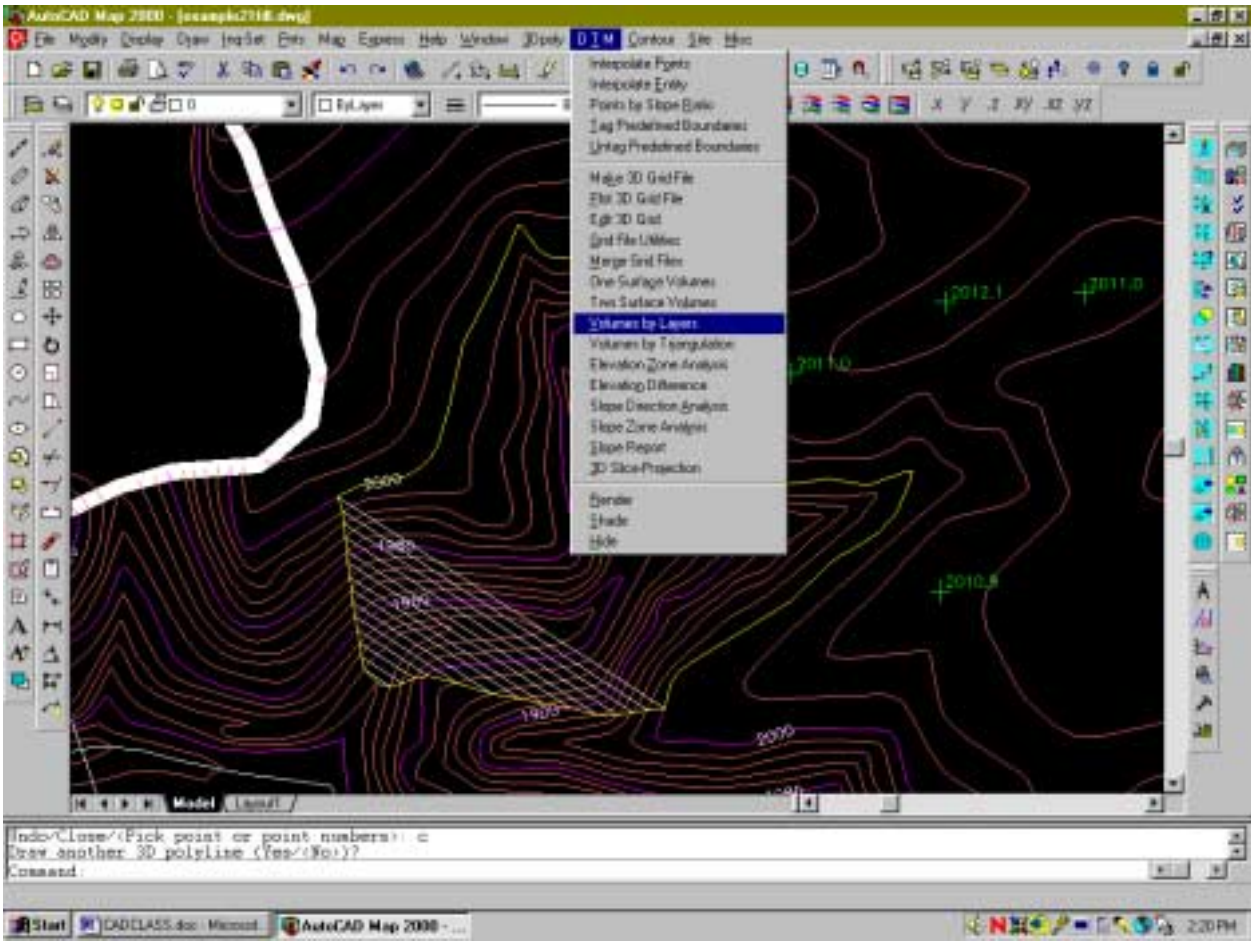


This perimeter line shows us how much of the hollow was filled by the face regrades.

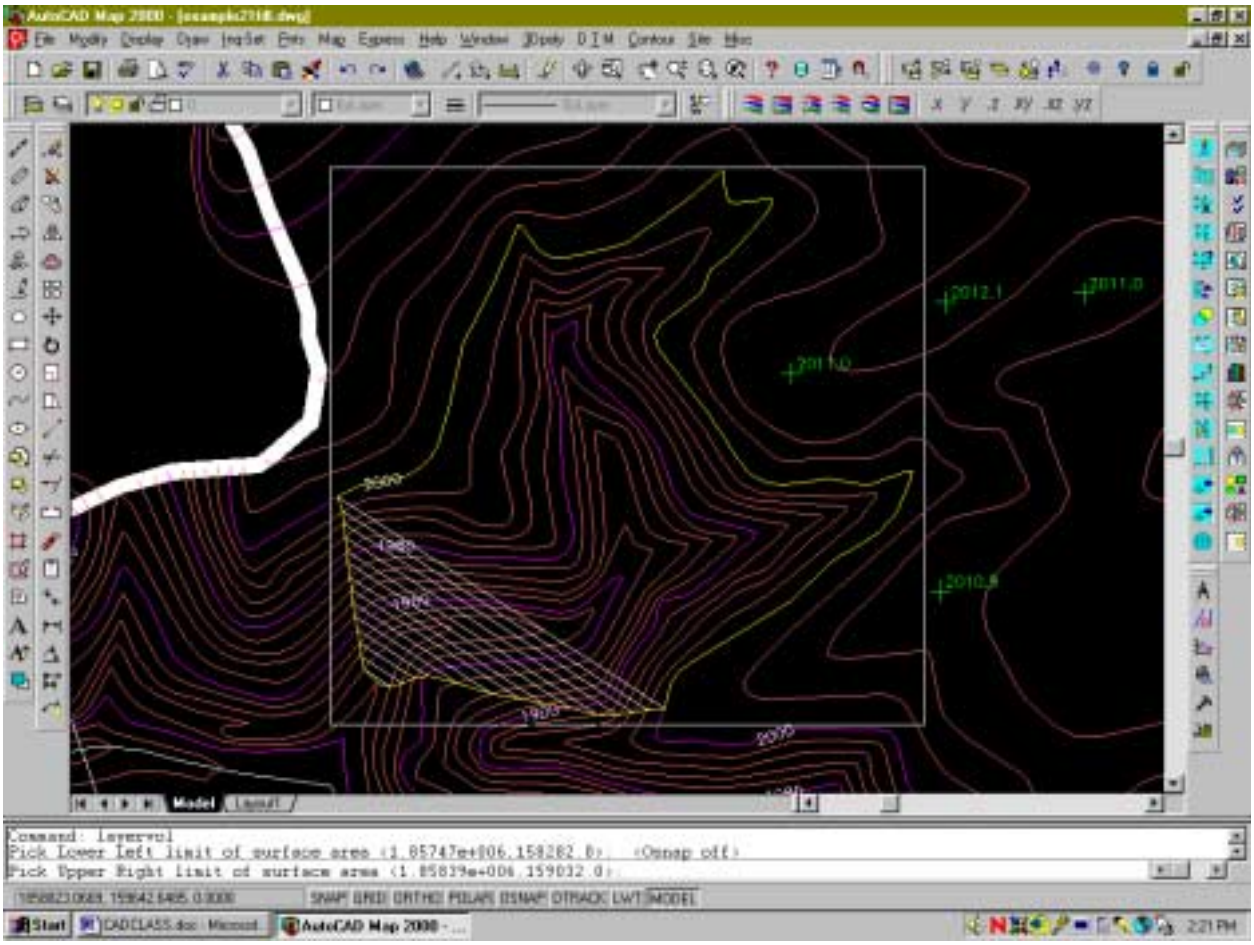
Before starting the volume calculation the user must be sure the entire perimeter line is shown on the screen. If you try to use the volume by layer command with part of the area not shown you will get erroneous volumes. The volume by layer command is a grid operation and the computer builds the grids Aon the fly@, but it must be able to Asee@ all of the lines within the inclusion perimeter.

Exercise B

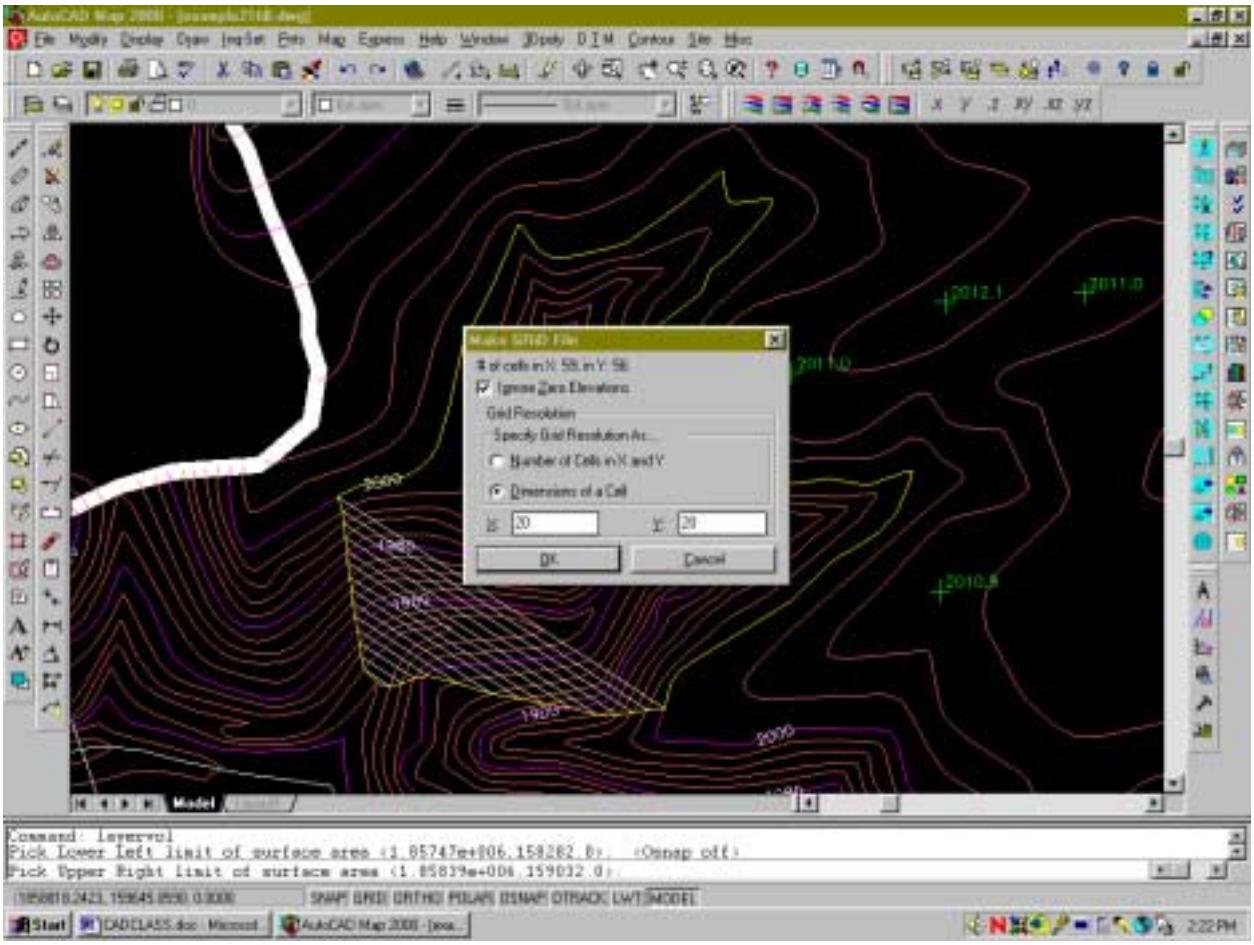
Go to the DTM, Volumes by layers command and select it.

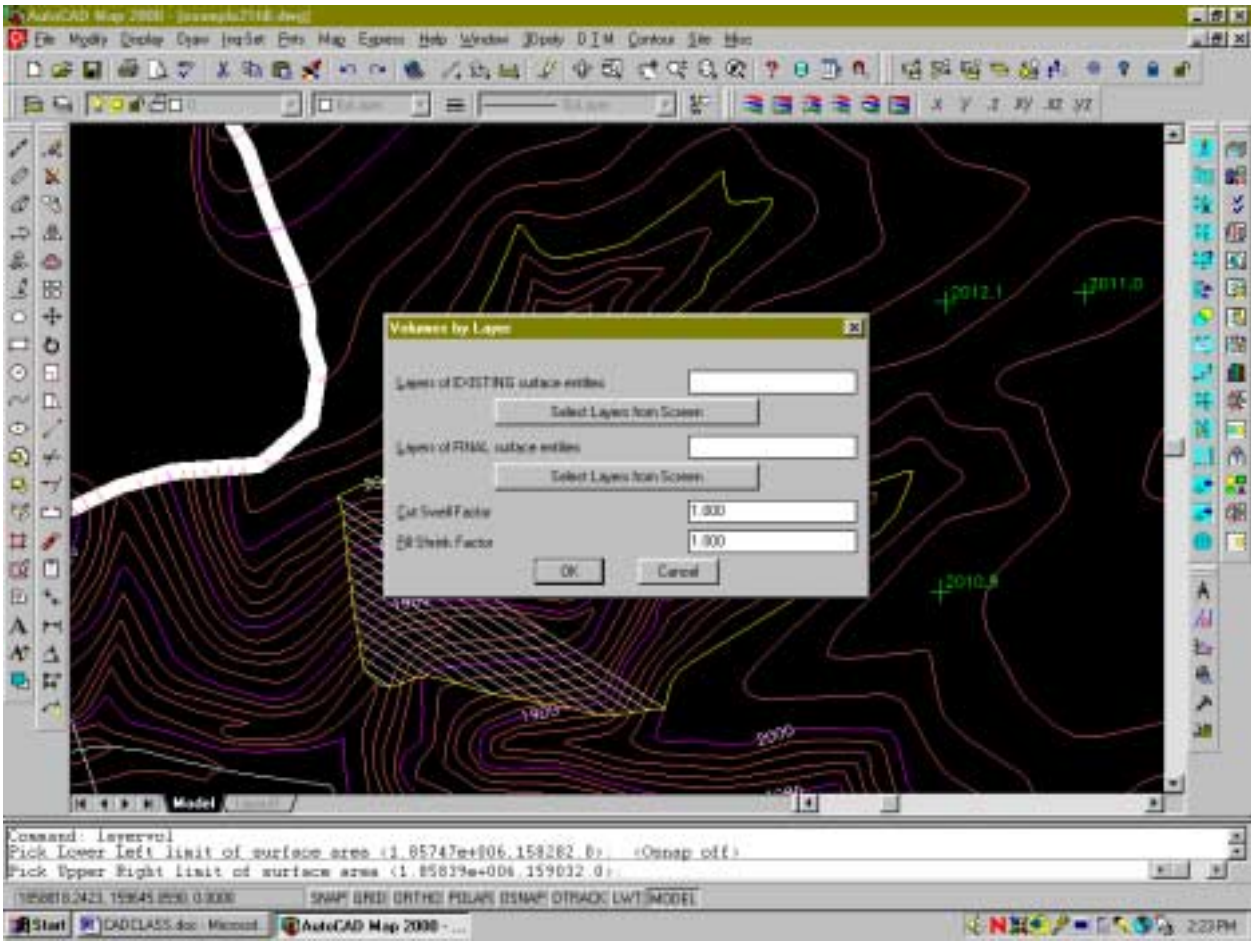


The user will be asked a series of questions at the command line and also with dialog boxes. The first two questions ask the user to select the lower left and upper right corners of the area to be calculated. Make the selection as small as possible but still including all of the inclusion perimeter within.



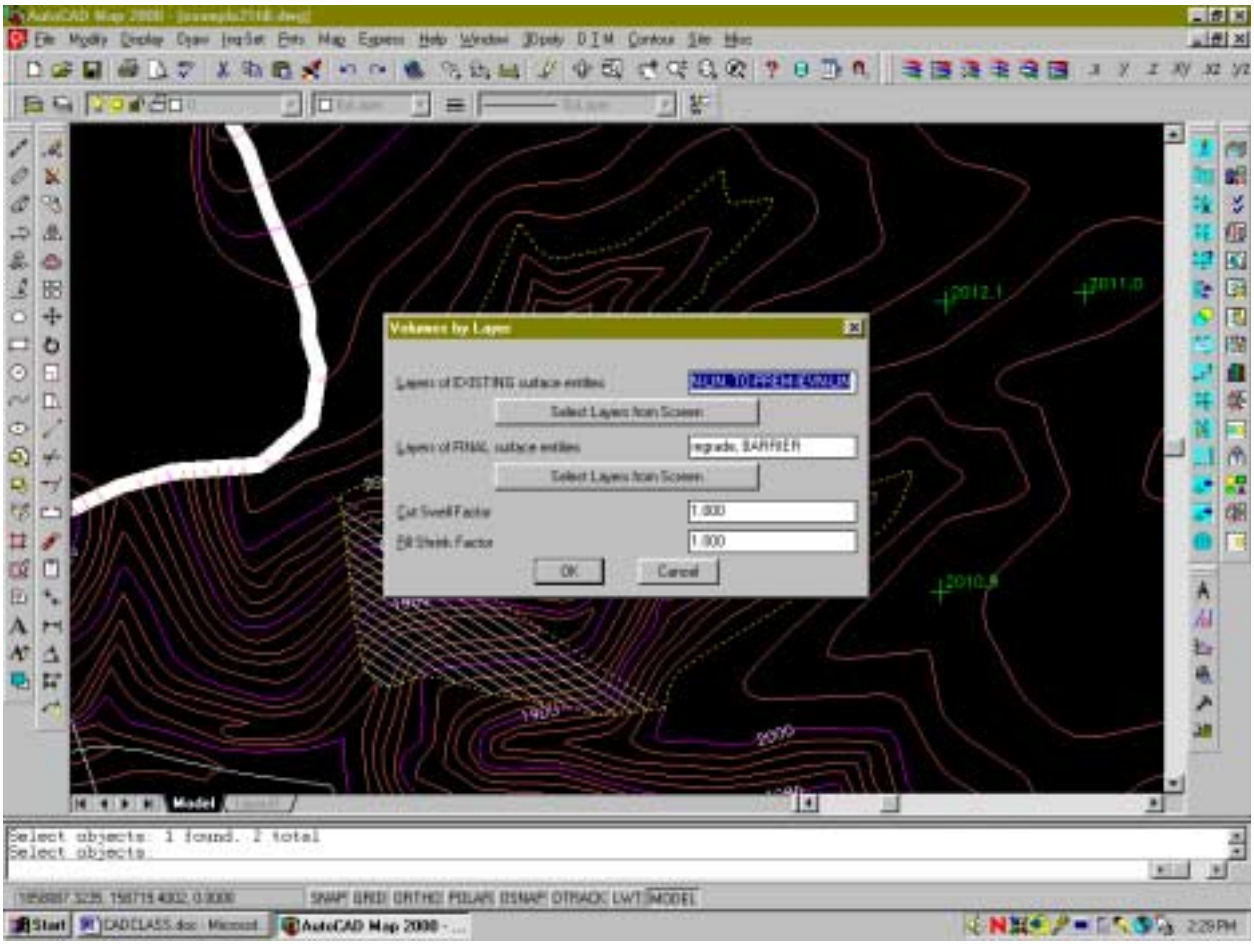
After picking the upper right end of the grid area a dialog box will appear. The user will be asked to ignore zero elevations and to select the size of the grid cells. Be sure the ignore zero elevation box is checked and select the Dimension of the Cells option and make the cell size 20 on each side. When you are finished select OK.


Make GRID File
# of cells in X
in Y
Ignore Zero Elevations
Grid Resolution
Specify Grid Resolution As...
Number of Cells in X and Y
Dimensions of a Cell
X: 20
Y: 20
OK
Cancel
Model
Command: layerval
Pick Lower Left limit of surface area <1.85747e+006,159282,0>: <Osnap off>
Pick Upper Right limit of surface area <1.85839e+006,159932,0>:
Start
CADCLASS.doc - Microsoft ...
AutoCAD Map 2000 - [exa...
2:22 PM



Next the user will be asked to select layer for the existing contours and for the final contours. First select the button Select Layers from Screen for the existing. The dialog will disappear and the user can select lines representing the existing contours. You only need to select one line representing each layer that you want. In this example you would select one of the red lines and one of the magenta lines. Hit return after selecting the two lines and the dialog will reappear with the layer names of the lines that you just selected.

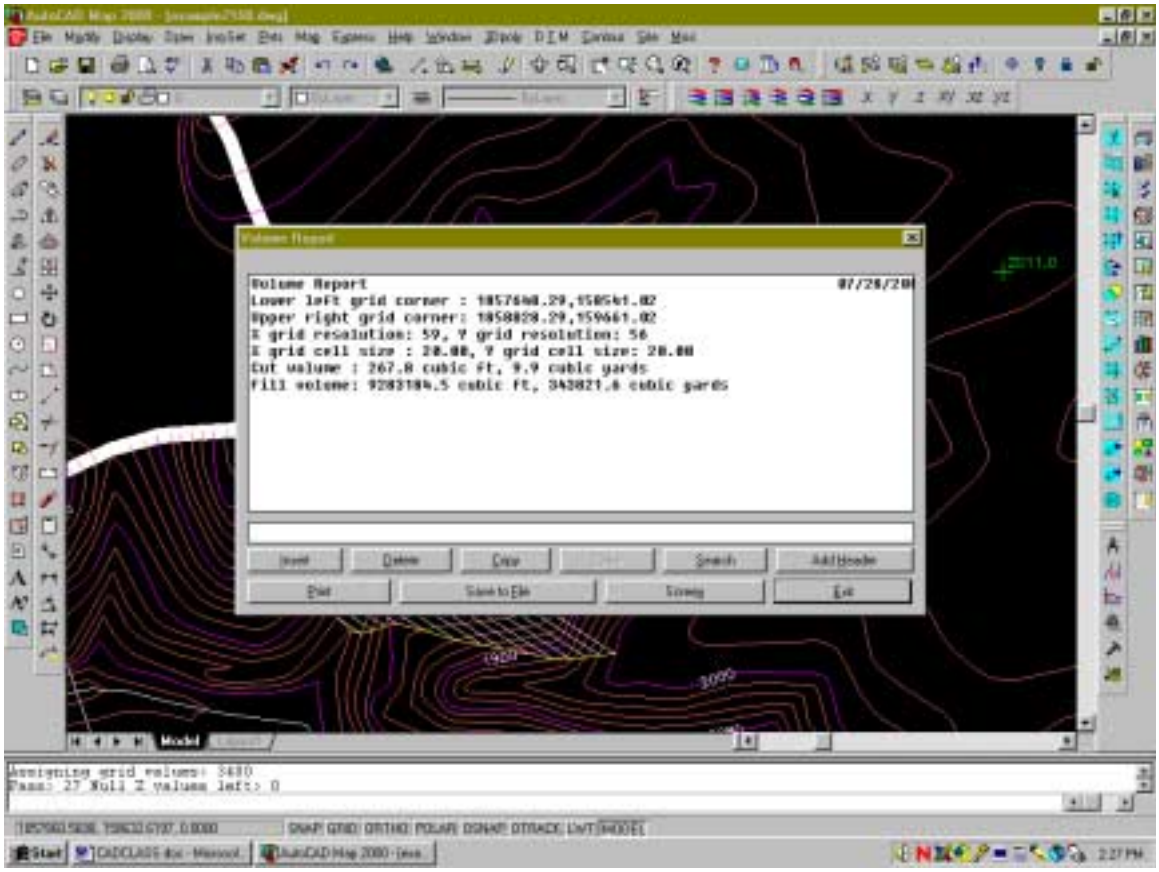
Do the same thing for the final selecting the white regrade line and the yellow inclusion perimeter. These lines represent the regrade or final layers.



When you are finished selecting the layers your dialog should look like the one above. This dialog also asks for shrink/swell factors. For this example we will leave them each at the default of 1.0. Select the OK button and continue.

The user will be asked to select the entities representing the surface features. With a crossing window select all of the entities with in the inclusion perimeter. Don=t worry about selecting more than you need, the computer will filter out the ones it doesn't want. Hit return and you will be asked to select the inclusion perimeter. Select only the yellow line and hit enter. You will be asked for an exclusion perimeter. There is not one in this example so hit enter again.

After some calculations the results dialog will appear showing the user the grid information and the cut/fill volumes.



This volume report can be printed, saved to a file or placed in the drawing. When finished hit exit and the command is finished.