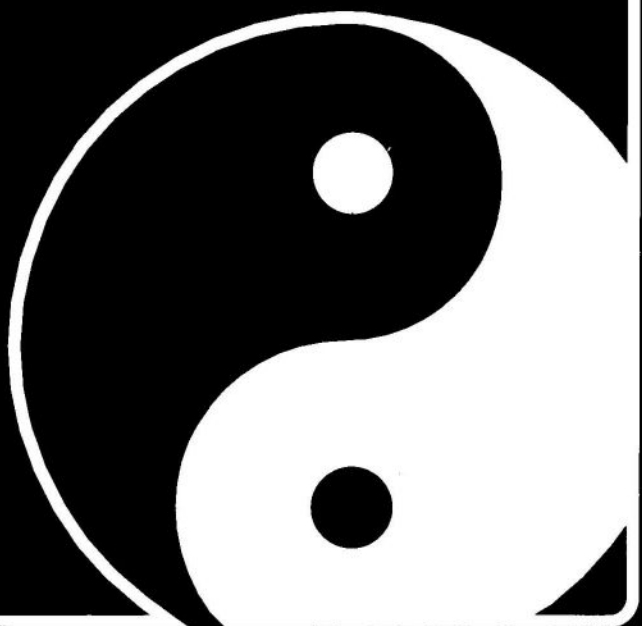
CADMOVER®

Real World
Graphic Translator



KANDU

software corporation

CADMOVER®

Vector Graphic Translator

Version 4.0

User's Manual

KANDU

software corporation

CADMOVER Program - Designed and Written by Ken Lathan.
CADMOVER 4.0 Manual - Written by Ken Lathan
CADMOVER 4.0 Manual - Edited by Dotti Lathan
ScriptMaker - Written by Conan Green

Published by Kandu Software Corporation

Special thanks go to the many users of the previous versions of CADMOVER for their helpful comments, real world files and constructive criticisms.

Table of Contents

1. Getting Started

2. The Options Menu

2. The Options Menu Cont.

3. The Lines Menu

4. Preferences Dialog

4. Preferences Dialog Cont.

5. File Formats

6. Batch Processing

6. Batch Processing Cont.

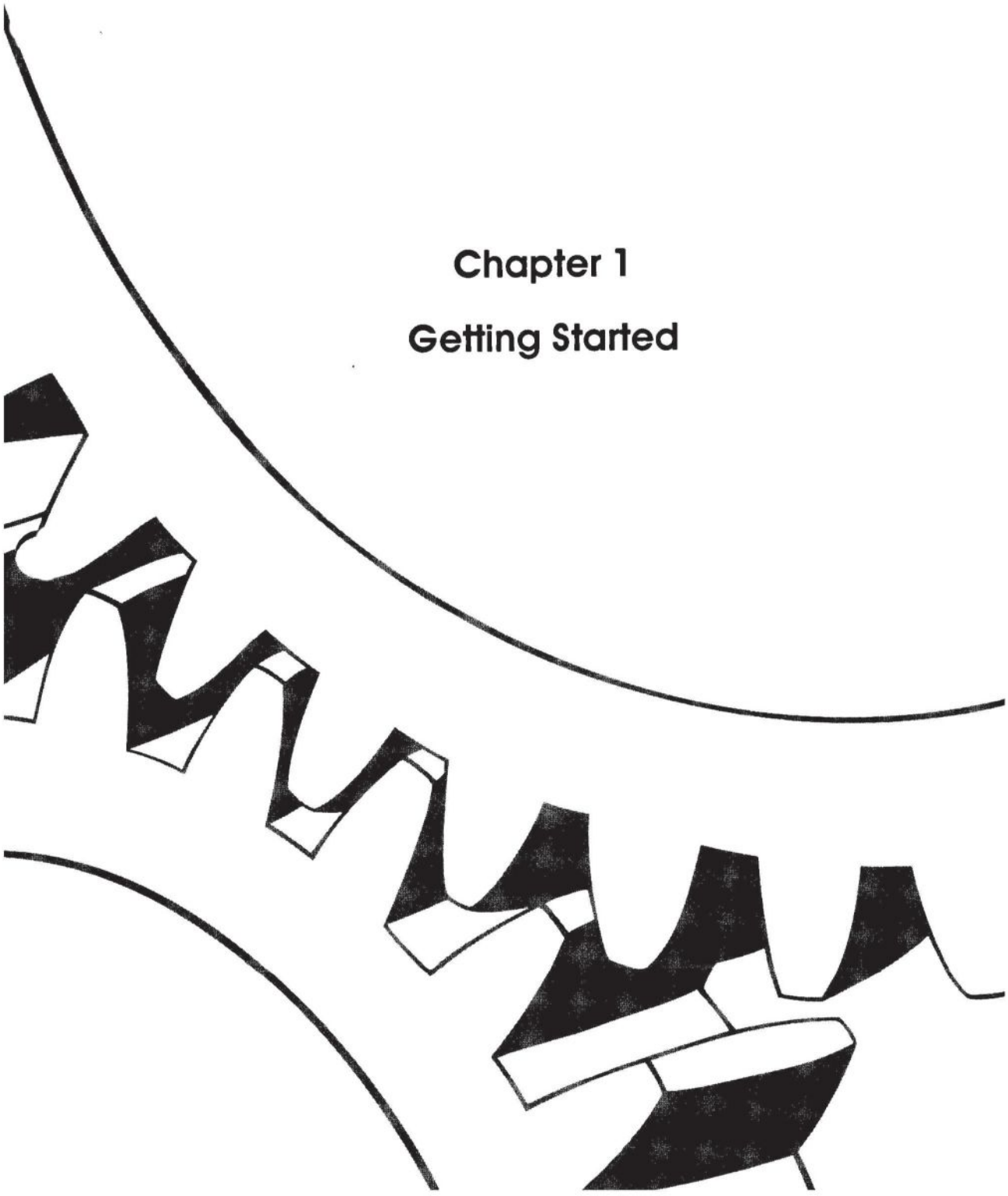
7. AperText

8. Troubleshooting

9. Appendex & Index

Chapter 1

Getting Started



Introduction

About CADMOVER

Since the dawn of time, people have communicated with each other through pictures. The first pictures were probably scratched on the ground with a finger or a stick. In later days, cave painting reached a high art. Even in those days, it was clearly evident that pictures held the ability to convey meaning, and people felt the need to use them to express their thoughts and feelings.

When printing was invented, a single picture could be cut into a block of wood, and then printed many times over. Cutting a picture into a block of wood is a laborious process, requiring ample quantities of skill and patience. Our search for better and faster ways to communicate graphical information continued.

Enter the Industrial Revolution. Photography. Offset printing. Wirephotos. Photocopiers. Facsimile machines. The list of media used for the production, storage, and transmission of pictures goes on and on.

Nowadays it isn't unusual for mechanical, architectural, and other engineering drawings to reach a high level of detail and complexity without ever setting ink on paper. Computers help us to make these drawings, more quickly and more accurately than we ever could before.

The purpose of pictures is still the same: to communicate ideas. And in today's global community, graphic communication is all the more important because, although the human race has hundreds of different spoken languages, one picture can convey meaning to everyone.

But —computers do not all speak the same language. A picture on one computer is likely to be meaningless garbage on another computer. Even on the same computer, different programs use different file formats, preventing graphics programs from sending pictures to one another.

That's why Kandu Software is proud to present CADMOVER. Pictures communicate ideas, and CADMOVER communicates pictures. It can read a variety of CAD/CAM file formats, and rewrite a picture in another format. The language barrier has been broken.

Getting Started

Installing CADMOVER On A Hard Disk

There are several steps involved in installing CADMOVER® on your hard disk.

Important Note: Anti-Virus Protection Software

Make SURE that there are no anti-virus checkers running during the installation process! This step is VERY IMPORTANT! Anti-virus software can damage CADMOVER during its installation. After the installation is complete, anti-virus software can be reactivated, and it will not interfere with CADMOVER's normal operation.

If your computer has System 7 or above installed, you can temporarily disable virus checkers by restarting your computer while holding down the SHIFT key. Continue to hold down the shift key until you see the message "Welcome to Macintosh / Extensions Off".

If you are using System 6 or earlier, then it will be necessary to drag the icons of any anti-virus software OUT of your System folder and then restart your computer. After you restart, you may drag the icons for virus protectors back into your System folder.

Personalize Your Floppy Disk

AFTER you have restarted your computer with virus protectors disabled, insert your master floppy disk into the floppy drive, and double-click on the CADMOVER icon to launch the application FROM THE FLOPPY DRIVE. The personalization screen appears (Fig. 1-1). Replace the word "UNOWNED" with your personalization (Fig. 1-2) information, preferably your name and organization name, using at least 7 characters, but not more than 71. After a delay of several seconds, the CADMOVER splash screen (Fig. 1-3) appears. When the splash screen appears, choose Quit from the File Menu (or type command-Q).

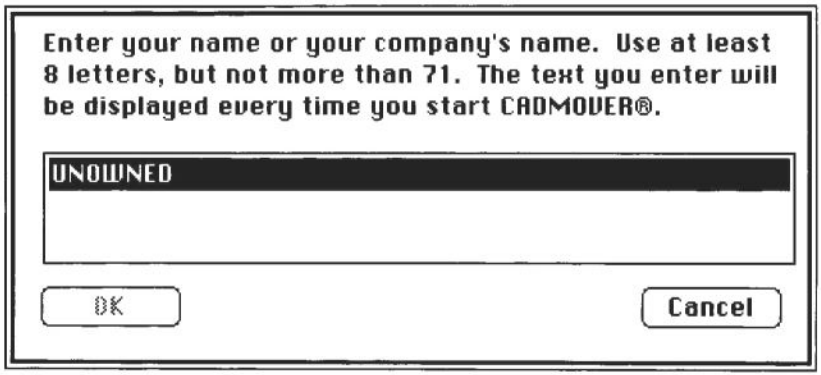


Fig. 1-1: The Personalization Dialog After Clicking On The Word "Unowned".

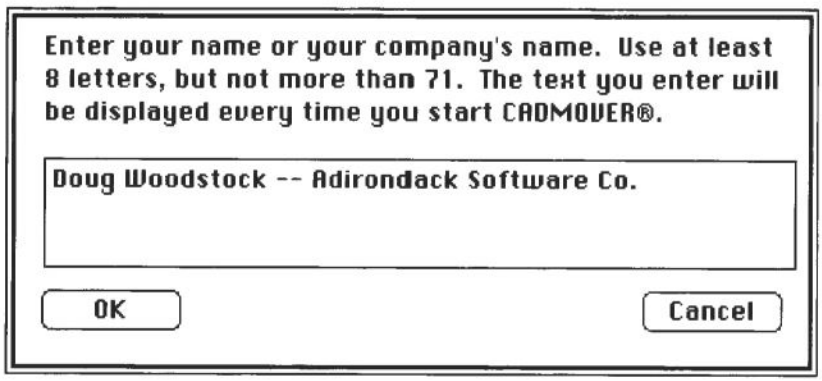


Fig. 1-2: A Completed Personalization Screen

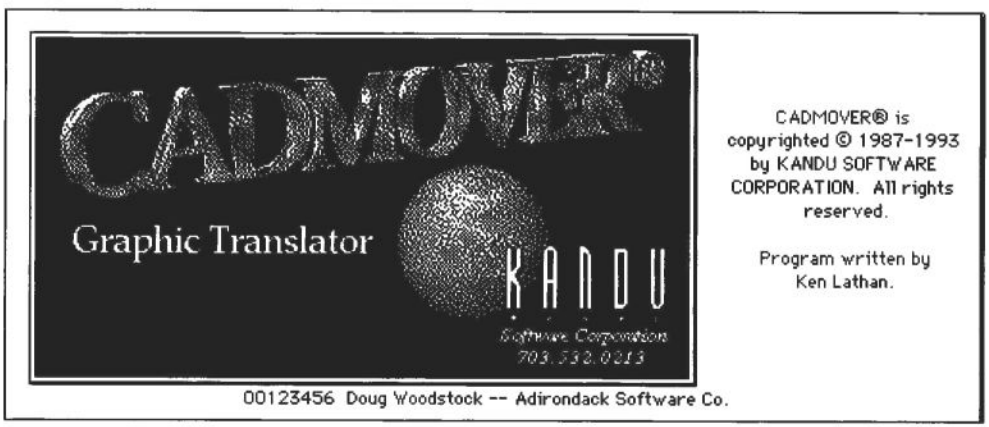


Fig. 1-3: A Personalized Splash Screen.

Run The Installer Application

The Installer application installs authorization to run CADMOVER on the hard disk you choose. The Installer application MUST be run from the master floppy disk, or it will not work properly. Detailed help for running the installer is available by typing command-H once you have started the installer. The installer application does NOT copy the CADMOVER application to your hard drive — that's the next step.

Copy The Application To The Hard Drive

Drag the CADMOVER® icon from the floppy disk onto the same hard drive that you selected in the Installer application.

- RESTART YOUR COMPUTER ONCE AGAIN

This is not a mandatory step, but if you have temporarily disabled your virus protection, this is a good time to turn it back on by restarting your computer.

Put Your Master Disk In A Safe Place

Once CADMOVER has been installed on your hard disk, the original master disk may be put away. You won't need it unless you wish to transfer authorization from the disk where you've installed it to a different disk. Your master disk will also be needed if you wish to reformat the hard disk where CADMOVER is installed (see below).

Now you're ready to translate graphic files!

Important Note: Before You Reformat Your Hard Disk

If you should ever wish to reformat your hard disk for any reason, be SURE to UNINSTALL CADMOVER (using the Installer application) before you reformat. After you have completed the reformatting process, CADMOVER may be reinstalled via the Installer application.

Using CADMOVER

Using CADMOVER to translate files is a straightforward process. This chapter constitutes a quick and easy primer for using CADMOVER. We have added a Short Course section for those of you who are very conversant with the Macintosh. All you have to do is select a file for input, and then select an output format and a name for the translated file. CADMOVER does the rest.

The Short Course - Open and Save

To translate a file, simply open the file you wish to translate, and wait until it has been read in. Opening a file is accomplished by selecting "Open..." from the File Menu. Then save the file by selecting "Save As..." from the File Menu. You may change the output format by making a selection from the popup menu in the "Save" Dialog before you click on the "Save" button. After clicking on the "Save" button, the file will be saved in its new format.

Opening A File - The Details

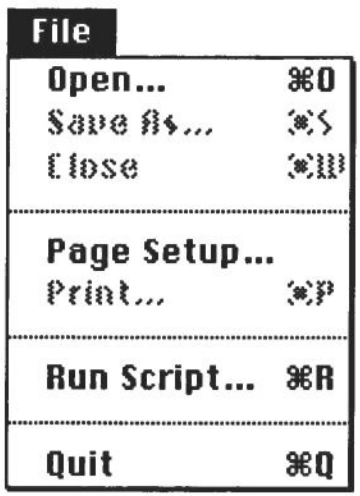


Fig. 1-4: The File Menu At Startup

When the program starts, the menu bar at the top of the screen will show the Apple Menu followed by the words File, Edit and Options.

Move the cursor over the word "Open...", and let go of the mouse button. Then you will see a list of file names. You may look at file names on other disks by

moving the cursor over the words Eject or Drive, and clicking the button once. Clicking on the arrows at the right side of the list will move the list up and down. Put the cursor on the name of the file you want, and double-click. The file will open, and CADMOVER will read its contents.

The list of file names shows ALL the files in the current folder which CADMOVER will be able to open. If you know that the file you want is in the current folder, but it doesn't show up in CADMOVER's "Open..." list, then that file is NOT in a format CADMOVER can read. See chapter 5, "File Formats" for a more complete discussion of this topic.

While CADMOVER is busy reading and analyzing a file, it will display a book with its pages turning. When the book disappears, and is replaced by the familiar "arrow" cursor, CADMOVER is ready to save your file in a new format.

Once a file has been read, that file is the Active file. You can display it, copy it to the Clipboard, or translate it to a new format. You can't open another file, however, until the active file is closed. This fact will be reflected in the File Menu itself.

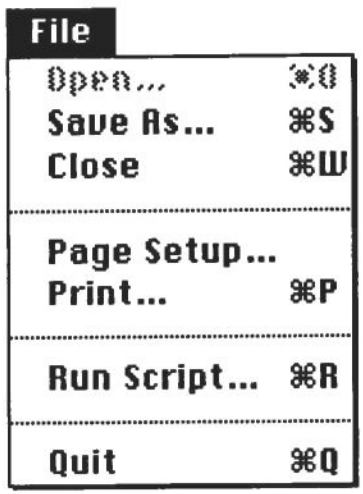


Fig. 1-5: The File Menu After Reading A File

The above figure shows that when CADMOVER has an Active file, you can save it or close it, but you can't open a second file at the same time.

Pasting A File Into CADMOVER

An alternate method of getting data into CADMOVER is to Paste it in from the Clipboard. Data pasted from the Clipboard may be translated to any format by selecting "Save As..." from the File Menu.

Copying A File Into CADMOVER

In a likewise fashion, data may be gotten from CADMOVER by first opening a file as above, but instead of selecting "Save As..." from the File Menu, select Copy from the Edit Menu. The data will be placed on the Clipboard.

Saving a File

All you have to do to save a file is select "Save As..." from the File Menu (or type command-**S**). The "Save As..." Dialog box will appear.

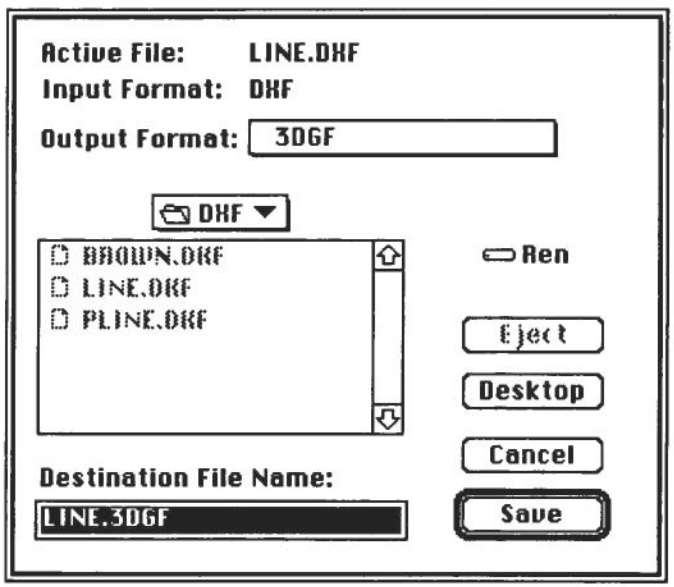


Fig. 1-6: The Save As Dialog

At the top of the "Save As..." Dialog box, the name and type of the active file will appear. CADMOVER fills in the file name box with a suggested name for the output file. The suggested name is a combination of the name of the input file and a standard suffix taken from the currently selected output format. The suffix changes whenever you select a different output file format. You can override the suggested name with one of your own choosing, simply by typing over it.

Before you save a file, you may want to set up some translation options. Options may be set at any time, except for the Units... and Scale... options, which cannot be used before the file has been read.

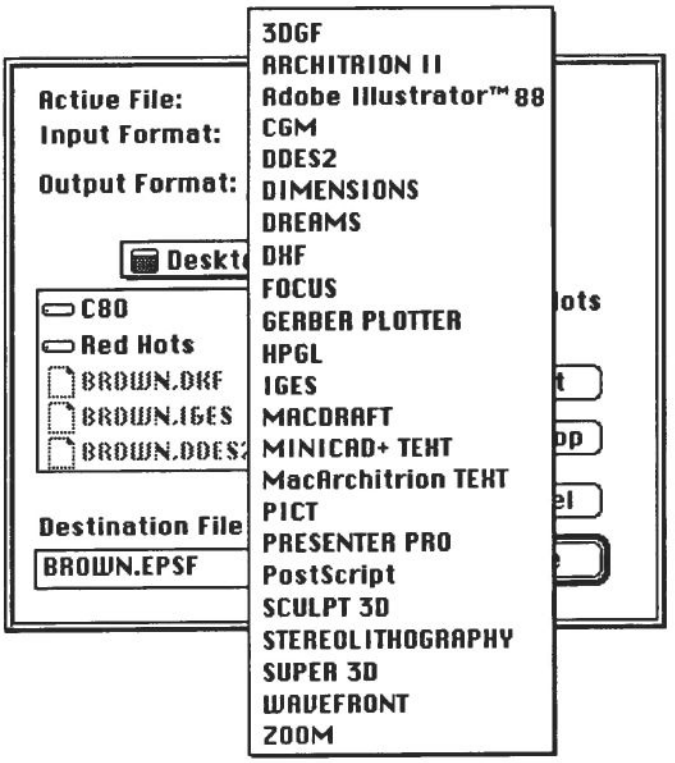


Fig. 1-7: Save As Dialog With (An Edited) Pop Up Menu.

Selecting a File Format

The dialog box which appears in response to the "Save As..." command contains a popup menu with an entry for each output format. To select the output format simply choose a format from the menu.

Variations Within File Type

CADMOVER supports various versions of several different file types, notably PICT and DXF. For these two formats, you can set up a preference that tells CADMOVER what version of the format you want to write. The "PICT Output..." and "DXF Details..." Dialogs (available from the main Preferences panel) have controls to do just that. For PICT format, versions 1 and 2 are available; unless you are running a Macintosh that does not have Color Quickdraw, in which case only version 1 of PICT is available. For the DXF format, versions 9 through 12 are available.

After your version Preferences have been set, you can then save files in PICT or DXF, and be sure of saving in the right version every time.

Adobe Output

For Adobe Illustrator output, various versions are available directly in the "Save As..." Dialog.

Naming the Output File

CADMOVER automatically suggests a name for your output file, based on a combination of the name of the input file, and the format you have selected for output. This name can be changed to any name you like, but there are very few reasons for doing so.

File Name Extensions

On various types of non-Macintosh computers, file types are identified solely on the basis of the file name extension. (The extension portion of a filename begins with the last or only period in the name, and continues through the end of the name. For example, a file named "ABC.DXF" has an extension of ".DXF." A file name which does not contain a period is said to have no extension.) CADMOVER will either add an extension if none is present, or modify the original extension, in a manner consistent with the file's format. Especially for DXF and IGES files, it is important to use the suggested extensions, because some programs will not read those files unless this convention is followed.

If you don't want to use the name CADMOVER suggests for your output file, simply type a new name in over the suggested one before clicking on the Save button. That's all there is to it.

Closing a File

Select Close from the File Menu, or just type command-W . The active file will be closed, and CADMOVER will be ready to open another file.

Important Note: Closing the Window

If you are displaying a picture of the active file, and you close the window by clicking in the close box at the upper left corner of the window, then only the window will be closed. The active file is still open. You can still retrieve the picture by selecting "Display Picture" from the Options Menu. CADMOVER has been designed this way, so that for very large files, where redraw time may be significant, you can use CADMOVER's options, system Desk Accessories, or switch to other programs, without having to wait for the window to redraw.

Pan and Zoom

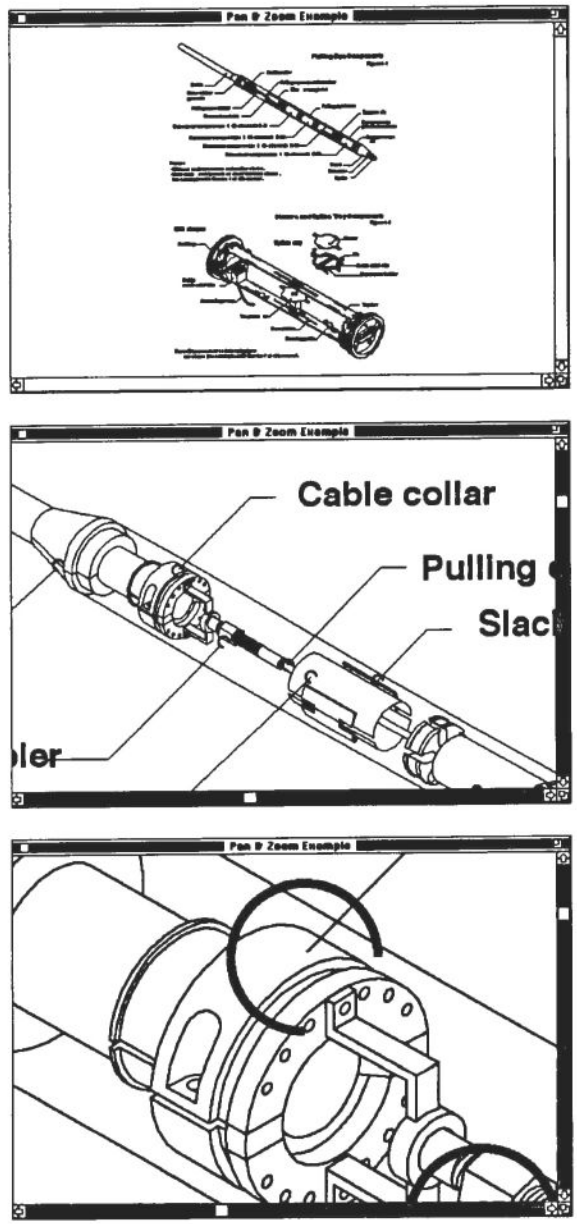
When CADMOVER displays a picture, the picture will be at "Fit to Window" scaling. If the drawing is already smaller than the window, it will be displayed at actual size.

Zoom In for Details

In order to examine detailed areas of the file, CADMOVER offers pan and zoom ability. All you have to do, using a mouse, is drag a rectangle around the part of the picture you want. When you release the mouse button, the picture will zoom in. To return the window to the normal view, simply hit the "=" (equals) key on the keyboard.

Once a picture has been zoomed in, the scroll bars become active. You can use the scroll bars to move the picture around without changing the scale. Again, hitting the "=" key will return the drawing to the normal "Fit to Window" view.

This feature is solely for on-screen inspection of the file, and does not affect the translation in any way.



Figs. 1-8: Pan And Zoom

Printing

The "Print..." option will be available after you have read in a file. CADMOVER always scales printed images to fit on one 8 1/2" x 11" page, unless the actual size of the graphic you are translating is smaller than one page, in which case it will print at actual size.

Page Setup

The "Page Setup..." command is a standard item among Macintosh applications. Depending on which printer you have chosen via the System's "Chooser" utility, you will get a dialog box which lets you specify details about how you want the printer set up. For example, you can use the "Page Setup..." command to say whether you want to print in Portrait mode or in Landscape mode. If you choose to print a document before you've used Page Setup, CADMOVER will automatically bring up the Page Setup Dialog before the Print Dialog. But after you've set up the page at least once, you may print as many documents as you like without having to change the setup again.

You Only Get One Page

No matter how big your drawing or model is, CADMOVER always scales it to fit on one page, just as it always scales drawings to fit on the screen.

High Precision

If you are printing to a LaserWriter, CADMOVER takes full advantage of the high precision drawing available on that device.

ImageWriter Support

CADMOVER also takes full advantage of the abilities of the ImageWriter. If you are running CADMOVER on a black and white machine, or if your color screen is set to monochrome mode, then CADMOVER will express gray levels as patterns on the ImageWriter, just as it does on a black and white screen.

Canceling an Operation

In keeping with the Macintosh standard, the way to cancel any operation in CADMOVER is to use the command-period keystroke combination (Command-.). The action taken depends on what was happening in CADMOVER at the time you hit command-period.

Reading a File

When you cancel a translation while CADMOVER is reading a file, the current input file will be closed, and CADMOVER will wait for a new "Open..." command.

Writing a File

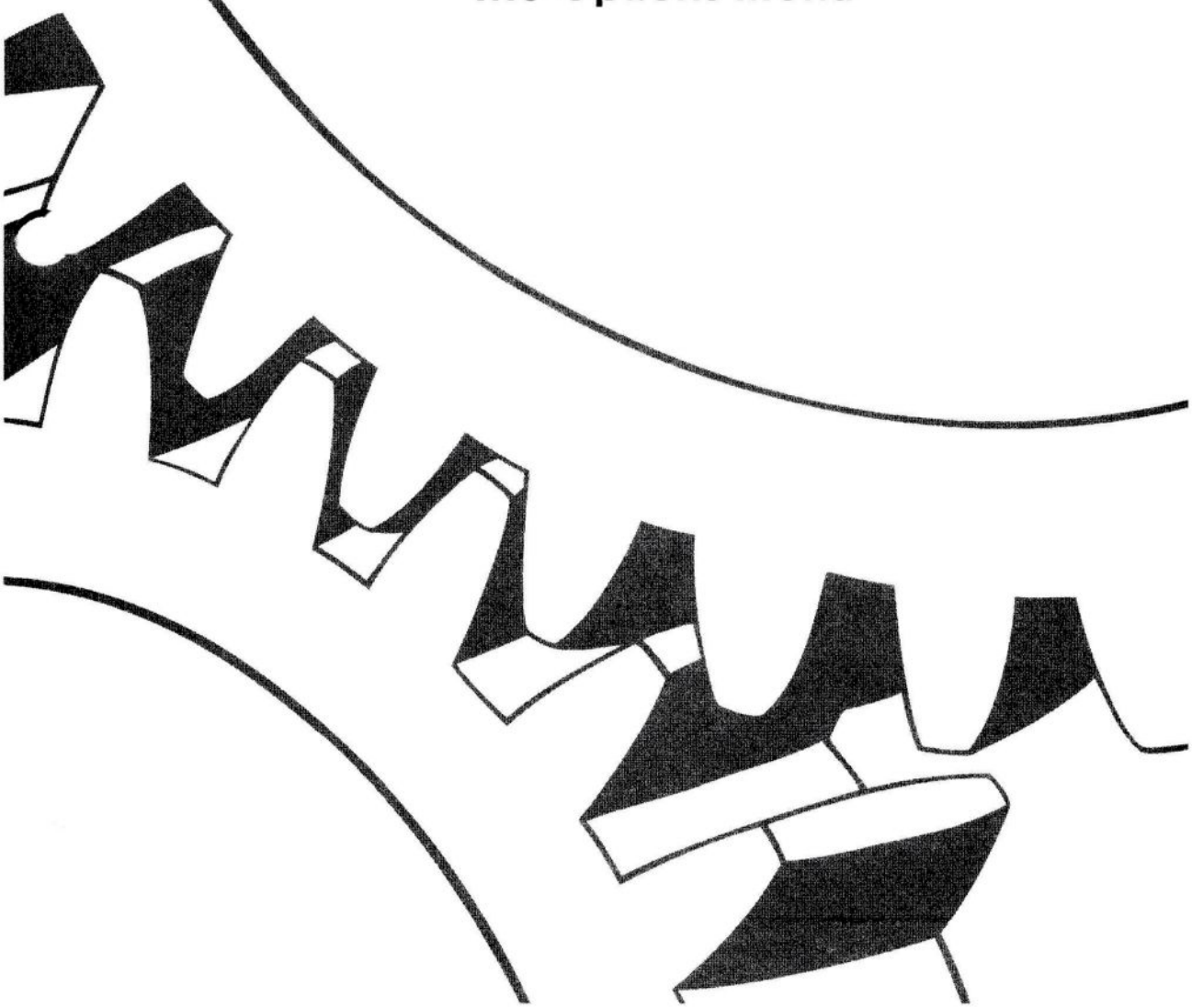
When you cancel a translation while CADMOVER is writing a file, the current translation will halt, and CADMOVER will wait for a new "Save As..." command.

Drawing a Picture

Screen operations can be lengthy for large files. If you hit command-period while CADMOVER is drawing to the screen, the drawing will halt and CADMOVER will wait for your next command, whatever it may be. The current file will still be open.

Chapter 2

The Options Menu



The Options Menu

Fig. 2-1: The Options Menu

The Options Menu lets you control the way CADMOVER will do its translations. Many of the items on the Options Menu can also be found in the Preferences Dialog; in these cases, the Preferences Dialog is used to control the default settings of the items in the menu.

The options which have corresponding Preferences are:

- Two Sided Surfaces
- PICT 1 / PICT 2
- Show Log File
- Gray Background
- Write Faces Only
- Display Picture

Important Note: Option Settings

Any options set via the Options Menu take effect immediately.

Units...

When CADMOVER starts up, this option is dimmed. It's necessary to read in a file before the "Units..." Dialog can be used. But, once there is an active file, the Units control gives you the power to control the usage of measurement units for the translation.

Using the "Units..." Dialog, you can:

- Convert from one measurement unit to another
- Convert between English system and Metric system
- Specify your own measurement units
- Override the units specification in an input file
- Override CADMOVER's assumptions about unspecified units

When you select "Units..." from the Options Menu, you'll see this dialog box:

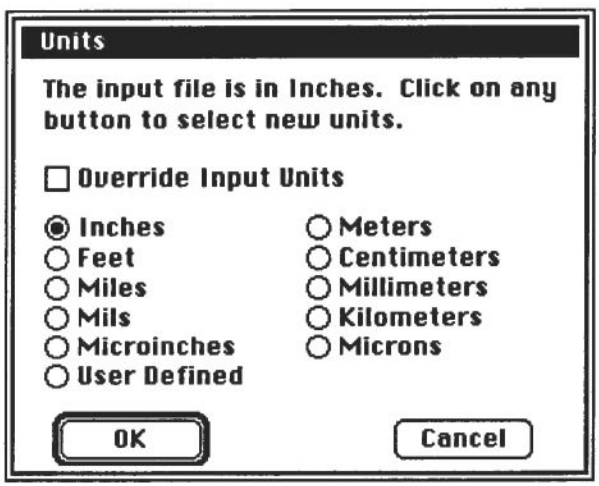


Fig. 2-2: The Units Dialog Box

Converting Measurement Units

The selected radio button will agree with the text, to show you what units have been specified for the input file. To change to a different unit when writing the output file, all you have to do is click on the appropriate radio button, and then click OK. CADMOVER will apply the correct scale factor.

The size of the drawing or model will remain the same (unless you change it with the "Scale..." Dialog). The numerical content of all coordinates will change to reflect the usage of the new unit system.

As an example, if you have a file which contains a line exactly one inch in length, and you specify conversion to Centimeters, the output file will contain a line 2.54 cm long (1 inch = 2.54 cm).

Converting Between English System and Metric System

No special controls are required. The general layout of the Units Dialog is such that units within the English system are shown in the left column, and Metric units are shown in the right column. The notable exception is the button for User Defined units, which is explained below.

Specifying Your Own Measurement Units

Click on the User Defined button to tell CADMOVER about your own measurement units. The button will be highlighted, and the Custom Units Dialog will appear.

Overriding the Unit Specification in an Input File

Click in the check box marked "Override Input Units". Now, instead of converting from the input units to your choice of output units, CADMOVER will actually consider that the input file contains the units you've specified.

As an example, if you have a file which contains a line exactly one inch in length, and you use the Units Dialog to specify Centimeters, and you turn on the Override control, then the output file will contain a line one centimeter long.

Overriding CADMOVER's Assumption about Units

This is done in exactly the same way you would override the units specification in an input file. Just check the box marked "Override Input Units". This ability can be particularly useful if you want to translate a DXF file which has been written using units not available in the "DXF Units" control on the Preferences Dialog.

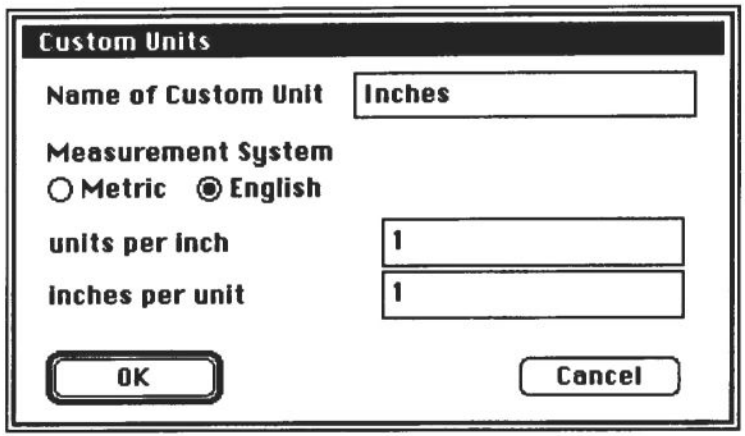


Fig. 2-3: The Custom Units Dialog Box

Using the Custom Units Dialog

There are three items to specify: (1) the NAME of your custom measurement unit, (2) whether it relates to the English system or the Metric system, and (3) how your custom unit is related to either inches or centimeters.

Select the unit system associated with your custom units. If you click on the Metric button, the messages which read:

inches per unit
units per inch

- will be changed to read

cm per unit
units per cm.

You may type into either of the two numeric boxes. This makes it easier to work in units which may be much bigger or much smaller than an inch or a centimeter. The other box will show the inverse of the number you enter.

The picture, on the next page, illustrates the Custom Units Dialog as it might have been used by an engineer in the employ of the Pharaoh Cheops, in the days when pyramids were a new thing.

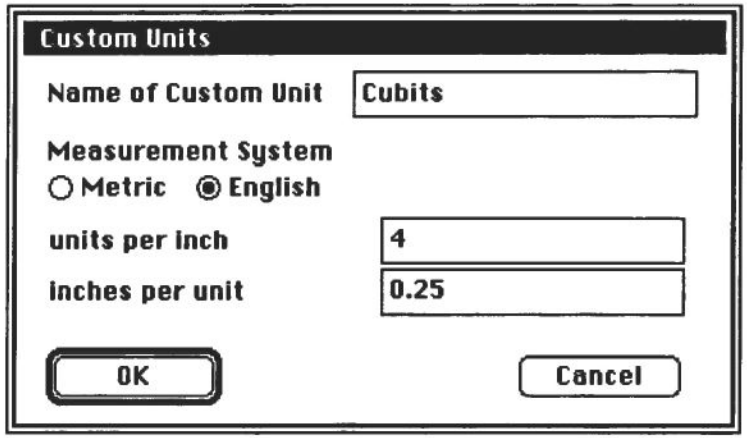


Fig. 2-4: The Custom Units Dialog Showing 4 Cubits Per Inch

Scale...

The Scale Dialog can give you very fine tuned control over the final size of the drawing being translated. The text at the top of the window shows you how big the drawing is to begin with. You can give it a new size in one of five ways:

- Specify a new width, or
- Specify a new height, or
- Specify a new depth, or
- Specify a reduction factor, or
- Specify an enlargement factor.

Aspect Ratio Maintained

The Aspect Ratio of the drawing is always maintained. Whenever you change one of the five parameters, CADMOVER changes the other four to stay in step with the new size you've specified. As an example, suppose you read in a DXF file which contains a drawing 6 inches high and 7.45 inches wide. You want to place the picture into a page layout program, where there is a space exactly 3 inches wide for the picture. There's no need to do complicated scale factor calculations. Just type a "3" into the "Width" box, and click OK. Then Save the file. The picture in the saved file will be exactly 3 inches wide.

Exact Height or Width Can Be Chosen

Using the same basic method, you can scale a picture to the exact height you need (instead of exact width). Or, if you need to know the reduction factor or enlargement factor, you can enter it directly.

Combination Method of Scaling

Using a combination of methods often achieves the goal of having a precise scale factor and a good fit, with a minimum of difficulty. First, enter the maximum allowable height or width for the output drawing. CADMOVER will fill in a new value for the scale factor. Then, simply round the scale factor up to the nearest acceptable value. The height, width, and depth will be reduced slightly, and you will have a drawing which will fit into the intended destination area, and has a known, exact scale factor.

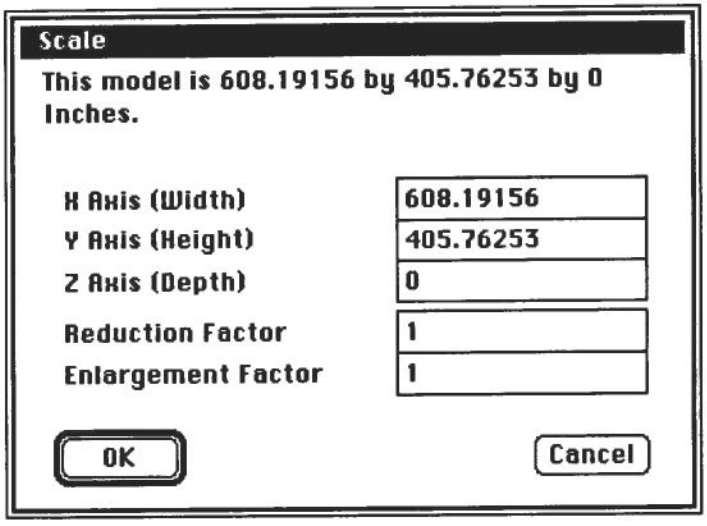


Fig. 2-5: The Scale Dialog

Figure 2-5 above shows a sample of the Scale Dialog. You may enter new values into any of the number boxes. No matter which of the five values you change, the other four will be automatically recomputed to preserve the original aspect ratio.

If any of the HEIGHT, WIDTH, or DEPTH values are ZERO, you will not be able to change that value, since that would make it mathematically impossible to preserve the aspect ratio.

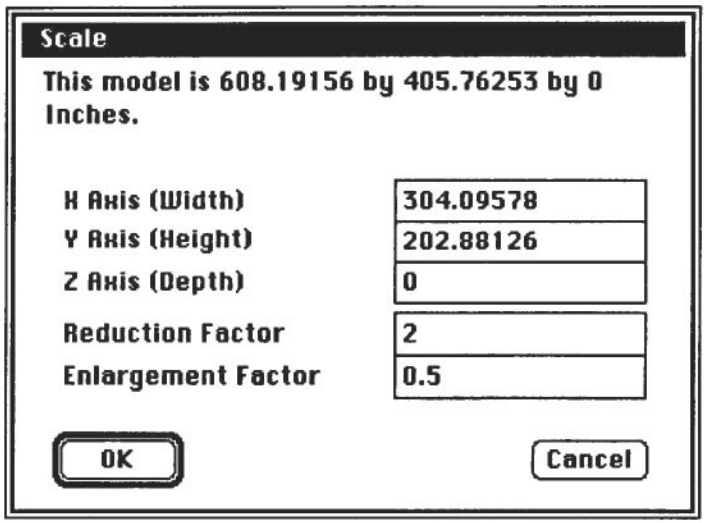


Fig. 2-6: The Scale Dialog Showing A 2:1 Reduction

Figure 2-6 shows the same "Scale..." Dialog, but this time with a reduction factor of 2 to 1 entered. Note that the height and width are half of their previous values.

Automatic Scaling Reductions for Some Formats

The scale factor set by the "Scale..." Dialog also applies to pictures copied to the Clipboard. Particularly when you intend to Copy to the Clipboard, or when you are saving a PICT file, it is important to be aware of the scale factor. It is not uncommon for files which have been created in real-world scale to be too large to represented in PICT. CADMOVER will automatically reduce such files before saving them in PICT format, and the new scale factor will be chronicled in the Log file. But results will usually be better if you set the scale yourself.

Two Sided Surfaces

You will usually want to turn this option ON when you are translating into a 3D format which considers surfaces to be one-sided. Super 3D and Zoom fall into this category. IGES, DXF, SCULPT 3D & 4D, and ENVISAGE 3D do not. CADMOVER will generate, for each input surface, two surfaces in the output file, facing in opposite directions. This makes the output file much larger, but makes it possible to do quality rendering after you've read the file to your target program. For more details see page 51 in the Preferences chapter.

Write Faces Only

This option, like the one above, is designed primarily for 3D output. It causes CADMOVER to filter objects as they are written, and it will not write out any object which is not a surface entity. It is especially useful for translating 3D DXF and Dimensions files into Super 3D.

Display Picture

The Display Picture option allows the user to control whether a picture of the input file will be displayed. When the option is OFF, CADMOVER will process files without displaying them. This can be a useful timesaver if you have a lot of files to translate, or if your files are very large. Also, if the picture window gets overlaid, it will try to redraw itself. Turning the picture OFF will avoid the redraw.

On the other hand, if you want to see the picture, turn this option ON. Closing the picture window by clicking in its close box turns off the Display Picture option in the menu. Turning the option back on will allow the picture to be redisplayed.

Gray Background

Some files read by CADMOVER contain pastel or white lines. To make it possible to see the graphics in this type of file, choose Gray Background from the Option Menu. When this option is turned on, CADMOVER will display the drawing on a gray rather then white background. The background will not be included in the output file.

CADMOVER uses a gray pattern to produce the gray background if your computer doesn't support multiple grayscale levels, or if you are running in black-and-white mode. When multiple levels of grayscale are enabled, CADMOVER takes advantage of the hardware, and uses a solid gray color as a background.

PICT 1 / PICT 2

CADMOVER imports as well as exports PICT files. The program does not distinguish between PICT1 and PICT2 when reading PICT files. CADMOVER will write PICT2 if you are running on a machine which supports color QuickDraw™, and PICT1 otherwise.

Many programs on the Macintosh are completely insensitive to variations between PICT1 and PICT2. A few programs on the Macintosh do make a distinction, usually in the way colors are handled. If you are using a program which gives different results depending on the PICT version used, make note of which version serves you better and set CADMOVER's Preferences accordingly. Various

programs on computers other than the Macintosh can read PICT files produced on the Macintosh. Most of these do not process PICT2 correctly, so PICT1 will probably be a better choice for you if you plan to take your PICT files to a different make of computer.

Once the type of PICT is selected then go to the "Save" Dialog and click on the PICT button.

3D View...

You can select any view of a 3D model for export to a 2D program by using the 3D View Option.

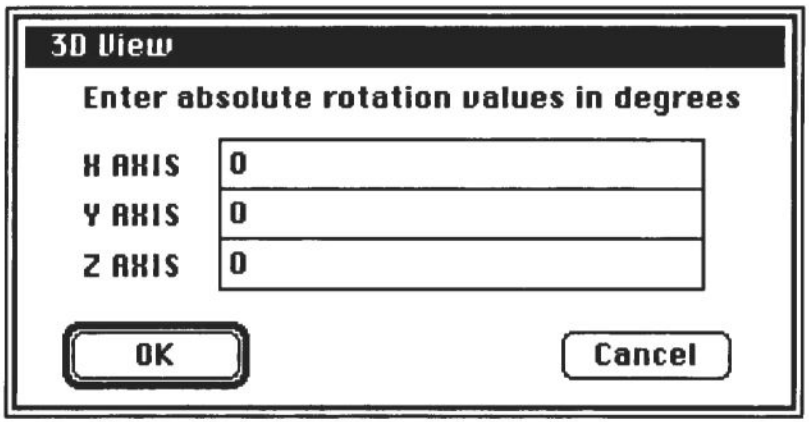


Fig. 2-7: The 3D View Dialog

Open a 3D file. After the file has been read, go to the Option Menu and select 3D View. A rotation dialog allows entry of X, Y, Z coordinates for angle of display of the 3D model. Enter the rotation angles. CADMOVER then displays the rotated view. The displayed view of the 3D model will be used for 2D export.

The X axis is the horizontal axis, Y is the vertical axis, Z is the depth. Enter angles in degrees. To change the view, choose "3D View..." from the Option Menu again, and continue as above.

Advanced 3D View Option

The 3D View Dialog gives you the ability to rotate 3D models to any point of view by entering rotation values for the X, Y and Z axes, in that order. Although this does make it possible to view your model from any angle, it is sometimes difficult to find the exact values that will produce the rotation that you want.

Specify Rotations Angles

Beginning with version 4.0, CADMOVER now offers a more comprehensive way to specify rotation angles. It is now possible to enter rotation values in any order, and more than one value can be entered for any given axis.

For example, if you are looking at a 3D object from the top, and you want to get an orthographic projection, the most straightforward way to do it is first to rotate the object 270° around the X axis to produce a side view, then rotate around the Y axis by 45° so that two sides of the object are visible, and then rotate AGAIN around the X axis, between 30° and 45°, to lean the near corner of the object downward and provide a view of the top. In order to specify this rotation, type "270x45y30x" into the "X Axis" field of the 3D View Dialog. When you use this method, the other two fields ("Y Axis" and "Z Axis") will be ignored.

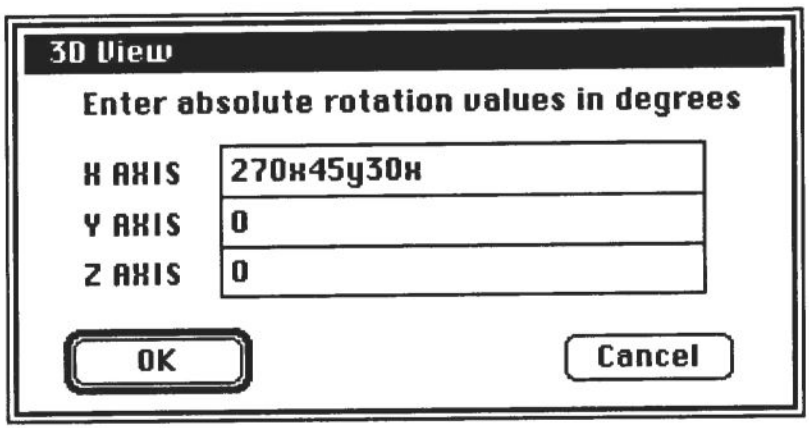
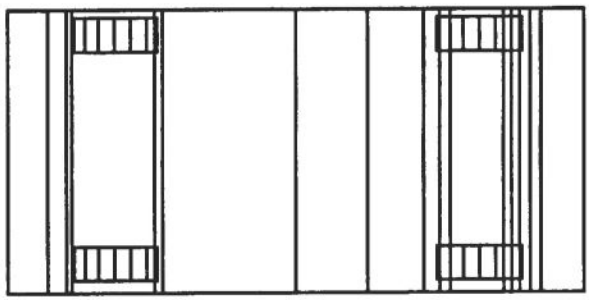


Fig. 2-8: The 3D Rotations Dialog With Advanced Rotation Input

When using this method, the various rotations will be applied in the order they are specified. This ability makes it MUCH easier to find any particular vantage point.

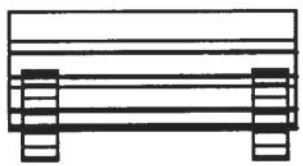
3D View Rotation Examples

The following example may help to illustrate the difference that changing the order of rotations can make. First, let's start with three views of a rather crudely constructed car:



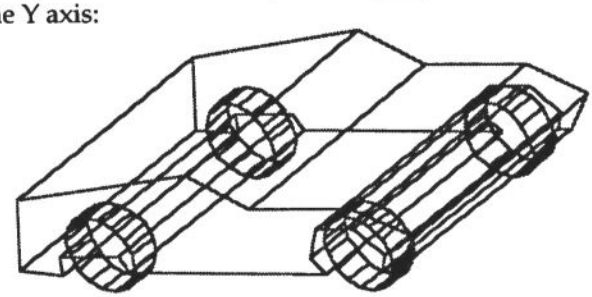
90° around Y axis

No Rotation

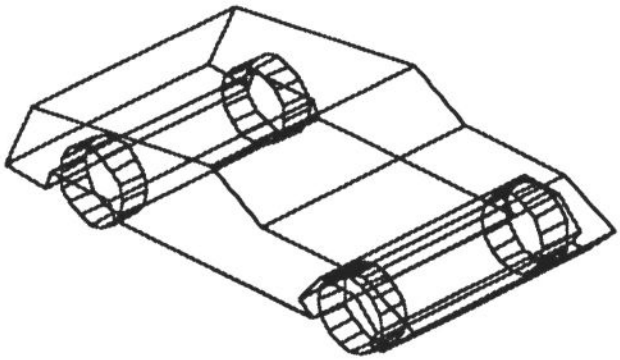


90° around X axis

Now, here's the same car, rotated first 30° around the X axis, and then 45° around the Y axis:



The next view uses the same rotation angles, but applies them in a different order, namely, first 45° around the Y axis, and then 30° around the X axis:



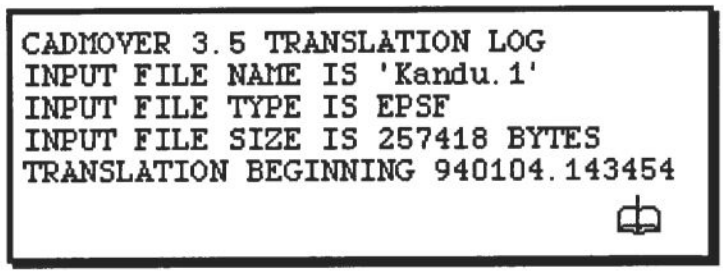
Show Log File

CADMOVER can create a log file which chronicles the translation of each file you open. The Show Log File option controls whether you see the log file as it is being written. Regardless of whether you choose to view the log file, the Keep Log File preference (on the Preferences... Dialog) determines whether the log file will be saved to disk.

CADMOVER creates a new log file for each input file you open, so long as either Show Log File or Keep Log File are set ON.

The Log File window displays a continuous stream of messages. The window does not close itself between files.

Log File Messages



The messages range from the simply informative (e.g. a time/date stamp which chronicles the processing of each file), to warning or error messages. WARNING messages inform you of unusual conditions; no error is indicated. ERROR messages relate to actual problems within the input file. CADMOVER may or may not be able to continue processing if an error condition is encountered.

Choose Text Reader...

Many of the files written by CADMOVER are text files. IGES and DXF files are both text based, and the Log files produced by CADMOVER are also text files. If you want to read these files, or print them, then you must use a text processing program. The Choose Text Reader option gives you the opportunity to inform CADMOVER what your preferred program is. Almost all text processing programs can open any text file, but when you double-click on a file icon in a Finder window, the Finder uses information in that file to decide which program you want. The Choose Text Reader option sets that information.

Choose PICT Reader...

This command is very much like the command above, except that it affects the PICT files CADMOVER writes. When you issue this command, a standard "Open FIle" type of dialog is displayed. The list in the dialog window shows only folders and programs. Choose the program which you would like to automatically launch when you double-click on the icon of a PICT file written by CADMOVER.

Line Terminators...

The line terminator option gives you the power to determine in advance the most likely destination of files written by CADMOVER. Many of the output formats, particularly IGES and DXF, can be formatted as they are being written to conform to the norms for Text files on the systems where the files will actually be used.

Macintoshes, MS-DOS computers, and most UNIX machines each have their own separate standards for Text files. There is even a fourth configuration for IGES files: fixed blocked records.

If you use CADMOVER's Line Terminator option, you can relieve yourself of the necessity of doing "system conversion" on the files before using them on other computers.

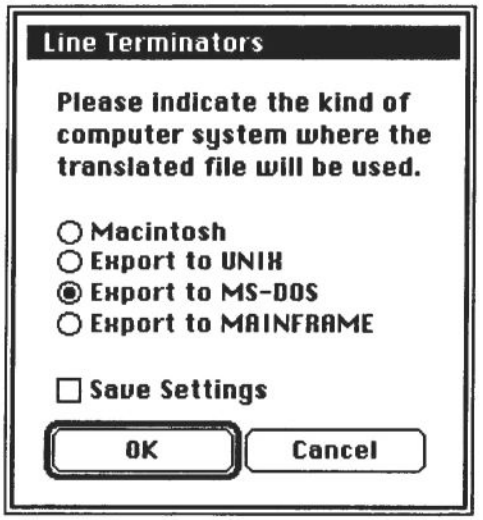


Fig. 2-9: The Line Terminators Dialog

Sound...

The "Sound..." control is a tool you can use to customize CADMOVER's choices of when to beep. Because translations can sometimes take more than a few minutes, and it's not always clearly evident when a translation is complete (especially when CADMOVER is running in the background), you have the option of having CADMOVER beep when an operation is complete.

An optional beep can be specified separately for when (1) a file has been completely read in, and (2) when a file has been completely written out. In addition, you may separately specify one set of beeper options for interactive mode, and a different set of beeper options for batch mode. In all cases, the "beep" referred to here is whatever sound you have chosen in your system to be the standard beep sound.

Preferences...

See Chapter 4 for complete information on the Preferences Dialog.

FontMap...

When translating PICT files into DXF format, CADMOVER keeps track of font information, so that different fonts may be distinguished from each other by the receiving system.

User Control

The selection of which font is used by the receiving system is under your control. If you are using AutoCAD to read the DXF files produced by CADMOVER, all the type faces will appear the same when AutoCAD first displays the drawing. To assign a different type face to a text string, first use the "List" command in AutoCAD to find out what the "Style" of the text is. Then issue the "Style" command, and type in the style name that was shown by the "List" command. AutoCAD will then ask you to select a font file for that style. Select an existing font file. After that, there will be a number of prompts having to do with details of the type style. You can just hit the carriage return for all of these prompts. After the last prompt, the screen will refresh, and you will see that all of the text in the style you have been working on displayed with its new text face.

Why Can't Cadmover Assign The Font Files?

For two reasons. First and foremost, there's no guarantee that the receiving system will have any particular AutoCAD font file on it. And if AutoCAD tries to read a DXF file which names a font file that's not on your system, it will issue an error message and discard your drawing. The second reason is to provide the maximum amount of user control over the relationship between Macintosh fonts and AutoCAD fonts. There are no clear guidelines, and since different users have different needs, it is up to each user to determine which AutoCAD font is the best match for each Macintosh font used.

Formats With Font Information

The formats that contain font information (other than Macintosh-native formats) are CGM, DXF, HPGL, and IGES. CADMOVER will enable the Font Mapping Menu selection after you have read in a file in one of these formats.

Styles vs. Fonts

DXF files contain text styles, which in turn refer to fonts, CGM HPGL, and IGES contain font codes. For the purposes of font mapping, CADMOVER treats DXF styles the same way it treats font codes in CGM, HPGL, and IGES. So at least for the purpose of this discussion, no further distinction is made between fonts and styles.

Importing To Macintosh

When you use CADMOVER to translate foreign files into many of the formats which are native to the Macintosh (e.g. PICT format), you can describe a mapping from their fonts into Macintosh fonts, by using the FontMap Dialog.

How To Get The Dialog

The FontMap Dialog is only available when there is a file open, whose format is CGM, DXF, HPGL, or IGES. Find the "FontMap..." entry in the Options Menu. If you don't see that item on the menu, there is no font information to be mapped.

How It Works

The dialog contains two popup menus. The menu on the left shows you the text styles which are defined in the input file. The menu on the right shows you the fonts which are currently available on your Macintosh.

First, select a font from the left hand menu. The right hand menu indicates the Macintosh font which will be used to represent that style. Normally, CADMOVER assumes that all foreign fonts will map into a Courier font. If your Macintosh doesn't have Courier installed, CADMOVER will assume that it should use a Monaco font, which is always present. These fonts are chosen because they are both monospaced fonts, i.e., each letter in the font takes up the same amount of space in the line.

Next, select an entry from the right hand menu. The Macintosh font you select will be associated with the input font which is currently showing in the left hand menu. Then click the OK button. If the picture window is open, the graphic will be redrawn with the new font information.

You Decide

Since there are no guarantees that any particular Macintosh will have installed fonts which resemble the fonts in any given foreign file, it's up to you to assign fonts which provide a good match to the original styles. In any case, Courier is a good general match, and CADMOVER will do a good job of matching the font size regardless of which fonts you have chosen.

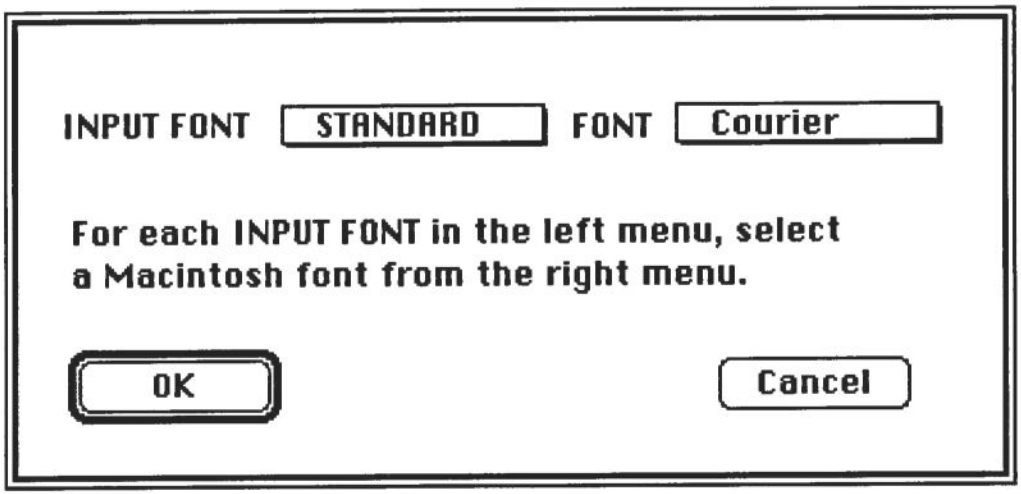


Fig. 2-10: Above Shows The FontMap Dialog As It Initially Appears.

Figure 2-10 above shows the FontMap Dialog as it initially appears. The left hand menu selects the input text style being converted, and the right hand menu shows the Macintosh text font that the style will be converted into.

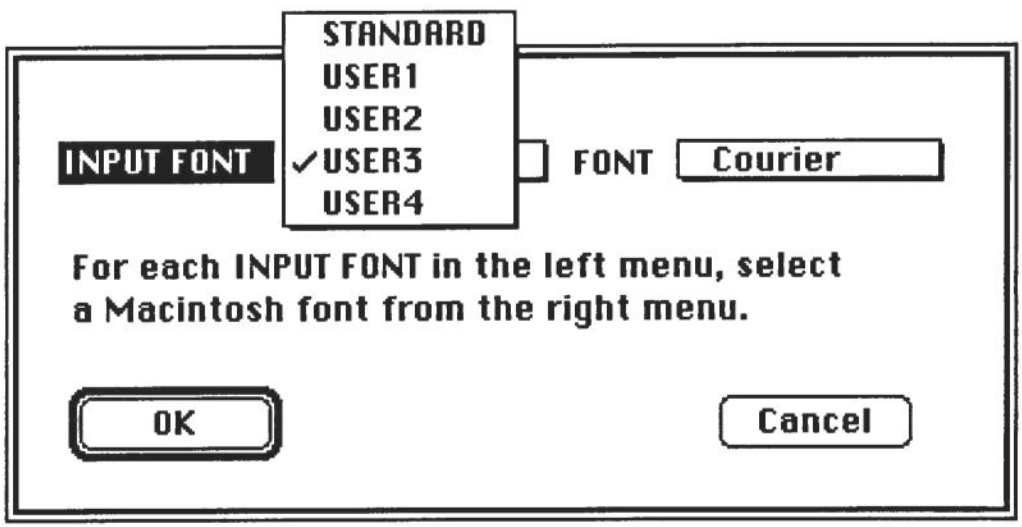


Fig. 2-11: The FontMap Dialog Showing A List Of Input Text Styles.

On the previous page, Figure 2-11 shows the same dialog with the left hand menu popped up. The exact contents of the menu will depend on the file being converted.

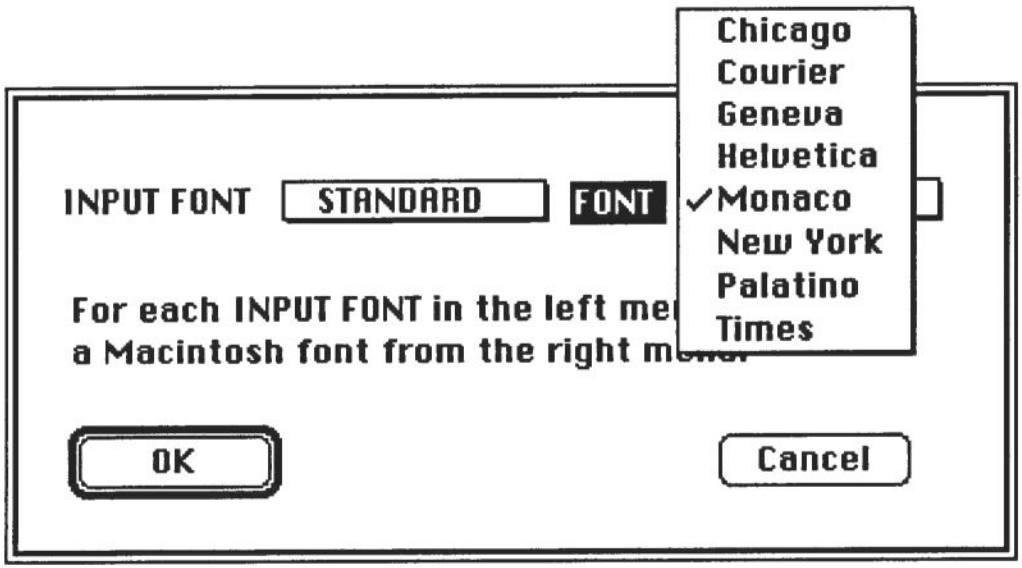
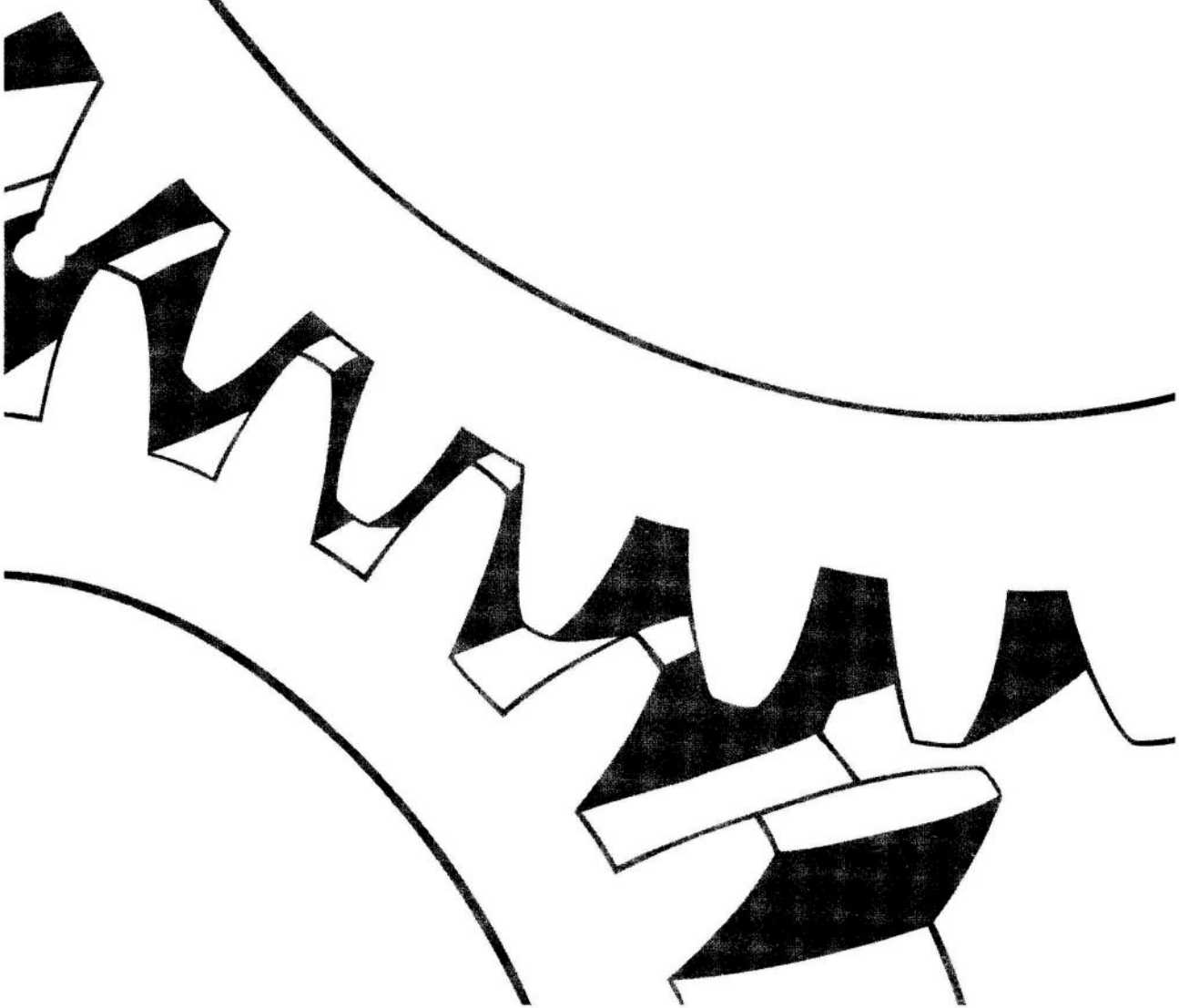


Fig. 2-12: Shows The Dialog With The Right Hand Menu Popped Up.

Above, Figure 2-12 shows the dialog with the right hand menu popped up. The exact contents of the menu will be determined by what fonts are installed on your Macintosh.

Chapter 3

The Lines Menu



The Lines Menu

The Lines Menu does not always appear; its presence is dependent on the contents of the file you are translating. But when it does appear, it is there to offer you a way to control the mapping of dashed line patterns into MiniCad+.

User Defined Dashes

A number of exchange formats, including IGES and DXF, support the notion of user defined line dashes. When user defined dashes are found in an input file, CADMOVER creates the Lines Menu, with one entry for each different dash style.

User defined dashes can express any combination of dashes and spaces. Each individual dash segment can be any length, and the spaces between dashes can also be any length. A dash style can contain any number of line segments and spaces.

CADMOVER can map user defined line styles into the predefined set of styles offered by MiniCad+. But since the number of possible user defined styles is effectively infinite, the onus falls on the user to tell CADMOVER which user defined styles correspond to which predefined styles. That's the purpose of the Lines Menu.

The Lines Menu

When the Lines Menu is present, it shows all the user defined dashing styles contained in the input file. Select a line style to map over, and the Line Mapping Dialog window will appear. Use the dialog window to associate a predefined line style with the style you have chosen.

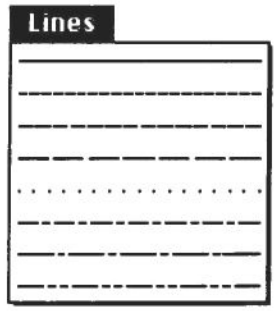


Fig. 3-1: Sample Lines Menu

When the Line Mapping Dialog (Fig. 3-2) first appears, it will show the name of the selected line style (if there is a name), and a sample of that line style, which should look identical to the style selected from the Lines Menu, except that it is drawn in a bolder stroke. The line that runs across the lower half of the dialog window shows you the currently selected output line style. In general, if no mapping has been specified, then CADMOVER will produce a solid line (no dashes).

All you have to do is select a target line style from the popup menu entitled "MAP TO:". When the popup menu appears, it will show you what predefined styles in MiniCad+ are available . When you let go of the mouse button, the mapping will be established to the style you've chosen.

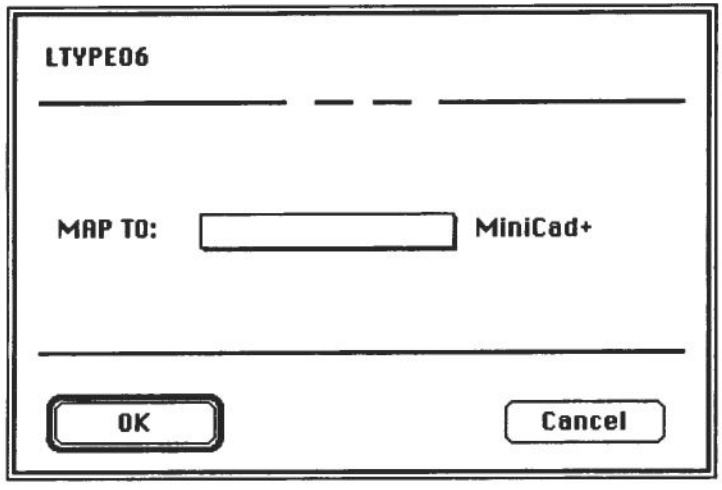


Fig. 3-2: The Line Mapping Dialog As It First Appears

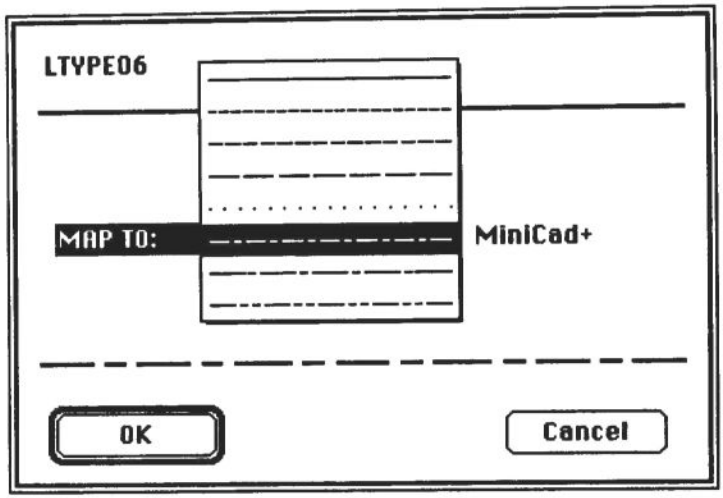


Fig. 3-3: The Line Mapping Dialog With Its Popup Menu Up.

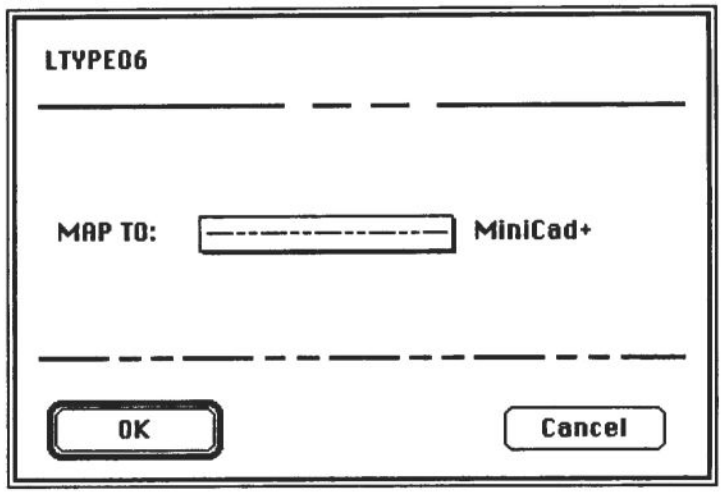


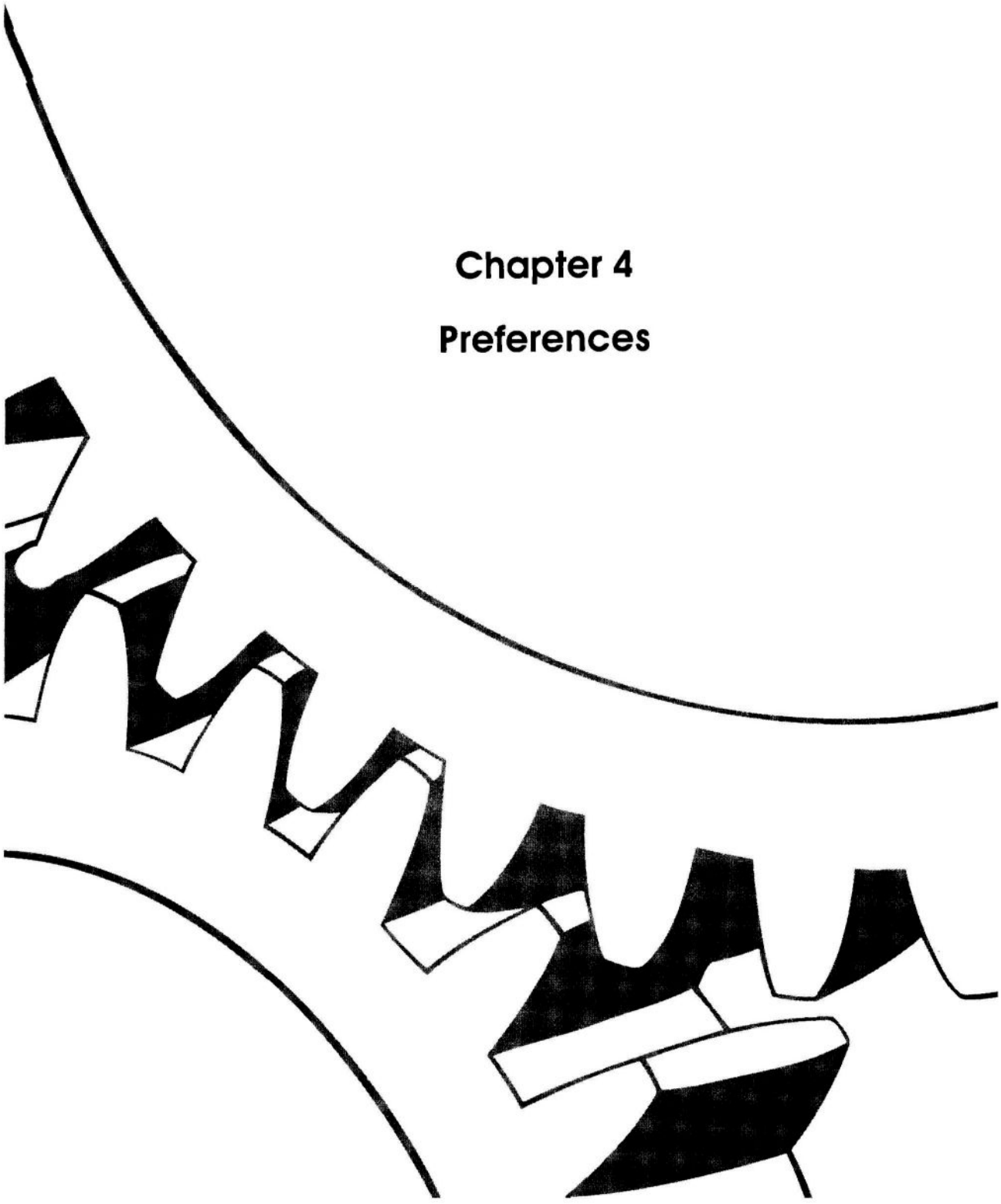
Fig. 3-4: The Line Mapping Dialog After Making A Selection

Figure 3-3 shows the Line Mapping Dialog with the popup menu visible, and a new choice selected. Then, Figure 3-4 shows that the line drawn across the lower half of the dialog window now corresponds to the entry selected from the popup menu.

Lines in the input file which are dashed according to the user defined style "LTYPE06" will be translated to MiniCad+ as the lower line in the dialog box (Figure 3-4) shows.

Figure 3-4 shows the predefined Line Style Selection Menu for MiniCad+. The selection is made based on the visual appearance of the dashes. The input style (the heavy line across the top) appears to be most closely matched by the sixth MiniCad+ line style (shown highlighted). When the mouse button is released, the connection is established.

If you click the OK button to accept the mapping, and then subsequently select the same line style from the Lines Menu, the lower heavy line will still show the sixth MiniCad+ style, indicating that a line style mapping has been established.



Chapter 4

Preferences

The Preferences Dialog

The Preferences Dialog gives you the ability to set many of the options available in CADMOVER to your customized configuration. The dialog box is pictured below:

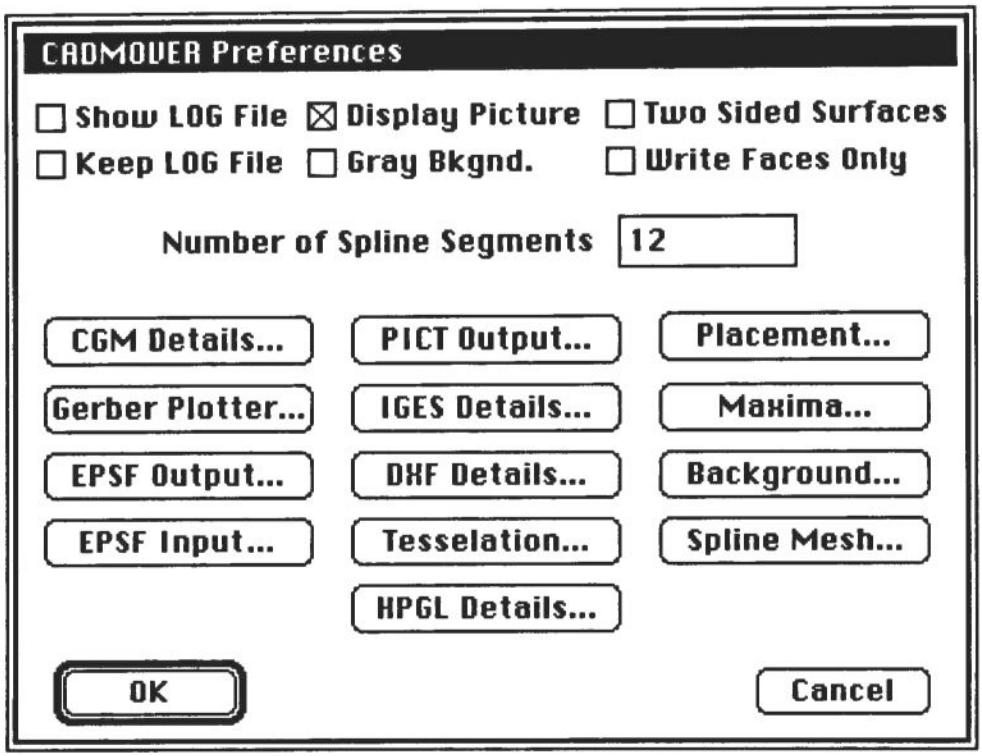


Fig. 4-1: The Preferences Dialog

The standard Preferences are shown. Use the various controls to customize CADMOVER for your own translation needs. Values you set using this dialog box will be used each time you run CADMOVER. The button controls present more detailed dialogs for each area of interest.

Show Log File

When this preference is checked, CADMOVER will display its Log File data for each file you translate. The Log File shows the time and date of each translation, and the name and type of every file read in and every file written out. Also, warning messages and error messages are displayed in the Log File window. The Log File will be saved to disk if the "Keep Log File" preference is checked (see below).

Keep Log File

When this preference is checked, CADMOVER will save a copy of its Log File data for each file you translate. (See the preceding paragraph.) The name of the log file will be made from the name of the input file, with ".Log" appended to the end. So, for example, if you read a file named "DRAWING6.IGS", its associated Log File will be named "DRAWING6.IGS.Log".

Display Picture

The Display Picture preference tells CADMOVER whether you want it to show a picture of each file you open. The initial setting of the "Display Picture" item in the Options Menu is taken from this check box. If you are working with a lot of files, or with very large files, the time taken to draw each file may be significant. So CADMOVER provides you with this control as a time saving device.

Gray Background

This preference controls the default setting of the Gray Background entry of the Options Menu. When it is turned ON, CADMOVER will use a gray color (or gray pattern on black and white screens) so that graphics drawn in a white color may be seen.

Write Faces Only

This preference corresponds to the item of the same name on the Options Menu. When it is turned on, CADMOVER will only write surface objects to the output file. Other data types, such as lines and curves, will not be written. This option is primarily intended to be used when translating Dimensions or Presenter Professional files into Super 3D format. It results in a cleaner looking model.

Two Sided Surfaces

This preference controls the way CADMOVER writes some 3D file formats. Many 3D programs treat surfaces as two-sided, i.e., no matter which side of the surface you are looking at, it is visible. Other 3D programs consider surfaces to have a distinct "front" and "back" side. Surfaces seen from the front are visible, but become invisible when they are turned around.

When the Two Sided Surfaces preference is turned on, CADMOVER will generate two surfaces, facing in opposite directions, for each surface in the input file. This can make the output file much larger, but the benefit is that you will get a much better representation of your 3D model.

Figures 4-2 and 4-3 show a 3D model, drawn in AutoCAD and saved in DXF format, taken to Super 3D.

The first picture shows the results obtained when the Two Sided Surfaces option is NOT used. Many of the surfaces appear to be missing.

Fig. 4-2: Two Sided Surfaces Turned Off

The second picture shows the same model, this time translated with the Two Sided Surface option turned ON.

Fig. 4-3: Two Sided Surfaces Turned On

Number of Spline Segments

This field controls the interpolation of parametric cubic and Bézier spline curves. The number represents how many straight line segments will be used to represent a single spline segment. The value 12 is a good general-purpose choice which will result in fairly smooth curves without using excessive amounts of storage or computation time. For small graphics or very gentle curves, a lower value such as 6 or 8 may be appropriate. For tight curves or very large graphics, numbers as high as 24, or even 32, may work better for you. Numbers higher than 32 generally do not improve the appearance of spline curves, and in fact may even degrade appearance.

CGM Details

CGM files are subject to wide variation in the ways that they are constructed. This is particularly true for the scale of the imported picture. Often CGM files have no particular scale at all associated with them; they are considered to be displayed correctly so long as their aspect ratio remains unchanged. This condition is known as abstract scaling.

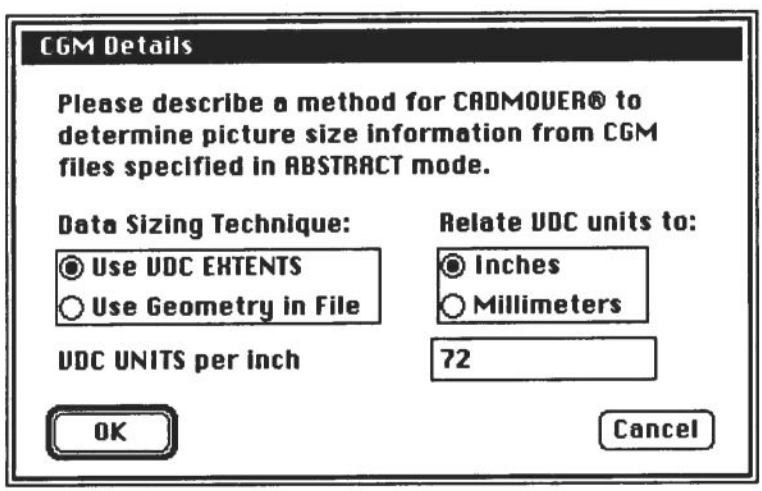


Fig. 4-4: The CGM Details Dialog

CGM Scaling on Input

Another variable in CGM files is the presence or absence of a "VDC Extents" descriptor. If such a descriptor is present in the file, it can be used to describe the scale of the picture.

CADMOVER offers several ways to control picture scaling on input. You can elect to use the VDC Extents, or ignore the extents and use the geometry in the file. If you do choose to use the VDC Extents, and a descriptor is present in the file, CADMOVER will determine the size of the picture from the VDC Units Per Inch (or VDC Units Per Millimeter) constant which you provide to the CGM Details Dialog.

If CADMOVER cannot determine an absolute scale for the file, it will generally default to a very large scale, in order to preserve the greatest amount of drawing accuracy. In such cases, you will probably want to use CADMOVER's "Scale..." Dialog to control the size of the output file.

Gerber Plotter Details...

The Gerber Plotter Preferences Dialog controls the way CADMOVER interprets Gerber Plotter files. CADMOVER will read both Pen Plotter and Photo Plotter files. Before you open any GERBER file for input, you must first be aware of whether the file is in Absolute or Incremental mode. CADMOVER gives you the ability to specify either English or Metric units, and the units per inch (or units per millimeter) can also be specified, giving you total control over drawing scale and accuracy.

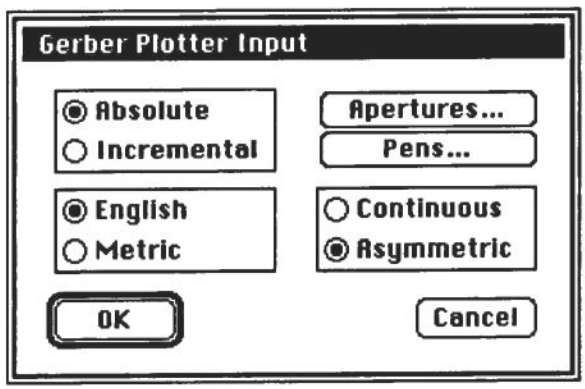


Fig. 4-5: Gerber Plotter Details Dialog

You can also specify Pen Weights for pen plotter files, and aperture sizes for photo plotter files. CADMOVER's interface allows you to specify the relationship between aperture numbers and D-codes.

Absolute or Incremental

Gerber files may be either in Absolute or Incremental mode. In absolute mode, the file contains actual coordinates describing the exact location where the pen or aperture is to be moved. In Incremental files (sometimes known as Relative files), the coordinates specify how *far* the pen or aperture is to move from its previous position. Therefore, each line is relative to the previous one.

CADMOVER accepts both kinds of files. The files themselves don't contain any information that can identify whether the file is absolute or relative. It's up to you to have the right settings in place when you read the file. It's pretty obvious when the wrong choice has been made — you'll get a picture that bears some resemblance to the aftermath of an explosion in a coat hanger factory. If that happens, just choose the other setting, close the file, and open it again.

Normally, files from the same source will always be absolute, or they will always be incremental. So long as you are translating Gerber files from the same source, you should not have to change this setting again.

English or Metric

The coordinates in Gerber files can be specified either in English units (i.e., inches,) or Metric units (centimeters.) When you click on either one of these radio buttons, a second dialog will pop up, asking you how many plotter units make up one inch (or centimeter). 1000 plotter units to the inch is a typical value.

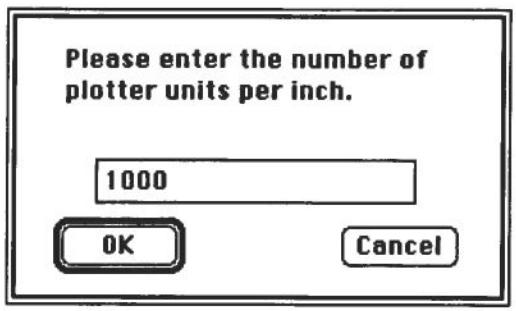


Fig. 4-6: Plotter Units Per Inch.

The appropriate Plotter Units Dialog will appear whenever you click on either the English or Metric radio button, even if that is the button that is already selected.

Continuous Or Asymmetric Line Control

These buttons control the way lines are treated as they are read in from a Gerber file.

Continuous

With "Continuous" selected, CADMOVER will build up polylines from the line strokes in the file. So long as a new line begins exactly at the point where the previous line left off, CADMOVER will be able to connect those segments into a single polyline construct. What is sacrificed in order to be able to do this is the exact aperture shapes specified for each segment. CADMOVER will assume a square aperture for polylines, since most CAD file formats do not allow different pen shapes for segments within the same polyline.

Asymmetric

When you select "Asymmetric," CADMOVER makes no attempt to connect segments. But all horizontal lines will be translated with a pen width consistent with the vertical extent of the specified aperture. Vertical lines will be translated with a pen width consistent with the horizontal extent of the specified aperture. And, diagonal lines will use the average between the horizontal and vertical extents as a pen width.

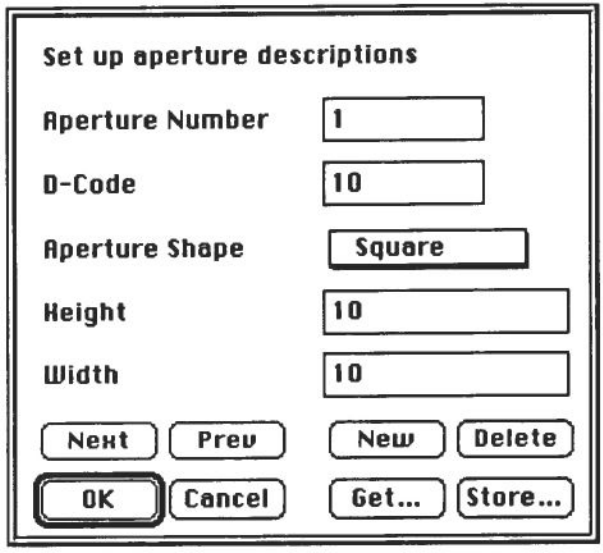


Fig. 4-7: The Apertures Dialog

Apertures...

Overview

Click on the "Apertures..." button to get the Apertures Dialog.(previous page). CADMOVER offers an unlimited number of apertures. For each aperture, you may set the associated D-code, the aperture shape, and its horizontal and vertical size. The shapes available are Square, Circle, Rectangle, and Ellipse. Of course, for Square and Circle apertures, the horizontal and vertical size must be identical. This is reflected in the fact that so long as either Square or Circle are selected via the popup menu, anything you type into the "Height" box will also appear in the "Width" box, and vice versa. For Ellipse and Rectangle aperture shapes, the height and width may be set independently.

Apertures are numbered sequentially, starting at 1. You may add new apertures by clicking on the "New" button. CADMOVER will suggest an appropriate D-Code for the new aperture. You may change it to any value you prefer, but please be careful. CADMOVER performs no validity checking of any kind on the aperture information you give it. Results are undefined if you create an aperture list with the same D-Code appearing more than once in the list.

The "New" button creates a new aperture at the END of the list. The new aperture being created will have the same particulars as the highest-numbered aperture at the time you hit the "New" button.

Important Note: Deleting Previously Set Apetures

> The "Delete" button deletes the current aperture and all higher-numbered apertures in the list. USE IT WITH CAUTION!

The "Get..." and "Store..." buttons provide a way to save and use many different sets of aperture configurations. It is likely that you may want to have a given aperture configuration in use to translate all files from one provider, and a different set of apertures for files from a different provider. That's where these buttons come in.

Clicking on the "Store..." button brings up a standard "Save As..." Dialog, letting you choose where your aperture Preferences are to be stored, and giving you the option to give a file name to your aperture file.

Conversely, once a set of aperture Preferences has been stored, you can click on the "Get..." button to get it back. When you click on the "Get..." button, a standard "Open..." Dialog appears. The dialog will list only folders and aperture preference files. Once you have selected an aperture file, click "OK", and the sorted aperture list will be read in.

AperText Application

There is even a separate application, "AperText", that converts properly formatted text files into CADMOVER aperture lists. AperText is documented in Chapter 7.

Writing Gerber Plotter Files

When you are using CADMOVER to generate Gerber Plotter files, CADMOVER will automatically build an aperture list, in text format, suitable for retranslation by AperText.

Drag & Drop

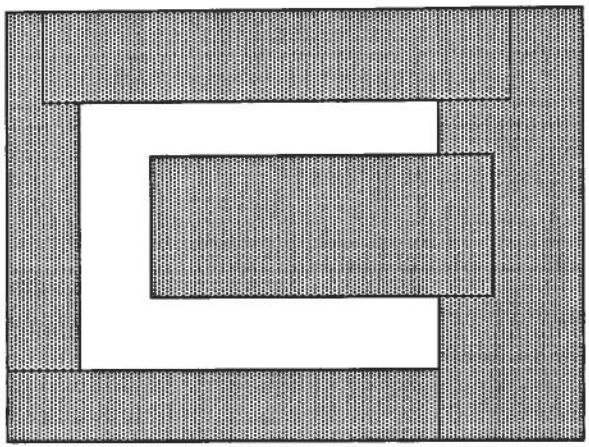
You can launch CADMOVER (under system 7) by dropping one or more document icons on the CADMOVER application icon. If one and only one of these icons is a GERBER Aperture Preference file, CADMOVER will automatically start up, using the apertures described in that file.

To do the equivalent action under System 6, highlight one or more documents and the CADMOVER icon. (All of the icons, including CADMOVER, must be in the same folder.) Then double-click on the CADMOVER icon, or type command-O.

Writing Filled Rectangles

When translating files made by applications which support an actual "rectangle" data type into Gerber Plotter format, CADMOVER will translate filled rectangles by generating a series of pen strokes, using a proper aperture width, to create filled rectangles in the generated file.

For example, the following set of filled rectangles generated in MacDraw II:



will create the following shape in Gerber Plotter format.

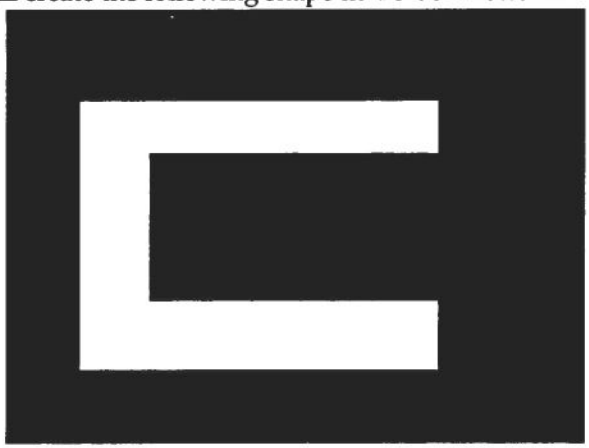


Fig. 4-8: Gerber Plotter Drawing Construction

The gray fill in the top part of the above picture is used here to demonstrate how the rectangles are overlapped. Because of the physical constraints of photoplotter technology, the second shape in Fig. 4-8 cannot be generated by starting with a black rectangle and overlaying a white "c" shape on it. That would result in an all-black area, since CADMOVER considers *all* filled rectangles to be black when generating photoplotter files.

EPSF Output...

EPSF Details give you the power to specify how line caps and line joins will be treated in files exported as EPSF.

CADMOVER offers you the option to control line joins and end caps when writing files in Adobe Illustrator format. The "EPSF Details..." button in the Preferences Dialog brings up another dialog box:

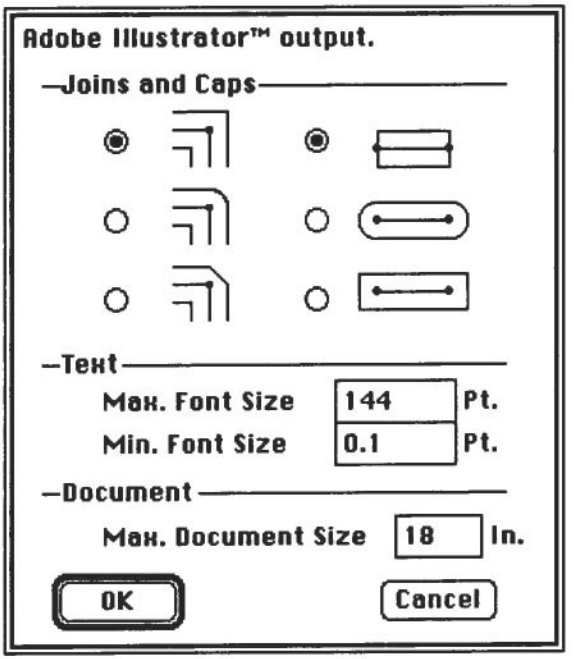


Fig. 4-9: The EPSF Output Dialog

Joins and Caps

In the above dialog the three diagrams in the left hand column indicate how lines will be joined; the three diagrams in the right hand column indicate how lines will be terminated.

It is not uncommon for lines which appear to be joined actually to be separate lines. If you have such a file, it is recommended that you use either the round or extended line caps, otherwise there will be a visible notch where two lines meet at the corner.

This picture shows how two lines will look if their endpoints are coincident but they are not actually joined, and butt end caps (top right symbol in the dialog) are selected. Selecting extended or round caps will make the lines appear to be joined.

Fig. 4-10: Lines With Butt End Caps

Text

The two fields in the Text section of the dialog let you specify the maximum and minimum font size for the output file. This can be important when writing Adobe Illustrator files, because Adobe Illustrator sets both upper and lower limits on the size of font it is willing to work with. When it tries to read a file containing text whose size falls outside those limits, it issues an error message, and refuses to read the file.

Different versions of Adobe Illustrator have different limits. Also, several other programs (such as Aldus Freehand and Corel Draw, for example, have different limitations. If you encounter such limitations in the program you use to read EPSF files, you can enter them into this dialog.

CADMOVER checks your output file as it is being generated. If the font size of any text string falls outside the established limits, CADMOVER will show you a message that gives the highest and lowest font sizes in the document as translated. The following example was made by scaling an architectural floor plan of a house down to less than postage stamp size:

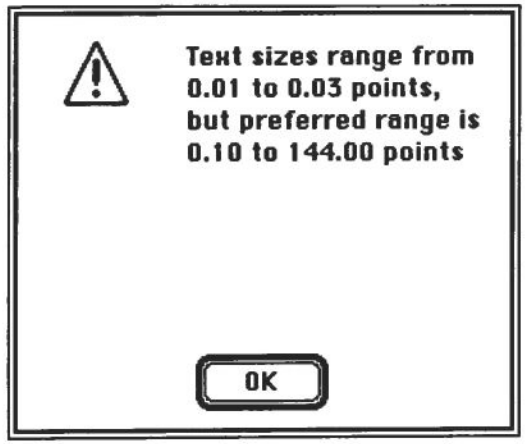


Fig. 4-11: Font Size Warning

You can use the numbers in this message to decide how to rescale your file. In this case, since the dialog tells us that there is text as small as 0.01 points, and the stated minimum point size is 0.1 points, the file should be scaled at least ten times larger.

Be aware that this is only a warning message, indicating that there probably will be trouble reading this file into another application. Continuing with the same example, this file was opened in Adobe Illustrator 3.2, and the following two messages were produced:

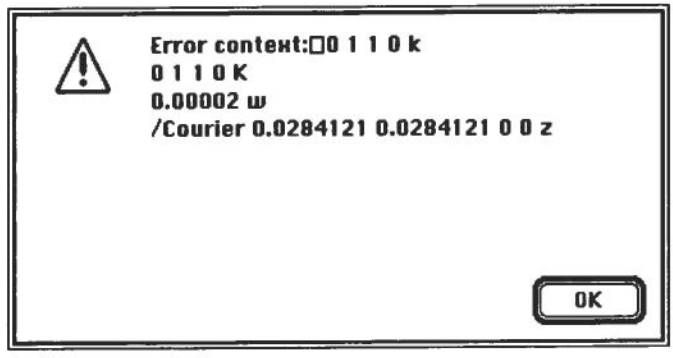


Fig. 4-12: Adobe Illustrator's First Error Message Relating To A Text Size Violation

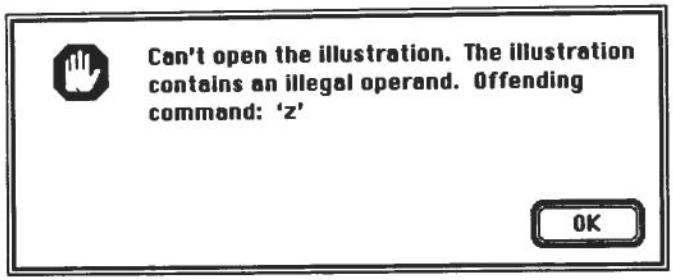


Fig. 4-13: Adobe Illustrator's Second Error Message Relating To A Text Size Violation

Max Document Size

This acts a lot like the font size limits, but it applies to the size of the entire document being translated. If you try to save a file whose size is in excess of the stated maximum, CADMOVER will offer you the following choices:

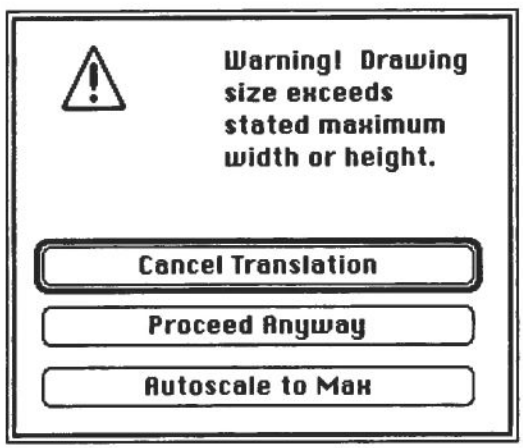


Fig. 4-14: Document Size Overflow Choices

If you cancel the translation (the default choice), you can then use the "Scale..." Dialog to resize your drawing, and then use "Save As..." to restart the translation. You don't have to read the original file back in.

You may also choose "Proceed Anyway." CADMOVER will have no trouble generating EPSF documents of practically any drawing size, but be aware that some other applications may not be able to read the file.

The third option, "Autoscale to Max," has the same effect as if you had used the Scale Dialog yourself to set the height or width (whichever is larger) of the drawing to the value you've entered into the Max Document Size field of the Adobe Illustrator Output Dialog. In Figure 4-9 page 62, the maximum document size is shown as 18 inches. Therefore, if you were to choose "Autoscale to Max," then your drawing will be scaled so that its larger dimension (height or width) is exactly 18 inches.

EPSF Input...

The EPSF Input ... Dialog can be useful when translating EPSF files into other formats, especially DXF and DDES2.

In EPSF files, curves are almost without exception represented by Bézier splines. When translating into a format that does not support Bézier splines, the curve will normally be represented by a polyline composed of short, straight line segments that approximate the path of the intended curve. The approximation can be as close as you want to make it by increasing the value of the "Number of Spline Segments" field on the main Preference Dialog. But this has the disadvantage of vastly increasing the amount of data in the output file, and, for very close work, still leaves a visibly "chunky" appearance to the curve.

By using the EPSF Special Edits, you can tell CADMOVER to translate Bézier curves as true arcs, so long as the curves actually do follow a circular path within some certain tolerance.

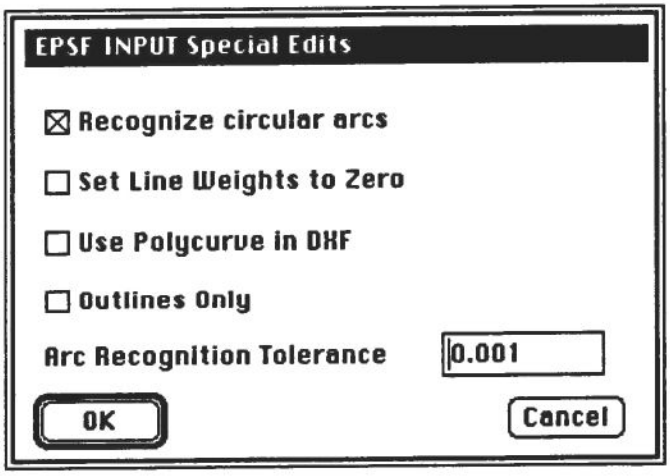


Fig. 4-15: EPSF Input Dialog

Recognize Circular Arcs

The first check box, "Recognize Circular Arcs," tells CADMOVER whether you want to generate arcs when possible. If this check box is left blank, then CADMOVER will not even try to find any arcs, and all Bézier curves will be translated in the usual fashion (i.e., as splines when possible, and as polylines otherwise). But when this check box is turned on, CADMOVER will translate splines as true arcs whenever they are close enough to the stated tolerance. The tolerance value itself, which has meaning only when this check box is turned on, controls how closely a curve must match a true arc before it is considered to be an arc. The lower the value is, the more closely a curve must match a circular path before it is translated as a true arc. In general, a value of 0.001 will be sufficient to recognize any arcs that have actually been created with the circle tool in either Adobe Illustrator or in Aldus Freehand. In some cases, the tolerance needs to be set as high as 0.075 to catch all curves that were actually intended to follow circular paths, but if the value is set much higher than that, then some curves which were not intended to be circular might be translated as true arcs. Conversely, if the tolerance is set much lower than 0.001, then some curves intended to be circular arcs might be missed.

Set Line Weights to Zero

The second check box, "Set Line Weights to Zero," does exactly what its title suggests. When the box is checked, CADMOVER will ignore any line weight information in EPSF files it reads, and will consider all lines, whether straight or curved, to have a zero line weight. This has several potential uses. For example, if you intend to drive a die cutting machine with the translated file, you may obtain superior results when the lines to be cut are infinitesimally thin.

Use Polycurve In DXF

The third check box, "Use Polycurve in DXF," tells CADMOVER to represent any arcs it has found as polycurves (instead of simple arcs) when writing DXF files. Polycurves and simple arcs are for the most part totally equivalent, but simple arcs cannot carry the line weight attribute, and polycurves can. Also, polycurves are mathematically more complex than simple arcs, and some programs that read DXF do not process polycurves correctly.

Arc Recognition Tolerance

The "Arc Recognition Tolerance" is discussed above, under "Recognize Circular Arcs". In general, if "Recognize Circular Arcs" is checked, then you will probably want to have at least one of the other check boxes turned on, especially if you are translating your data into DXF.

Outlines Only

The "Outlines Only" check box controls the way CADMOVER treats objects read in from any PostScript file. When this check box is ON, CADMOVER ignores the fill colors or patterns associated with area outlines. The outlines themselves are carried as true Bézier curves, so that actual spline information can be written out in any format which supports either Bézier or Parametric Cubic splines, such as IGES, Dimensions, or Dreams. When the check box is turned OFF, CADMOVER carries the area fills, but converts the outlines to polylines instead of spline curves. The polylines will be a good fit for the original Bézier curve. This is usually the better setting for formats which do not support Bézier curves, e.g., PICT.

Important Note: DXF Arc Simulation

If "Recognize Circular Arcs" is checked, but both of the other check boxes are off, CADMOVER will do all the computations necessary to identify arcs, but then when writing out a DXF file, it will simulate the arc by using a segmented polyline in order to preserve nonzero line weight. The same results can be obtained with less computation by leaving the first check box blank.

PICT Output...

PICT File Format Generation Options

PICT files may be generated in various ways for various reasons. The PICT Format Generation Options Dialog in CADMOVER gives you control over some of the variables. You can modify the generation of Thick Lines, Conic Arcs, Area Fills, and Border Pixels

Thick Lines

CADMOVER normally generates thick lines in PICT format by setting the correct pen size and then drawing a single line. This is usually what works best, but there are situations in which the 'custom' lines work better. The difference between the "Standard" setting and the "Custom" setting is best illustrated by example: The line on the left is Standard, the one on the right is Custom.

Surprisingly, even when "Custom" can appear to be the right choice for a file when you are looking at a zoomed-in view, the same file may look better in a zoomed-out view when "Standard" is selected. The only way to make this determination for certain is by experimentation.

Conic Arcs

With the "Standard" selection for conic arcs, CADMOVER simply places an appropriate "ARC" command into the output file. The arcs which are supported by the PICT format, however, are limited to sweeps of a whole number of degrees. This is usually not a problem for small arcs, but when the arc radius is more than an inch or so, problems can begin to crop up, as the effects of rounding off all arcs to the nearest degree become more evident. The most obvious manifestation of this problem is that when an arc is supposed flow smoothly into a straight line segment, the arc endpoint may not meet the endpoint of the line. That's why CADMOVER provides you with the option of "Custom" arcs. When you select "Custom" as the arc preference, CADMOVER carefully draws a series of small line segments to approximate the arc. The finished product will still look like an arc, even under very close scrutiny. And the endpoints of the arc will lie exactly where they are supposed to. CADMOVER's internal arc mechanism is accurate to better than one trillionth of a degree, so roundoff is not a problem when Custom arcs are selected.

Area Fills

Area Fills are divided into "Artistic" and "Mechanical/Architectural". These appellations are largely arbitrary, but they do indicate the areas of endeavor likely to influence your choice of technique regarding area fills. What the difference really comes down to is that with "Artistic" area fills selected, CADMOVER adds an extra one-pixel strip of the fill color on the right side and bottom edge of all filled rectangles. This eliminates the problem of visible borders between areas which really should be touching but are not touching because of roundoff errors somewhere along the translation line. When "Mechanical/Architectural" is selected, no such extra pixels are added. Let experience be your guide in determining which selection you want for this option.

Border Pixels

This control is a tool for controlling the amount of margin placed in PICT files written by CADMOVER. Most files look best when there is at least a small border of empty space around them, so CADMOVER offers a default value of 36 pixels, which, on most Macintosh screens, is one half inch.

Fig. 4-16: Standard And Custom Thick Lines

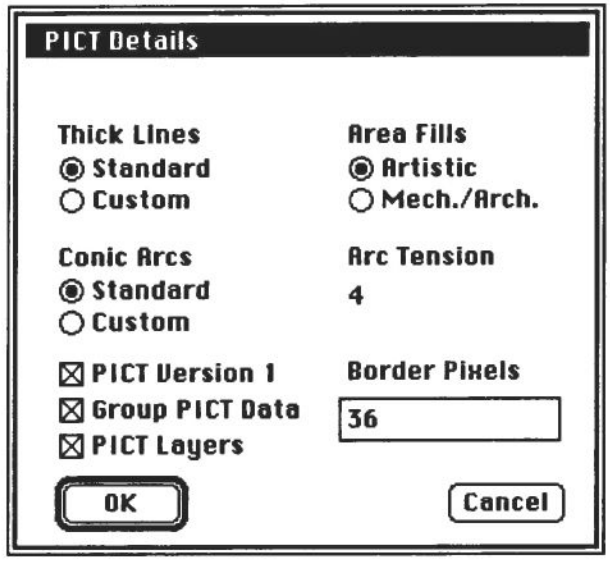


Fig. 4-17: The PICT Format Generation Options Dialog

Figure 4-17 shows the PICT Format Generation Options Dialog. Of course, if you click the "Cancel" button, any changes will not be recorded. When you click the "OK" button, then CADMOVER will apply your new Preferences, and continue to use them until you quit CADMOVER (if the "Save Settings" box is not checked), or until you explicitly change the settings again (if the "Save Settings" check box is checked.)

PICT Version 1

CADMOVER normally writes PICT files using PICT Version 2, which offers a full range of RGB color support, unless you are running on a machine which doesn't have color. Usually, there's no reason not to use PICT 2. But there are a few reasons:

- For export to non-Macintosh machines
- For use in programs which don't offer full RGB support

In general, if a program, on a machine which is not a Macintosh, offers to import PICT data, that means PICT Version 1. This preference is provided to give you the ability to communicate with such programs, while still allowing you to use the full range of RGB color when communicating with programs which support it.

On the Macintosh, if a program can read a PICT Version 1 file, it can read a PICT Version 2 file. But it might lose some color information. Colors are approximated by their closest match out of black, white, red, green, blue, yellow, cyan, and magenta. Unfortunately, it's not uncommon for light colors to be approximated by white, which makes them invisible. When the PICT Version 1 preference is checked, CADMOVER does its own approximations, and tries to avoid rendering lines as white, unless they are truly white to begin with.

Group PICT Data

This preference controls the grouping of objects within PICT files written by CADMOVER. When the option is turned ON, CADMOVER will collect all data in a PICT file into a single group, so that when you arrive in the target program of your choice, you won't have to go through the two step process of selecting all the elements and then grouping them before you move the whole picture around. Groups within groups are not affected by the addition of an upper-level group.

PICT Layers

Before you save your multi-layered file in PICT format, for each layer select all objects and group them together. Then be sure that the PICT Layers preference is turned on before you open the PICT file in CADMOVER. CADMOVER will create a new layer in the output file for every top-level group in the input file, as long as you save your file in a format which supports layers. CADMOVER will also make a separate layer for all objects that are not grouped at all, if there are any.

A top-level group is simply a group which is not contained inside of any other group. CADMOVER does not create layers for groups which are nested inside other groups.

Important Note: Layer Creation

> You can even use this feature to create layers where there were none before. If you have a picture in a program which does not support layers, simply group objects together into the layers which you would like CADMOVER to produce.

IGES Details...

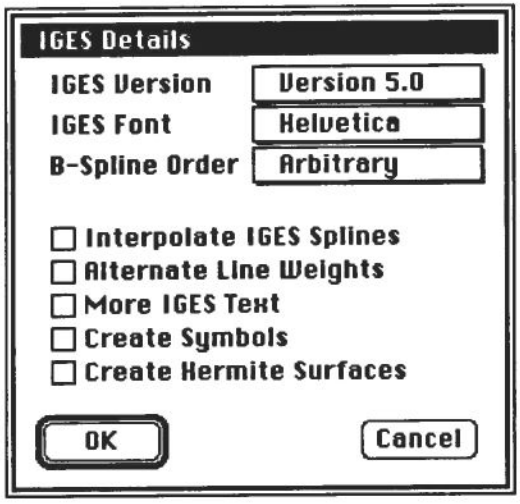


Fig. 4-18: The IGES Details Dialog

IGES Version

You may select IGES Version 3.0, 4.0, 5.0, or 5.1 for the IGES files CADMOVER writes. CADMOVER preserves the full amount of graphic information regardless of which version of IGES you choose to create.

IGES Font

The IGES standard recognizes a number of text fonts. The IGES Font control allows you to specify a preferential font for IGES files written by CADMOVER. The available fonts are:

Standard Block
LeRoy
Futura
Comp 80
News Gothic
Lightline Gothic
Simplex Roman
Century Schoolbook
Helvetica
OCR-B
Symbol Font 1
Symbol Font 2
Drafting Font

The default selection is Standard Block. The font you select will be used for all IGES output.

B-Spline Order

The B-Spline Order control tells CADMOVER whether to assume that all B-Splines are quadratic, or to do the full processing required to compute curves and surfaces for higher-order B-Splines. When the control is set to "Quadratic", processing is very quick, and will produce correct curves for the use of many Macintosh applications. If splines of higher order are encountered, the paths they follow will be approximate. This often produces acceptable, if not perfect, results. When the control is set to "Arbitrary", CADMOVER brings some considerably more complex math into play, which can have a noticeable effect on execution time. However, B-Spline curves and surfaces of any order can be translated with complete accuracy.

Interpolate Splines

This check box controls the way parametric cubic spline curves are represented internally. When the check box is ON, CADMOVER will convert parametric cubic splines into polylines as soon as it reads them. This generally results in faster performance. However, if you are translating IGES to a format which directly represents splines, leave this check box OFF. Performance will be a little slower, but spline curves will retain their true representation as splines.

The control "Number of Spline Segments" (see page 52) determines the density of the interpolation.

Alternate Line Weights

The "Alternate Line Weights" check box is there to help overcome discrepancies in the way different manufacturers encode line weights in IGES files. Basically, if you translate an IGES file, and you find that lines are either far too thick or far too thin, change the state of this check box and try again.

The PICT format does not support the concept of layers. But there is a way to preserve layer information when you use CADMOVER to translate a PICT file which was created by an application that does support layers.

More IGES Text

This preference may be used to overcome a problem which has plagued many IGES users: missing text. A little explanation is in order.

The IGES Standard, as defined by the National Institute of Standards and Technology, has been subjected to a wide variety of interpretations. As a result, unfortunately, IGES now comes in "flavors," each with its own peculiarities.

If you have an IGES file, which you know contains text, and some (or all) of the text is missing from your output file after you've translated the IGES to something else with CADMOVER, then try turning this preference on. CADMOVER will change the way it looks for text in the IGES files it reads. In general, if you have an IGES file which exhibits this "missing text" syndrome, it's likely that all IGES files from the same source will have the same problem. Once this preference has been checked, CADMOVER will treat all IGES files as "missing text" files, until you turn off the preference.

Don't turn on this preference, however, if your IGES files are not missing any text. Using "More IGES Text" when it's not needed will almost certainly cause too much text to appear; a single text note may appear multiple times and at various places in the drawing, when it should only appear once.

Create Symbols

This check box allows CADMOVER to use predefined symbols and instancing (IGES Entity Types 308 and 408). When this check box is OFF, CADMOVER will "explode" all symbols into their component parts. But when the check box is ON, CADMOVER will generate IGES files which may contain subfigure definitions and instances.

Create Hermite Surfaces

This check box tells CADMOVER how to deal with parametric cubic (Hermite) surfaces. When the check box is OFF, CADMOVER will convert Hermite surfaces into a polygon mesh, according to the setting of the "Spline Mesh" control. The polygon mesh will be written as a series of Type 106 (Copious Data) entities. When the check box is ON, CADMOVER maintains Hermite surfaces as primitive objects, and will write them into an IGES file using Entity Type 114 (Hermite Surface).

DXF Details...

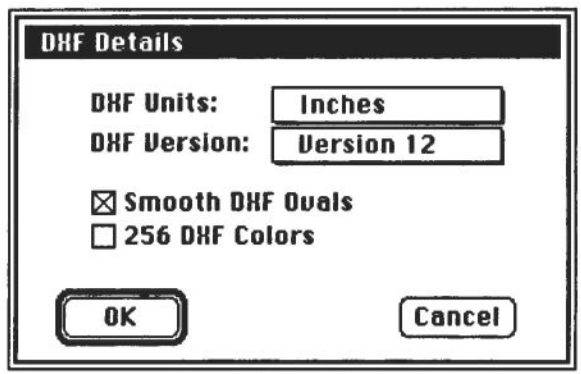


Fig. 4-19: The DXF Details Dialog

DXF Units

With this control, you tell CADMOVER what measurement units are used for DXF files which you read in. DXF files are expressed in terms of pure numbers, without reference to any measurement system. Therefore, a value of 7.5 could mean 7.5 millimeters, or 7.5 inches, or 7.5 kilometers. Unless you tell it otherwise, CADMOVER will assume that DXF files are in inches.

The control is a popup menu. Select the measurement unit which describes the DXF files you want to read. The unit you select will be used for all DXF files you read, until you change the value again.

DXF Version

This popup control allows you to specify the version of DXF that CADMOVER will write. Many programs, particularly 2D programs, cannot read DXF versions above version 9. If you are writing DXF files for such a program, select Version 9 for output. However, if you are interested in preserving full 3D information, and your target CAD system can accept Version 10, 11 or 12 DXF files, then select the appropriate version.

Smooth DXF Ovals

The DXF format does not have any construct to represent an oval directly. CADMOVER provides two different methods by which ovals may be represented in DXF files: either by using a large number of short, straight segments laid end to end, or by using a circle which is disproportionately scaled to represent the original oval. When the check box is ON, CADMOVER will used the "scaled circle" method, and when the check box is OFF, CADMOVER will generate a segmented polyline.

256 DXF Colors

The DXF standard can accommodate up to 256 colors. However, because the displays of many common video monitors are limited to 8 colors, CADMOVER offers the ability to match colors to the closest color out of a palette of eight colors: black, white, red, green, blue, cyan, magenta, and yellow. If this check box is ON, then CADMOVER will use the 256 color scheme to write DXF files. If the system destined to receive the DXF files written by CADMOVER has a display limited to eight colors, you may prefer to leave this check box OFF.

Tessellation...

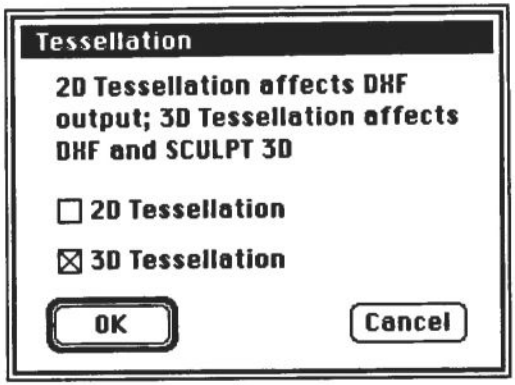


Fig. 4-20: The Tessellation Dialog

TESSELLATION is the process of tiling a plane surface. CADMOVER offers two kinds of tessellation: 2D and 3D.

2D Tessellation

2D tessellation applies to DXF output. In general, closed areas with more than four sides cannot be represented in DXF as filled areas. When you turn on 2D tessellation, CADMOVER will create a number of smaller "tiles", which it strategically places into polygons and closed areas to fill them with a solid color. Then, translations from such formats as PICT or CGM into DXF can maintain their filled areas. When 2D tessellation is OFF, such areas are translated to DXF as outlines only.

See Figures 4-21 and 4-22, on the next page, for a comparison of how graphics will appear after rereading DXF files generated with different settings of the 2D Tessellation option.

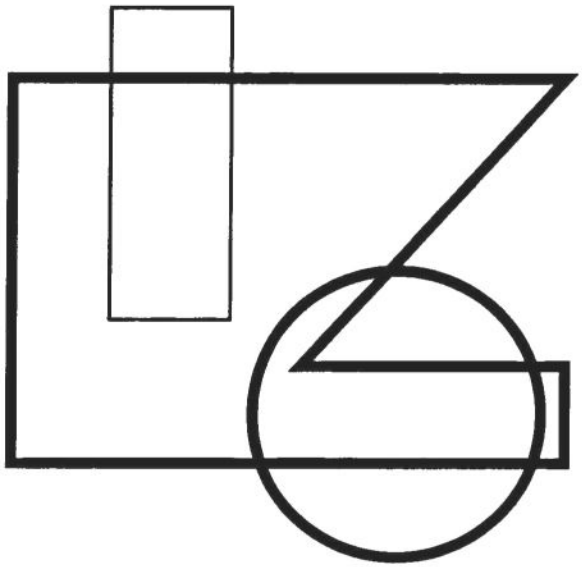


Fig. 4-21: DXF Graphic Generated With 2D Tessellation Off.

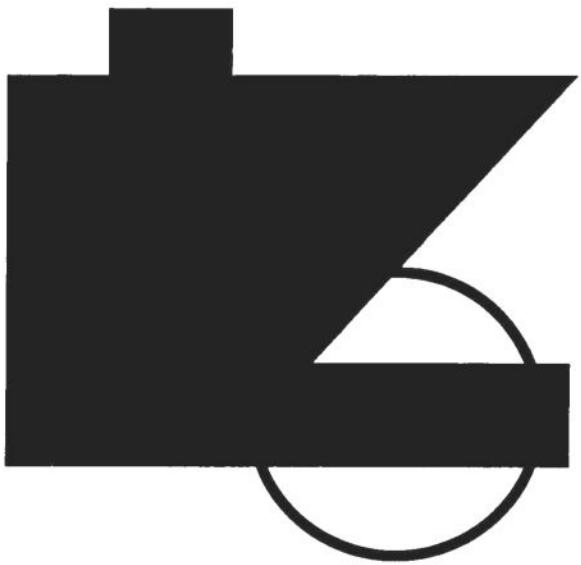


Fig. 4-22: DXF Graphic Generated With 2D Tessellation ON.

3D Tessellation

3D Tessellation is a similar issue which relates primarily to writing files in Sculpt 3D, Sculpt 4D, or ENVISAGE 3D formats. Object faces in Sculpt 3D format are always composed of triangles only. When CADMOVER writes a file in Sculpt 3D, all triangles from the input file become faces in Sculpt 3D, and all quadrilaterals are divided along their shorter diagonal, yielding two triangles in Sculpt 3D.

But when faces in the input file (in whatever format it may be) have more than four sides, the 3D Tessellation option comes into play. When 3D Tessellation is OFF, faces with five or more sides will simply be outlined. When 3D Tessellation is ON, CADMOVER divides every face of five or more sides into a series of coplanar triangles, so that complex faces will render properly as faces.

HPGL Details...

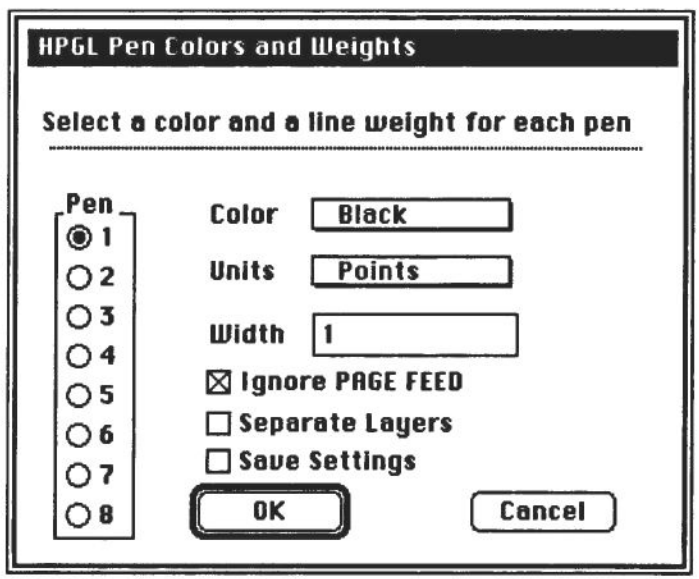


Fig. 4-23: The HPGL Details Dialog

Pen Colors

Plotter files use different pen numbers to represent different colors. There is no standard correspondence between pen number and pen colors, because an actual plotter device may have pens of any color loaded into the various carousel slots by the plotter operator. This dialog gives you the opportunity to select a color and a line weight for each of the eight possible pens. For each of the eight pens, there are eight possible colors, one of which is white. The default settings do not include

white, because few plots actually require white pens. But since you will be translating the plot file to another format, there may be reason to select white, and the option is there if you need it. The available colors are black, white, red, green, blue, cyan, magenta, and yellow.

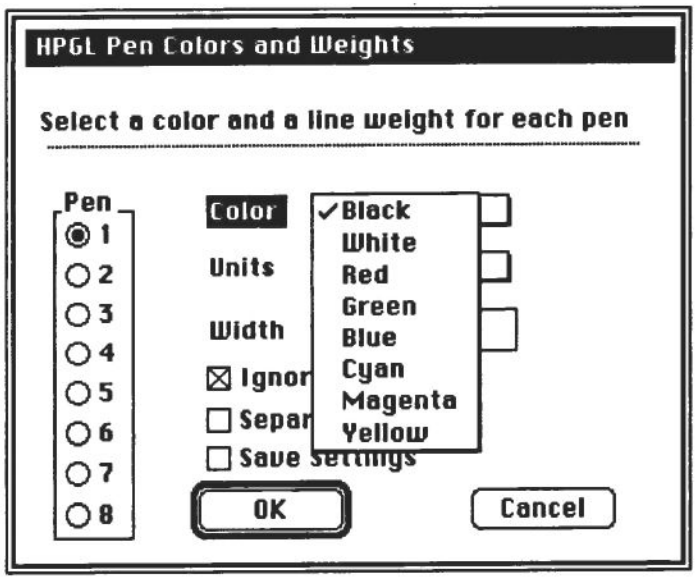


Fig. 4-24: The HPGL Details Dialog With The Pen Colors Popup Menu Open

Figures 4-23 and 4-24 shows the Pen Colors and Weights Dialog. To use the dialog, first click on a button in the left hand column to select the pen you wish to modify. Then you can select a color and a line weight for that pen.

Creating Separate Layers

HPGL files do not inherently have a layer structure. But when you are translating HPGL files to a format which does support layers, you may wish to separate the drawing into layers. When the "Separate Layers" check box is on, CADMOVER will assign drawing elements to different layers according to which pen is used. This is independent of pen color — you may assign the same color to all the pens, and CADMOVER will still segregate layers by pen number. If you do not want to separate your plot files into layers, turn off the "Separate Layers" check box.

Line Weights

Since HPGL files do not carry information about how thick a line should be drawn, it is up to you to specify line weight. Initially, all lines are assumed to be 1 point (1/72 inch) thick. To modify the line weight, first use the Units popup menu to select how you want to measure the line weight. You may select among Inches, Centimeters, or Points. After selecting the units, type a line weight into the

"Width" box. CADMOVER will use the indicated color and line weight for the selected pen.

As you select different units, the number in the Width box does not change. Be sure that the number is expressed in the units you actually want.

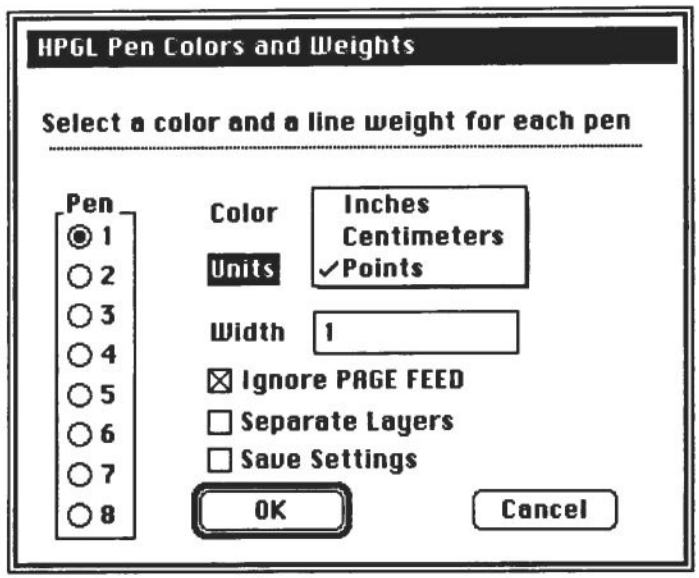


Fig. 4-25: HPGL Details Dialog With The Units Popup Menu Open.

Ignore Page Feed

In order to prevent the accidental merging of plots that are supposed to be separate, CADMOVER will stop reading an HPGL file when it encounters a 'PG' (Page Feed) instruction. This plotter instruction tells the plotter device to eject the current sheet of paper from the plotter bed, which effectively terminates the current plot. It is not uncommon, however, to find plotter files that have a 'PG' instruction near the beginning of the file, to ensure a clean sheet of paper in the plotter. When CADMOVER translates HPGL files set up in this way, if it were to stop on every 'PG' instruction, it would result in empty output files, accompanied by all-blank pictures.

If that happens to you, turn on the "Ignore Page Feed" checkbox. Then CADMOVER will skip over 'PG' instructions in HPGL files as though they were not there.

Save Settings

The Save Settings check box allows you to assign a standard set of colors and line weights to HPGL files. If the box is checked at the time you click on the OK button, then CADMOVER will remember these settings and use them every time you import an HPGL file. On the other hand, if the check box is left blank, then CADMOVER will apply the colors and weights you have chosen to the current file only, and will revert to the previous configuration the next time you import another HPGL file.

Placement...

This option refers to the placement of the translated document or model within the target coordinate system.

CADMOVER gives you the option to specify how you would like your translated documents or models to be positioned.

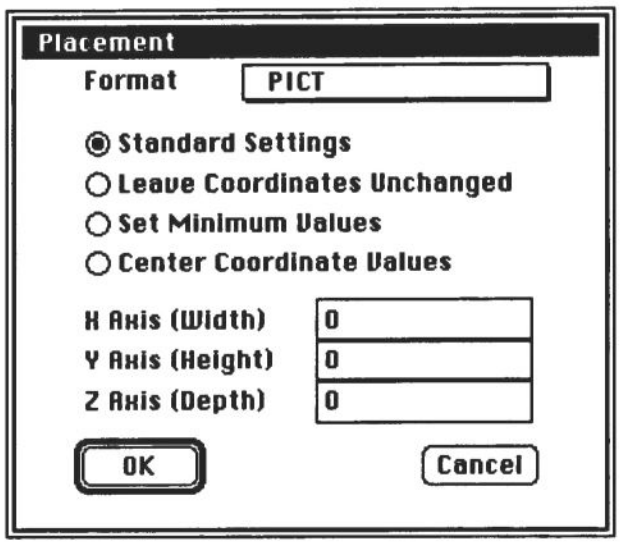


Fig. 4-26: The Placement Dialog

Important Note: Placement Is Format Dependent

Not all options are useful for all formats. For instance, if you specify that coordinates are to remain unchanged when you translate DXF into PICT, some programs may load the generated PICT image into an area that is off the page, making it inaccessible to you. In some cases, this may even cause the program which reads the PICT file to malfunction. Note: If STL files are positioned so that any coordinates have negative values, it probably will not be able to be built by a stereolithography (STL) machine.

CADMOVER's default approach to document placement is as follows:

1. For all 3D formats except STL, the model will be centered around (0,0,0).
2. For STL, the model will be placed entirely within the first octant, so that the STL file will contain no non-negative coordinates.
3. For PICT, MacDraft, and Dreams files, the file will be placed so that the upper left corner of the drawing appears in the upper left corner of the window of the corresponding application program.
4. For HPGL, the lower left corner of the plot will be placed at (0,0).
5. For other 2D formats, the document will be centered around (0,0).

There are two primary reasons for this placement scheme:

1. So that translated models and documents will appear in a valid portion of the coordinate space supported by the application that will read the files created by CADMOVER.
2. To help you find your translated document or model after you have opened it in another application. Particularly in 3D programs, it is surprisingly easy for your model to be "lost in space." Imagine opening your translated model in a 3D editor program, and seeing only a blank screen, even though a status window tells you that there are over a thousand surfaces in the current model. If you don't know what coordinate values were in use in your model, it can be time consuming to actually locate your model and position it within your program's working space.

There are four basic placement options that can be used:

Standard Placement

This is the factory default setting described above. Its action varies depending on the output format, but will generally be the "right" thing.

Leave Coordinates Unchanged

This is almost certainly the option you will want if you are translating a model from various parts files and intending to assemble all of the parts in some target program while keeping their original relative placement to each other.

CADMOVER will leave the coordinate values absolutely unchanged so long as the source program and target program use the same units of measurement (e.g., inches), and the same scale. Even if measurement units or scale do change, translations involving multiple files will generate new files with the parts having the same *relative* placement as they did before translation.

Set Minimum Coordinate Values

This setting allows you to specify placement of a document or model, based on the minimum coordinate values for each axis. The values you type in here, which are expressed in the measurement system of the translated file, will be the minima for each respective axis.

Set Center Coordinate Values

The values you type in here, which are expressed in the measurement system of the translated file, will be the centers of each respective axis.

Note that when translating to any 2D format, the settings for the Z axis (depth) are ignored, since 2D coordinate systems do not have a Z axis. When translating from 2D to 3D, the Z value for all coordinates may be set to any desired value (normally zero.)

Maxima...

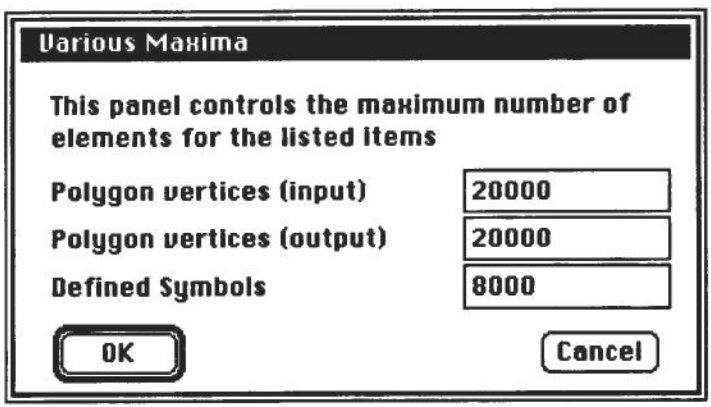


Fig. 4-27: The Maxima Dialog

In order to process large files efficiently, CADMOVER needs to allocate some storage areas before it knows how big some of these areas have to be. The assumptions that CADMOVER makes are adequate for most files, while still allowing CADMOVER to run in a modest amount of memory.

If, however, you get a message from CADMOVER that there are too many blocks, or that the maximum polygon size has been exceeded, etc., it is possible to reconfigure memory and try again.

In addition, some programs which process files created by CADMOVER have their own limitations. For instance, some applications that read Adobe Illustrator files limit the number of vertices in a polygon to 1000. CADMOVER offers you the

option to limit the number of vertices in all polygons written to an output file, so that the application you intend to use next on that file will function correctly.

Important Note: Maxima for Vertices & Symbols

As of this writing, the maxima that can be set, are the maximum number of vertices for a polygon on input, the maximum number of vertices for a polygon on output, and the maximum number of symbols (a.k.a blocks) that can appear in a file.

Background...

Fig. 4-28: Background Speed Selection Dialog

Using The Background Operation Dialog

You can select one of three speeds: Fast, Medium, or Slow. These speeds refer to the way CADMOVER itself will run. When you select Fast, CADMOVER will run at nearly the same speed it does when it is the foreground task. When CADMOVER is running Fast in the background, your foreground task may seem somewhat sluggish or slow to respond to your commands. When you set CADMOVER's background speed to Medium, it will take a little longer to translate your files, but foreground applications will run normally, quickly responding to your commands. If CADMOVER is running at Slow speed, the translations will take even a little longer, but the background operation will have no effect at all on foreground applications. Be sure to read the section in this manual about Batch Mode operation in order to get the most benefit from background operation.

Spline Mesh...

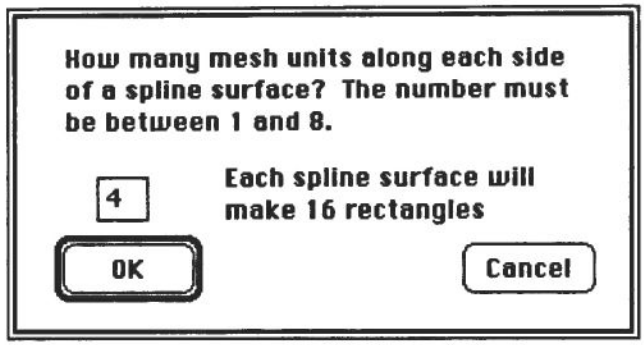


Fig. 4-29: The Spline Mesh Dialog

The Spline Mesh Dialog gives you control over how coarse or fine a mesh is generated whenever Hermite surfaces are converted into a polygon mesh. It is important to note that this control is active even if the "Create Hermite Surfaces" check box in the "IGES Details..." Dialog is checked ON. In fact, if your input file contains Hermite surfaces, you can interactively change the mesh parameter, and observe the effect on the screen, before saving a file.

Parametric Cubic Spline Surfaces

CADMOVER deals directly with parametric cubic spline surfaces. Many of the file formats which CADMOVER translates do not have a direct representation for these entities. Therefore, when translating to such a format, CADMOVER approximates spline surfaces as a collection of rectangles. The collection of rectangles which represents a spline surface can be referred to as a "mesh".

You can control the meshing density CADMOVER uses when it converts a spline surface to rectangles. The lowest density results in a small file which both translates and displays quickly. The highest density results in a much larger file, which takes significantly longer to translate and display. For rendering purposes, however, the higher density mesh gives vastly superior results.

The Mesh Density Dialog can be accessed by starting with the Preferences... command, then hitting the "Spline Mesh..." button. The dialog asks for a number between 1 and 8 (the default value is 4). The number represents the number of segments into which each edge of the spline surface will be divided. The number of individual rectangles that will be made from each spline surface is the square of this number. Therefore, at the lowest setting, no subdivision at all takes place. The entire spline surface will be represented by a single rectangle. (The word

"rectangle" is being used rather loosely here; "quadrilateral" would be more precise.) The default setting results in sixteen rectangles for each spline surface, and the highest density mesh results in sixty-four rectangles for each spline surface. From this, it is easy to see why file size and execution time may be strongly affected by the mesh density setting.

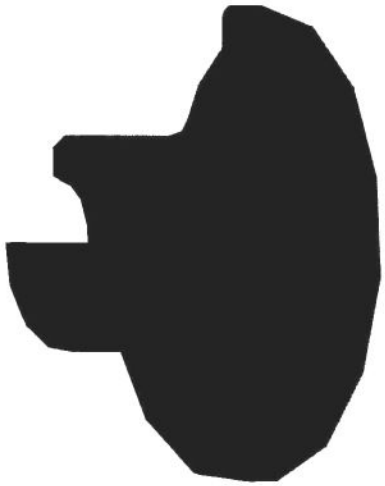


Fig. 4-30: Mesh Density =1

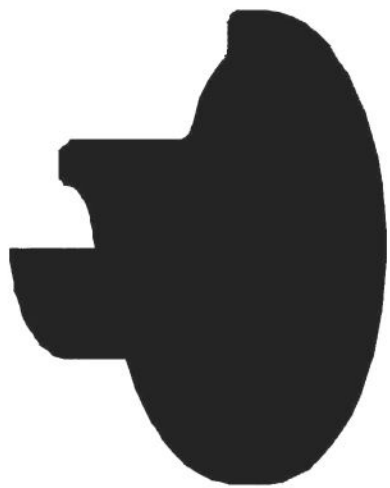
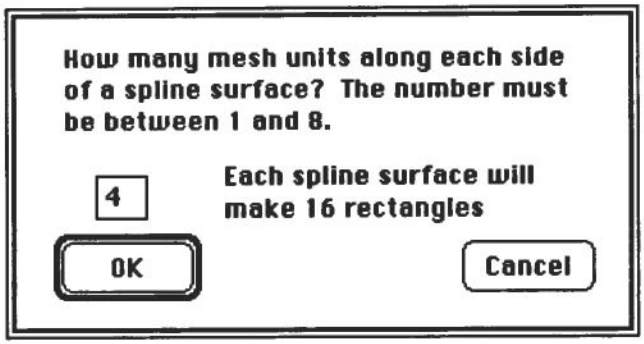
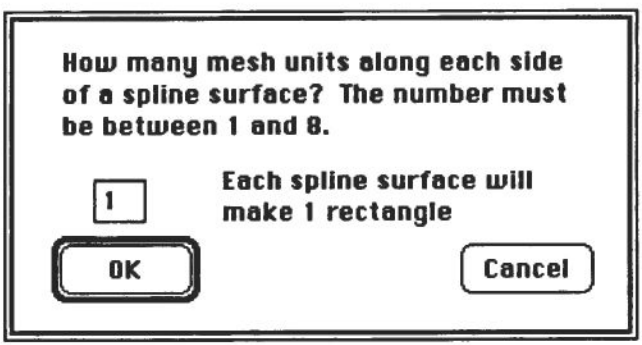


Fig. 4-31: Mesh Density = 4

The two illustrations (Figures 4-30 & 4-31) on the preceding page illustrate the effect of the Mesh Density Dialog. The top picture was done at the lowest density. The boundaries of each spline surface are visible, but there is no interior detail within the surface. The bottom picture was made from the same IGES input file, with the default meshing density (4x4). Each individual spline surface is represented by sixteen rectangles.



Figs. 4-32 and 4-33: The Spline Mesh Dialog At Its Lowest Setting, And Below At Its Default Setting.



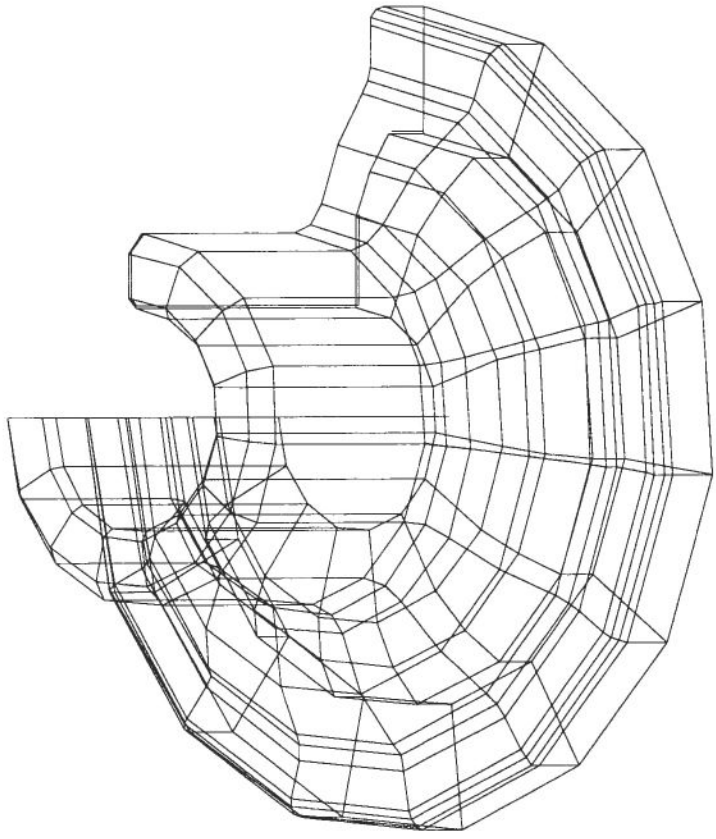


Fig. 4-34 Mesh Density = 1

The wireframe illustrations on this page and the next page also show the effect of varying the mesh density. As in the previous figures, the object was translated first with a mesh density of 1 (this page), and then again with a mesh density of 4.

The original model from which these figures were derived was produced on a Schlumberger MacBravo! system, and saved as an IGES file. CADMOVER rotated the object to the view you see here, then projected it onto a 2D plane.

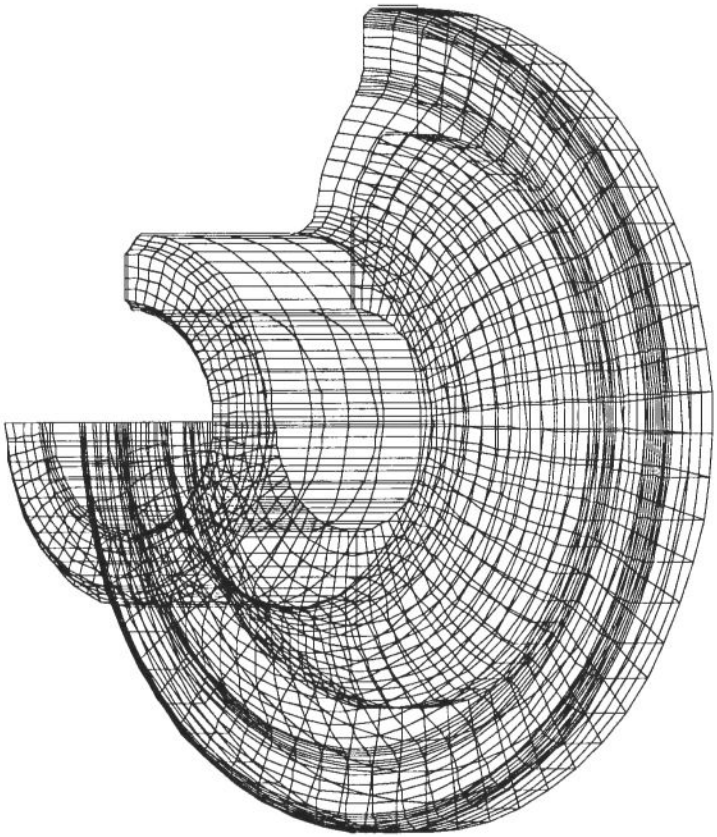
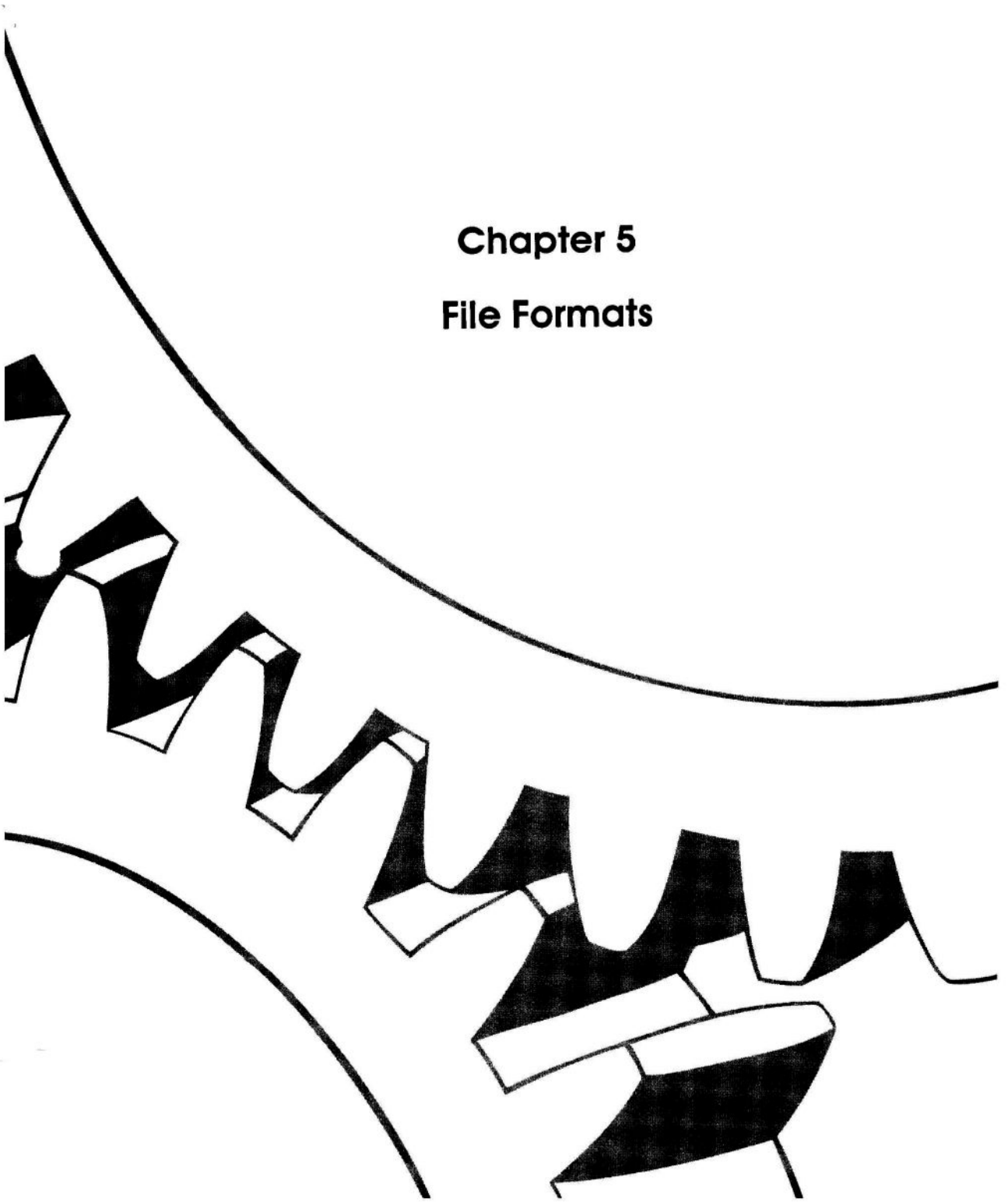


Fig. 4-35: Mesh Density = 4

Chapter 5

File Formats



File Formats

Text vs. Binary

CADMOVER deals with a wide variety of file formats. Some of the file formats are Text formats; others are Binary formats. The basic distinction between the Text and Binary formats is that Text files can be opened in text editors, word processors, etc., and the contents are generally eye-readable. Binary files, on the other hand, cannot be read or printed by a word processor.

Another difference between Text and Binary files on the Macintosh is that, in general, binary files are closely associated with one and only one program, whereas Text files are generic; i.e., it doesn't matter what program created a Text file; any program which will open any Text files at all can generally be counted on to open all Text files.

Selecting a File to Open

When you select "Open..." from CADMOVER's File Menu, a standard "Open..." Dialog box appears. CADMOVER displays the name of every file in the current folder which it believes it can process. If you know that a certain file is in the current folder, and its name is not displayed in the list CADMOVER shows you, then that file is not in a format which your copy of CADMOVER can process. CADMOVER always displays all files in any binary format that it can translate. In addition, it will display all files of type Text.

Importing Text Files from Other Computers

Different computers may have slight variations in the way Text files are formatted. Text files contain lines of text which are separated by line separator characters. The particular line separator characters used are generally dependent on the originating system. CADMOVER has been specifically designed to ignore variations in line separator characters, so that no special processing is required to prepare a Text file for use by CADMOVER. The only requirement is that the file's type be Text.

Automatic Format Recognition

As mentioned above, Binary files are usually tightly associated with one particular program. Any binary file which is displayed in the list CADMOVER shows you can be opened and translated. Since Text files are not associated with any particular program or format, CADMOVER actually has to open the file and perform some analysis of the file's contents in order to figure out what format the file contains. If you select a file, and CADMOVER is unable to determine what format the file contains, a message to that effect will be displayed, and the file will be closed.

3DGF Files

CADMOVER can both read and write files in 3DGF format. 3DGF is a 3D format which describes objects composed of surfaces.

Colors

3DGF files may or may not carry color information. When a 3DGF file does have a color value for each face, CADMOVER will use the given value; otherwise it will assign the color black to each face. When CADMOVER writes 3DGF files, it will assign a color to each face if there is any color information in the input file being translated to 3DGF.

Non-Surface Objects

Things which are not surfaces, such as lines, open polygons, and text, are not translated to 3DGF.

CADMOVER Reads It All

CADMOVER is always able to represent surfaces. Since all the geometric entities in 3DGF files are surfaces, CADMOVER can read and translate all of the geometry in any 3DGF file.

Measurement Units

3DGF files do not actually have any measurement units. However, CADMOVER treats them as though the units are centimeters. When you are writing a 3DGF file from CADMOVER, it is generally a good idea to scale your model so that it will be on the order of 5-10 cm in size before you save it.

Architrion II Files

CADMOVER can read files produced by Gimeor's "Architrion II" application.

3D Only

Architrion II files contain exclusively 3D information, which consists of solid blocks and holes through the blocks. CADMOVER translates all blocks as solids with their surface normals pointing outwards from the body of the solid. Holes are also translated as solids, but with their surface normals pointing inward toward the center of the block.

Colors

The colors used in Architrion II files are derived from the system color table on whatever Macintosh was used to produce the original file. CADMOVER will apply color to the models it reads in from Architrion II files. Whether it applies the "right" colors is dependent on the similarity between the active color table on the generating system and the active color table on the receiving system.

Each Block is an Object

CADMOVER treats each block within an Architrion II file as a separate object. This will allow you to manipulate individual blocks after you've translated your Architrion II file to another 3D format and you are working in whatever target application you have chosen.

Input Only

CADMOVER only reads Architrion II files; it does not write them.

CGM Files

An Exchange Standard

The CGM (Computer Graphics Metafile) format is not associated with any particular program. Rather, it is an exchange standard supported by the American National Standards Institute.

2D

The CGM format is designed for the exchange of 2D drawing information. It does not support 3D modeling.

Text or Binary?

The CGM standard defines both a binary format and a clear text equivalent. CADMOVER can be configured to read and write the binary format. It will not read the clear text version of CGM.

Color Table Support

CGM files may contain color table information. If you have a CGM file which does contain color table information, CADMOVER will faithfully reproduce it.

Machine Independent

CGM has been carefully specified in a way which makes it independent of any particular hardware platform. Programs which have support for CGM can be found on a wide variety of different computers, from small personal systems through large mainframe computers.

Check Your Measurements

CGM files optionally contain measurement information. When a CGM file does contain measurement information, it's no problem for CADMOVER to tell exactly how large the drawing should be. However, it is perfectly legitimate for a CGM file to omit any reference to size; in other words, it could be correctly displayed at any size, so long as the ratio between height and width is maintained.

Because most file formats are sensitive to real world measurements, it always pays to take a look at CADMOVER's "Scale" Dialog after reading a CGM file, but before translating it to some other format. If you don't set the scale yourself, the results of the translation could be unpredictable.

Different Dialects

Ideally, CGM files should be completely interchangeable among any two programs or machines which claim to support CGM. In practice, this is not the case. Different vendors have taken different approaches to CGM, and some of the language in the written standard is subject to some interpretation. CADMOVER already modifies some of its heuristics based on the vendor identification in the CGM file. If you have a CGM file which you feel is not being read correctly, please call Kandu Software Corporation to let us know. You may have a CGM file in an unknown dialect. If this occurs, the technical staff at Kandu Software would like to become aware of a new dialect, in order to add a little more knowledge to the next version of CADMOVER.

ClarisCAD Files

CADMOVER can read and translate the data in ClarisCAD files.

Text with Multiple Attributes

ClarisCAD gives you the ability to use different fonts, font sizes, and attributes within a single text object. CADMOVER does not support multiple attributes within a single text string; instead, it will use the font and font size from the first character in the string. If you need to have exact font and font size information translated, then Save your file in PICT format. You'll be pleased with the results.

Layer Support

ClarisCAD supports named layers. CADMOVER gives full support for an unlimited number of layers. The structure of the layers will be maintained when translating to any format which has layers, and even the layer names will be preserved when translating to formats which support named layers. When translating to IGES, the layer structure is maintained, but layers will be numbered instead of being named.

Higher Precision

Files stored in ClarisCAD format are capable of retaining a higher degree of precision than files stored in PICT format. CADMOVER fully supports the precision used by ClarisCAD.

DDES2/IT8.6 Files

General Information

The DDES2 (IT8.6) format is generally used to communicate the design of die-cut objects. CADMOVER both reads and writes DDES2 files.

Design vs. Layout

DDES2 files contain a Design section and a Layout section. The Design section describes the actual geometry of the object contained in the file. The Layout section describes how multiple copies of the design may be laid out on a larger sheet of material, e.g., a sheet of vinyl, cardboard, or plastic.

When CADMOVER reads a DDES2 file, it operates only on the Design section. This means that you don't have to do any additional work to untangle the design from the layout, such as deleting extra copies of the design, rotating to zero, etc. So no matter how many copies of the design have been specified in the Layout section of the DDES2 file, CADMOVER always ensures that you have one unambiguous copy of the design to work with.

When CADMOVER writes a DDES2 file, it includes a Layout section which specifies one copy of the design with no rotation, scaling, nor mirroring.

Bridges

DDES2 files often specify Bridges, i.e., gaps in a cut line. CADMOVER correctly interprets bridge information in straight lines and in arc sections. Cut lines with bridges in them will appear on the screen as dashed lines.

Dimensions Files

Application Support

This format is for data exchange with the Design Dimensions application by Visual Information, Inc.

The Dimensions Format

The Dimensions format is designed for 3D surface modeling. It is very useful for representing smoothly curved surfaces. CADMOVER can be configured to both read and write the Dimensions format.

Spline Surfaces

Dimensions files typically contain spline surfaces. Please read the section in this manual about 3D Meshes for a more thorough treatment of this subject.

Color

Dimensions files carry color information. CADMOVER will correctly translate the colors in Dimensions files into any other format.

Writing Dimensions Files

Since most 3D formats are designed to represent polygons, CADMOVER writes polygons into the Dimensions format when you are saving a file in Dimensions format. The Dimensions application has facilities to convert polygons into spline surfaces automatically, so the transition from polygons to spline surfaces is an easy one.

Dreams Files

A 2D Drawing Format

Dreams, from idd (Innovative Data Design), is a 2D drawing package. CADMOVER gives full support to Dreams files.

Layer Support

CADMOVER is sensitive to Dreams' layer structure, and will faithfully copy Dreams layers into any other drawing format for which layers are supported. Conversely, if CADMOVER has read a drawing file in any format which has a layer structure, the layers from the original drawing will be preserved when you Save the drawing in Dreams format.

Custom Colors

CADMOVER gives full support to the custom colors available in Dreams. You may create any colors you like; CADMOVER will get the same color into your

output file. And when you are translating files into Dreams format, CADMOVER will create the correct custom color table for you.

Line Styles

CADMOVER also correctly reads and translates dashed lines from Dreams files. And when translating into Dreams, CADMOVER will preserve the first sixteen dashed line styles it encounters.

Home View

In order to maintain proper scaling, Dreams drawings should be saved when the home view is active, i.e., not in a zoomed or custom view. When CADMOVER creates a Dreams file, the file will be created in the home view.

Spline Curves

Dreams can work directly with Bézier splines. CADMOVER represents these splines correctly, and can maintain them as true Béziers when transferring data among formats which support true splines, such as IGES and Adobe Illustrator formats.

DXF Files

The DXF standard is promulgated by AutoDesk, Inc., the makers of AutoCAD®. "DXF" is an acronym - it stands for "Drawing eXchange Format". Although it was originally designed to get data in and out of AutoCAD, the DXF format is now used by a wide variety of programs. CADMOVER adheres to DXF standards supported by AutoCAD; in short, if AutoCAD will read a DXF file, then CADMOVER will read it, and conversely, if AutoCAD will not read a given DXF file, then CADMOVER may not read it, either. For that matter, if AutoCAD won't read the file in response to a "DXFIN" command, then that file is not considered by CADMOVER to be a "real" DXF file.

CADMOVER is generally more tolerant than AutoCAD is when reading DXF files. A benefit of this fact is that DXF files which are rejected by AutoCAD itself can sometimes be read in to CADMOVER, and then saved as DXF again. In so doing, CADMOVER will "clean up" inconsistencies in DXF files. The resulting "new" DXF file is much more likely to be accepted by AutoCAD.

DXF Version Must Be 2.54 or Higher

Note that the above statements apply to AutoCAD releases 2.54 and up. There is an older flavor of DXF which was used by version 1.0 of AutoCAD; current releases of AutoCAD won't read these obsolete files, and neither will CADMOVER.

Both 2D and 3D

DXF files may represent 2D data or 3D data. CADMOVER deals with both possibilities. When you are using CADMOVER to generate DXF files, be aware of the setting of the "DXF Version" preference on the main "Preferences..." Dialog. CADMOVER will restrict itself not to include any features in DXF files it writes which are associated with a version number higher than the version you have selected. That means that, on one hand, you can use CADMOVER to create DXF files compatible with older versions of AutoCAD (or any other software which reads DXF files), while on the other hand, you can also use CADMOVER to create DXF files which exploit the latest features of DXF. You can even use CADMOVER to convert one of the later versions of DXF into an earlier version of DXF.

DXF Files are Usually Text Files

When you are bringing DXF files into the Macintosh environment from some other system, be aware that they are DXF Text files. No special reformatting is necessary for CADMOVER to read the file; it already contains logic to adjust itself automatically to be able to read text generated on many different systems.

In fact, DXF files may contain binary data instead of text data. CADMOVER reads and writes either variation with equal aplomb. But in either case, no reformatting is required.

Be sure to read the section in this manual about Line Terminators. The Line Terminators setting applies to DXF files which CADMOVER writes.

User Defined Line Styles

The DXF format supports customized line styles. The style information for dashed or dotted lines is copied exactly into IGES files. Dashed and dotted lines are rendered as a collection of small line segments when translating DXF into a format which does not support dashing. If you are translating into MiniCAD+, CADMOVER offers you the facility to map line styles into one of the styles available within the framework of those programs. See the section in this manual about the Lines Menu for more information on this subject.

Multiple Layers

DXF also supports layers. CADMOVER gives 100% support to the layer configuration within a DXF file. Within a DXF file, layers may be "frozen." CADMOVER explicitly ignores frozen layers in a DXF file.

When CADMOVER is generating a DXF file based on input in a different format, the layer names may have to be changed to fall within acceptable guidelines for DXF layer names. CADMOVER does this automatically. It is careful to avoid "collisions" between layer names which might otherwise occur if two layer names, neither of which is a valid DXF layer name, were to be changed to the same new name.

Binary DXF

CADMOVER reads and writes binary DXF files. To save a file in binary DXF format, select "Binary DXF" instead of "DXF" from the format menu in the "Save As..." Dialog.

ENVISAGE 3D Files

The ENVISAGE 3D program, from BYTE BY BYTE Corporation of Austin, Texas, is a 3D modeling and rendering program which provides a full set of 3D editing tools, and is capable of generating excellent photorealistic renderings.

2D or 3D — CADMOVER Does Both

CADMOVER both reads and writes the ENVISAGE 3D format. As its name implies, ENVISAGE is a full 3D program. CADMOVER can move data to ENVISAGE 3D from any other 3D format which it recognizes, including IGES and DXF. In addition, CADMOVER can translate 2D data into ENVISAGE 3D format.

ENVISAGE 3D Extrudes 2D Data

2D data moved into ENVISAGE format can be easily extruded into the third dimension by the ENVISAGE program.

Translating ENVISAGE 3D Data to Other Formats

CADMOVER offers two-way linkage to the ENVISAGE application. Models built or modified in ENVISAGE 3D or ENVISAGE 4D can be exported to the full spectrum of formats supported by CADMOVER. Full RGB color, measurement units, and surface geometry are maintained for all translations.

Tessellation

See the section in this manual about "3D Tessellation" to find out how you can detail CADMOVER's translation of data into the ENVISAGE 3D format to either include or exclude surfaces with five or more sides.

Focus Files

Even though CADMOVER can read and write Focus's text exchange format, it also has the capability of reading and writing Focus files directly.

What's the Difference?

Mostly, color. The Focus Text format does not support color, but the Focus format does. CADMOVER can read the color information from Focus files, and when files are translated into Focus format by CADMOVER, color information will be stored in the file.

Time is also a factor. Using the direct Focus format is faster, both reading and writing, for both the CADMOVER application and the Focus application.

Focus Text Files

Focus is a 3D modeling and rendering package from Abvent, of Paris, France. CADMOVER has a bidirectional exchange capability with Focus Text files.

Text vs. Binary

Focus normally stores its files in a binary format, but it will alternately use the text format. The advantage of using the text format is that it is much less subject to change than the binary format; therefore, as time goes on, if you upgrade to a new level of Focus, and the binary format has undergone changes, you can still continue to use the same version of CADMOVER.

Discrete Objects

Focus works with 3D objects. When CADMOVER creates a Focus file, each set of contiguous surfaces defines an object. It doesn't matter whether your input file defines objects -- CADMOVER assembles them for you.

Color

The Focus Text format does not support color. All objects in a Focus Text file will be treated as though they are black.

Gerber Plotter Files

The Gerber Plotter format is a 2D file format used for the storage and transmission of photoplotter data. Several attributes make this format unique, including the use of apertures of various shapes and sizes, and the total lack of any color information.

CADMOVER both reads and writes files in Gerber format. The list of aperture descriptions is not embedded in the file itself, so another avenue is needed to communicate aperture information.

Reading Gerber Files

When reading Gerber files, CADMOVER refers to the aperture descriptions provided by the Preferences you've set up. See the section in this manual entitled "Gerber Plotter..." in the "Preferences" chapter for a detailed description of the aperture setup.

Writing Gerber Files

When writing Gerber files, CADMOVER produces a secondary file, in Text format, describing the aperture setup it has generated for the output file. If you are keeping a Log file, CADMOVER will place a copy of the generated aperture list into the log file for reference. The aperture lists generated by CADMOVER are suitable input for the AperText application program, which is also described in this manual.

HPGL Files

CADMOVER can translate HPGL plotter files into any other supported format. In order for CADMOVER to recognize an HPGL file, the file must either be of type 'Text', or the file's name must end in '.HPGL'.

Stroked Text vs. Label Text

Depending on the program which originated the plot file, text may either be stroked or label text. Stroked text is actually composed of a large number of small pen movements. The advantage of stroked text is that it will always look the same on any computer, regardless of what fonts are available. The disadvantage of stroked text is that it cannot be edited as text. Label text actually retains its identity as text, and therefore can be edited as text after you translate the file to a different format and open your drawing in the program of your choice. The disadvantage of label text is that its appearance may change somewhat, depending on what fonts are installed on your system. When translating label text in HPGL files, CADMOVER will usually choose Courier font, if it is available, to represent the text. If the Courier font is not available, CADMOVER will use the Monaco font,

which is always available. If neither of these fonts are acceptable, you can use CADMOVER's Font Mapping control to override the font assignments.

More Information

See the "HPGL Details..." section in Preferences chapter for more information about manipulating HPGL files.

IGES Files

The IGES (Initial Graphics Exchange Specification) format is documented by the National Institute of Standards and Technology. It is designed to be an exchange format, and therefore has been designed to support a wide variety of graphic concepts. IGES files are produced by many CAD systems, CAM systems, and other software packages which deal with geometric concepts. IGES is an appropriate standard for exchanging data among machines of different types, for instance, mainframe to minicomputer or vice versa.

IGES files may contain 2D data or 3D data, or, as is often the case, they contain a 3D description of a set of objects and instructions for laying out projections of those objects on a 2D plane. CADMOVER deals with all of the possibilities mentioned here.

Multiple Drawings Per File

An IGES file may specify more than one drawing per file. Sometimes these drawings are meant to be translated together, and sometimes they should be translated separately. When CADMOVER opens an IGES file which contains more than one drawing, it will show you a dialog panel with a check box for each drawing in the file. Initially, all the boxes will be checked with an 'X', meaning that all the drawings should be translated together. If you want separate translations, click on the appropriate check boxes to remove the 'X', thereby telling CADMOVER not to process the indicated drawing. Each check box will be accompanied by either a number or a name. CADMOVER shows you the drawing name by each check box if the IGES file includes that information, and shows you a sequence number if the drawing name is not available.

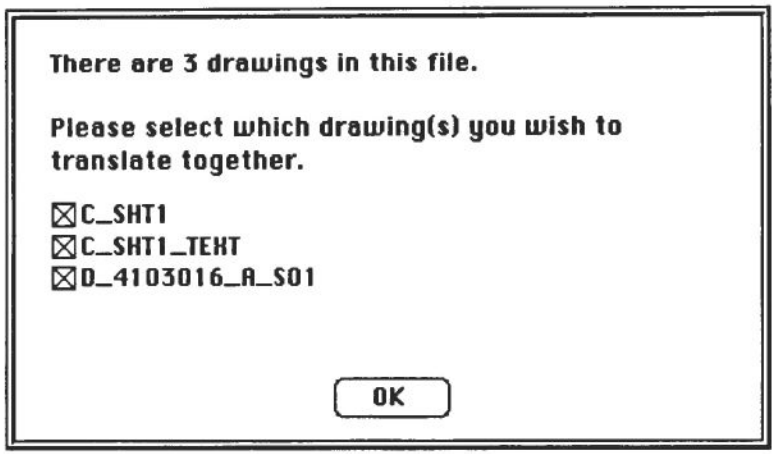


Fig. 5-1: Multiple Drawings In A File Dialog.

IGES Files Contain Plain Text

When you are bringing IGES files into the Macintosh environment, be aware that they are Text files. No special reformatting is necessary for CADMOVER to read the file; it already contains logic to adjust itself automatically to be able to read text generated on many different systems.

IGES Has Dialects

The IGES standard has been subjected to various interpretations by different vendors of IGES translators. This is to say that although IGES is one language, there are many 'dialects'. CADMOVER is very tolerant of variations among IGES implementations, often employing artificial intelligence techniques to smooth over inconsistencies or ambiguities in such a way that the user isn't even aware that any problems exist. Furthermore, most IGES files contain information which identifies the program which created the file. CADMOVER uses that information to resolve nuances of translation.

Identifying the Origins of an IGES File

When CADMOVER writes an IGES file, you will be prompted for information to put in the header of the file. This information is intended to identify the object being represented, the person responsible for making the file, and the company or organization with which the file is associated.

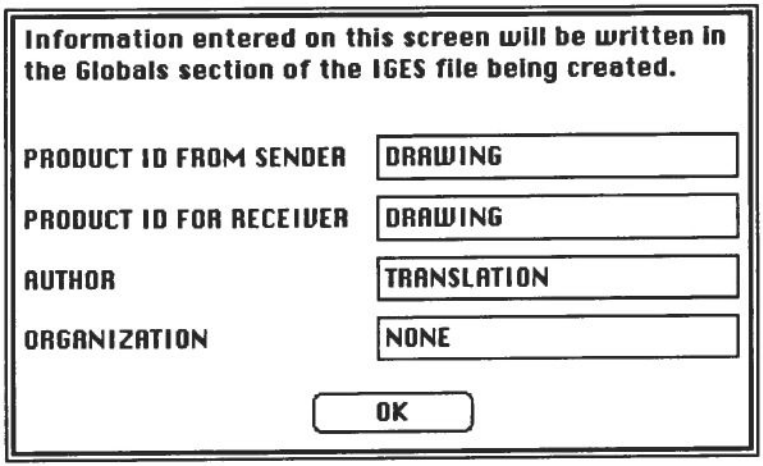


Fig. 5-2: The Drawing Origin Dialog

Be sure to read the section about Line Terminators on page 21 of this manual. CADMOVER can generate files which are preformatted for use on almost any type of computer you choose, so it is a good idea to be aware of how the Line Terminator preference is set.

IGES Entities

The IGES standard defines a wide variety of entity types. Not all of the entity types are recognized by CADMOVER; however, the vast majority of graphic data can be expressed by combinations of the entity types which are supported by CADMOVER. A list of these follows:

100 Circular Arc Entity. This entity is fully supported. CADMOVER recognizes the difference between a full circle and arcs of other than 360°, and treats each case accordingly.

102 Composite Curve Entity. The Composite Curve is fully supported.

104 Conic Arc Entity. CADMOVER implements all forms of this entity, including the Ellipse, Parabola, and Hyperbola.

106 Copious Data Entity. Most forms of the Copious Data Entity are supported. CADMOVER optimizes the translation of multiple connected line segments for target formats which provide a multi-line data type.

108 Plane Entity. CADMOVER recognizes the Plane Entity, and can use it to orient an associated curve or polygon on the surface of the plane.

110 Line Entity. CADMOVER fully implements the Line Entity. Standard fonts (Solid, Dashed, Phantom, and Centerline) are supported for receiving systems which can display these types. References to entities of type 304 (Line Font Definition) are currently ignored.

112 Parametric Spline Curve Entity. The linear, quadratic, and cubic forms are supported.

114 Parametric Spline Surface Entity. The linear, quadratic, and cubic forms are supported. For most output formats, CADMOVER translates the spline surface into a polygon mesh. User control of meshing density is available.

116 Point Entity. CADMOVER fully supports the Point Entity.

118 Ruled Surface Entity. CADMOVER gives full support to the Ruled Surface Entity.

120 Surface of Revolution Entity. This entity defines a swept surface indirectly, by referring to an axis and a generatrix curve. CADMOVER supports it fully.

122 Tabulated Cylinder Entity. This is a compound entity, and it is supported by CADMOVER. It is strictly a 3D entity, but CADMOVER will project it on to a 2D plane for export to a 2D format.

124 Transformation Matrix. This entity is central to IGES, and it is supported fully. CADMOVER uses full 3D transforms for all internal calculations.

126 B-Spline Curve Entity. This entity is fully supported. Curves with any number of control points may be specified.

128 B-Spline Surface Entity. CADMOVER reads all forms of the B-Spline Surface, including quadratic, cubic, and higher orders.

134 Node Entity. This entity, used in Finite Element Analysis, is translated by CADMOVER as geometry.

136 Finite Element Entity. This supported entity is a grouping of Node entities (see above).

142 Curve on a Parametric Surface Entity. CADMOVER reads all forms of the Type 142 Entity. This entity is generally used in conjunction with the Type 144 (see below)

144 Trimmed Surface Entity. CADMOVER reads all forms of the Trimmed Surface Entity. It will translate the entire surface to any chosen output format.

202 Angular Dimension Entity. CADMOVER supports the Angular Dimension Entity.

206 Diameter Dimension Entity. Fully supported.

210 General Label Entity. CADMOVER supports General Labels. See the notes for type 212 (General Note Entity) below.

212 General Note Entity. IGES defines a wide variety of character sizes and fonts. CADMOVER carefully selects a type font and font size which will give the closest visual representation on the receiving system. In general, only characters within the range of the standard ASCII character set will translate exactly.

214 Leader (Arrow) Entity. CADMOVER supports the Leader Entity. At the present time, all arrowheads are treated as the form 3 type (filled triangle).

216 Linear Dimension Entity. Fully supported.

222 Radius Dimension Entity. Fully supported.

228 General Symbol Entity. This composite entity may contain any number of text items, leaders (arrows), and geometric items. Its intended purpose is annotation. CADMOVER offers full support of the General Symbol.

230 Sectioned Area Entity. This entity is used to specify both solid and patterned fills for closed areas. It may be associated with a Color Definition entity (type 314) to specify a closed area filled with a solid color.

304 User Defined Line Style. CADMOVER supports the standard IGES indexed line fonts as well as all variations of the user defined line style.

308 Subfigure Definition Entity. This is a powerful feature in IGES, and CADMOVER gives it full support. In general, the Subfigure Definition entity

corresponds to a symbol library entry. It is a composite entity, and may contain references to any geometric entities.

314 User Defined Color. CADMOVER supports all manifestations of RGB colors.

320 Network Subfigure Definition Entity. The Network Subfigure Definition Entity is closely related to the type 308 (Subfigure Definition entity). CADMOVER treats both the type 320 and type 308 as symbol library entries.

402 Associativity Instance Entity. CADMOVER supports the Views Visible form (form number 3) of the Associativity Instance Entity.

404 Drawing Entity. CADMOVER offers good support for the Drawing Entity. IGES allows multiple drawings to be specified in a single source file; CADMOVER recognizes these as separate entities, and gives the user the option to select individual drawings for translation. Drawing scale and placement on the page are controlled by the View entity (type 410), which is also supported.

406 Property Entity. CADMOVER recognizes and supports forms 15 and 16 (Name and Drawing Size) of the Property Entity.

408 Subfigure Instance Entity. This entity is closely related to Entity 308 (Subfigure Definition Entity), and is fully supported by CADMOVER.

410 View Entity. CADMOVER uses the Orthographic Parallel Projection to transform model space into view space. The View Entity may be used to specify a scale factor for the transformation from view space to drawing space, and a transformation matrix controls the placement of the view within the drawing.

412 Rectangular Array Subfigure Instance Entity. This entity allows a subfigure to be defined once, and then place multiple copies in a rectangular grid arrangement. It is related to types 308, 408, 320 and 420. CADMOVER gives full support to this entity.

414 Circular Array Subfigure Instance Entity. Very much like Type 412, except that the copies will be arranged in a circular fashion. Fully supported.

420 Network Subfigure Instance Entity. CADMOVER treats the Network Subfigure Instance entity as though it were a type 408 (see above), which implies full support for this entity.

MacArchitrion 3D Text Files

MacArchitrion, from Gimeor, is the predecessor to Architrion II. It can export 3D models in a Text format, which CADMOVER can read.

Just Like Architrion II

The main difference between Architrion II files and MacArchitrion 3D Text files is quite simply that Architrion II files are native binary files, and MacArchitrion 3D Text files must be exported from the MacArchitrion application before they can be translated. All of the statements about color tables and solid blocks in Architrion II files apply to MacArchitrion 3D Text files as well.

French and English

The MacArchitrion has been marketed in both French and English versions, and this dichotomy is reflected in the Text files exported from MacArchitrion 3D: the text files themselves are written in either English or French. CADMOVER reads both.

MacDraft Files

A 2D Drawing Format

MacDraft, from idd, is a 2D drawing package. CADMOVER gives full support to MacDraft files.

Only Recent Versions

CADMOVER directly supports the reading and writing of MacDraft files at a version level of no less than 2.0. In other words, CADMOVER writes MacDraft 2.0 files and reads MacDraft 2.0 files. If you are using a version of MacDraft whose version number is less than 2.0, CADMOVER can still help you. All you have to do is use PICT format instead of MacDraft format. But the 2.0 (and higher) levels of MacDraft support many features which cannot be readily expressed in PICT format, such as multiple drawing layers.

Layer Support

CADMOVER is sensitive to MacDraft's layer structure, and will faithfully copy MacDraft layers into any other drawing format for which layers are supported. Conversely, if CADMOVER has read a drawing file in any format which has a layer structure, the layers from the original drawing will be preserved when you Save the drawing in MacDraft format.

Custom Colors

CADMOVER gives full support to the custom colors available in MacDraft. You may create any colors you like; CADMOVER will get the same color into your

output file. And when you are translating files into MacDraft format, CADMOVER will create the correct custom color table for you.

Line Styles

CADMOVER also correctly reads and translates dashed lines from MacDraft files. And when translating into MacDraft, CADMOVER will preserve the first four dashed line styles it encounters.

MacDraw II Files

CADMOVER can read and translate the data in MacDraw II files.

Text with Multiple Attributes

MacDraw II gives you the ability to use different fonts, font sizes, and attributes within a single text object. CADMOVER does not support multiple attributes within a single text string; instead, it will use the font and font size from the first character in the string. If you need to have exact font and font size information translated, then Save your file in PICT format.

Layer Support

MacDraw II supports named layers. CADMOVER gives full support for an unlimited number of layers. The structure of the layers will be maintained when translating to any format which has layers, and even the layer names will be preserved when translating to formats which support named layers. When translating to IGES, the layer structure is maintained, but layers will be numbered instead of being named.

Higher Precision

Files stored in MacDraw II format are capable of retaining a higher degree of precision than files stored in PICT format. CADMOVER fully supports the precision used by MacDraw II.

MiniCad+ Text Files

The MiniCad+ application, from Graphsoft, Inc., is the successor to MiniCad, and it offers many more features and tools. CADMOVER can read and write Text files in MiniCad+'s own internal scripting language, which is called MiniPascal.

Exporting From Minicad+

In order to write a MiniPascal file from the MiniCad+ application, select "Export..." from the File Menu in MiniCad+, and then select "Text Format" from the submenu that pops up. If you are working in 3D in MiniCad+, and you have

sweeps or extrusions in your document, then before you export your file from MiniCad+, follow these steps:

1. Be sure that you have saved your document before exporting it.
2. Select all sweeps and extrusions, and use the 3D tool "Convert to Mesh" on those objects.
3. Export the file as above.
4. Close your document without saving it (unless you want to keep the effect of the "Convert to Mesh" operation).

The file you have exported can be read by CADMOVER.

Importing Into Minicad+

After CADMOVER has exported a file in MiniCad+ Text format, launch the MiniCad+ application, and select "Text Format" from the submenu appended to the "Import..." item on the File Menu in MiniCad+. Then select the file which you exported from CADMOVER.

Versions

The MiniCad+ Text files written by CADMOVER can be imported into any version of MiniCad+. CADMOVER can read MiniCad+ Text files written by all versions of MiniCad+ up to version 3.1.

Attributes

CADMOVER maintains all geometric attributes when reading or writing MiniCad+ Text files. Color, line weight, arrowheads, and layer attributes are all there. And symbols stay symbols.

PICT Files

Different Versions Of Pict

At the time of this writing, there are two "flavors" of PICT files: the original PICT 1, and the newer PICT 2 format. Most of the time, the distinction between PICT 1 and PICT 2 is not important.

The basic difference between the two formats is that PICT 2 supports a full range of RGB color, while PICT 1 does not. PICT 1 does support eight basic colors: black, white, red, green, blue, magenta, cyan, and yellow.

CADMOVER reads either format with equal aplomb. If you are running on a machine which does not support Color QuickDraw, and you are reading a PICT 2 file, the RGB colors within the file will be mapped to the closest match out of the eight colors listed above.

Bitmaps

The PICT format does support bitmaps. CADMOVER does not. Therefore, when you are using CADMOVER to read a PICT file which contains bitmaps, the bitmaps will be ignored. If you need to translate bitmaps to other formats, you'll need to use a program which can turn a bitmap into vectorized PICT before running CADMOVER. One such program is Super Paint from Silicon Beach. Super Paint has an AutoTrace feature which lets you take outlines of bitmaps and save them as lines and curves into a new PICT file. CADMOVER will be able to process the new PICT file in a meaningful way.

Pict Is "Built In"

As mentioned above, PICT is the "native" format of the Macintosh. The Macintosh operating system contains built-in routines for dealing with PICT objects. This is why you can successfully read a PICT 2 file on a monochrome Macintosh. But other computers don't have this kind of built-in support for PICT. Some programs which run on other computers claim that they can import Macintosh PICT files. In almost all cases, this means PICT Version 1 only. CADMOVER has a feature designed to deal with this situation: the PICT 1/PICT 2 switch. Even on a color Macintosh, you can generate PICT Version 1 files. For more information about PICT versions, see the sections in this manual about the Options Menu and the Preferences Dialog.

Drawing Size Is Limited

Because of the way data are stored in PICT files, there is a maximum size limit imposed on drawings. If you have a file in some other format, which represents a drawing larger than 200 inches in either direction, you will probably need to scale it down using the Scale Dialog from CADMOVER's Options Menu. In fact, if your drawing exceeds this limitation, and you do not use the scaling dialog yourself, CADMOVER will automatically scale its output PICT file down so that the maximum size represented in the file falls within these limits. A message to that effect will be placed in the Log file whenever CADMOVER does invoke automatic scaling. Either one or both of the options "Show Log File" or "Keep Log File" must be turned on in order for you to see this message. If logging is turned off, you may be unaware that CADMOVER has invoked automatic scaling. But in any case, even if your input file depicts objects many kilometers across, the output PICT file will be able to contain all the geometry which was expressed in the input file.

Drawing Accuracy

PICT files are expressed in PIXEL units. Each pixel is 1/72" (inch). For this reason, no matter what unit system is used in your input file, CADMOVER always translates measurement units to inches, with an accuracy of 1/72, when you are writing a PICT file.

Getting Rid Of Jagged Edges In PICT

Sometimes drawings which were made at a larger scale, or with a program which supports very high drawing accuracy, look a little rough when they are translated to PICT format. There is a technique which may help you avoid that rough, jaggy look. Suppose you have a drawing in some other format (not PICT), which is 18" high and 24" wide. You want to bring this drawing into a drawing program which is able to support fairly high accuracy, but it only reads its own format and PICT files. Use the "Scale..." Dialog in CADMOVER to increase the drawing size to something larger — for example, a magnification of 4X would bring your 18" x 24" drawing up to 72" x 96". Now Save the file in PICT format, import it into your high-accuracy drawing program, and use the facilities provided by that program to reduce the drawing size to 25% — to undo the effect of magnifying 4X in CADMOVER. The resulting accuracy will be four times as great as that normally afforded by the PICT format. Instead of being represented to the nearest 1/72 inch, your drawing will now be represented to the nearest 1/288 inch — a vast improvement!

Postscript Files

CADMOVER can read files produced by Adobe Illustrator 88, and translate them into virtually any format you like. It works equally well with black and white files, or full color files. In addition to Adobe Illustrator 88 files, CADMOVER can also read PostScript files exported from Aldus Freehand 2.0.2, EPS Exchange, Super 3D, Schema, MicroStation, AutoCAD®, and a few other sources.

PostScript Dialects

CADMOVER does check the creator ID which is placed in every PostScript file by the creating application. If the creator ID is missing, or is not one recognized by CADMOVER, then CADMOVER will display a message telling you that the dialect is not recognized. The Technical Support group at Kandu Software will be happy to look at your PostScript file to determine whether a new translator for that dialect can be added to CADMOVER.

Filled Areas vs Spline Curves

Many users will be interested in translating filled areas as such. However, if you are interested only in the line work, and are translating to IGES format, there is an option within the Preferences Dialog which will tell CADMOVER to ignore the area fills. When the area fills are ignored, the smooth curves represented in the Illustrator file will be translated as true spline curves. Otherwise, the boundaries of areas which are both filled and curved will be translated as polylines.

Screen Appearance

If you are translating filled areas, and you are running on a machine which does not have a color monitor (or if your monitor is set to monochrome mode), then CADMOVER will use a dithered pattern to represent different colors or gray levels on the screen. If you then translate the file to PICT format, you may be surprised to see the screen turn solid black. This happens because, even though dithered patterns are used on the screen, CADMOVER places color information into any PICT files it writes. After a file has been saved in PICT format, CADMOVER displays the actual PICT data from that file on the screen. In monochrome mode, most colors other than white appear as solid black.

Writing Adobe Illustrator Files

CADMOVER can also translate a wide variety of file formats into a form compatible with Adobe Illustrator. To translate files into an Illustrator compatible format, first be sure that the "PostScript" popup menu in the Preferences Dialog is set to "Illustrator". Then, in the "Save As" window, choose EPSF.

Be Aware Of Your Drawing Size

Many CAD programs work in real world units, so that, for instance, a floor plan of a building could actually be several hundred feet in size. CADMOVER always automatically sizes drawings to fit on the screen when it displays them. When you save these files in PostScript format, it is important to scale them to fit on paper. This usually means the size of a single sheet of paper; some users may have drawings as large as poster size. Use the Scale Dialog from the Options Menu to be sure that your drawing is scaled to the right size before saving it as a PostScript file.

The Illustrator 88 program from Adobe has a maximum window size of eighteen inches square. However, it will work with drawings which are larger than the window size. We recommend that you scale your drawings down to no more than eighteen inches on a side before translating them to Illustrator format, so that you will not have any trouble locating the picture when you open it in the Illustrator program. (A large drawing may lie entirely outside of the visible window). So long as you record the scale factor you use in CADMOVER to reduce the drawing size, you can use Illustrator's scaling tool to restore the original size. Remember that the scale factor in CADMOVER is a reduction factor, and the scale factor in Adobe Illustrator is an enlargement factor. So, for example, if you tell CADMOVER to reduce your drawing by 4-to-1, then after you open the drawing in Adobe Illustrator, you can enlarge it by 400% to restore the original drawing size.

Printer PostScript

When the PostScript control in the Preferences Dialog is set to "Printer", CADMOVER will save PostScript files in a format which can be placed in almost any page layout program, and ultimately sent to a PostScript printer device, such as a LaserWriter or a Linotronics printer.

Screen Graphic Included Automatically

A "picture preview", suitable for use on any Macintosh, is automatically included in saved PostScript files. The preview can be used by other programs (such as desktop publishing applications or word processors) to position PostScript graphics within a page.

Presenter Professional Files

Application Support

This format is for data exchange with the Presenter Professional application by Visual Information, Inc.

The Presenter Professional Format

The Presenter Professional format is designed for 3D surface modeling. It is very useful for representing smoothly curved surfaces. CADMOVER can be configured to both read and write the Presenter Professional format.

Spline Surfaces

Presenter Professional files typically contain spline surfaces. Please read about 3D meshes in the Spline Mesh section, page 86, for a more thorough treatment of this subject.

Color

Presenter Professional files carry color information. CADMOVER will correctly translate the colors in Presenter Professional files into any other format.

Writing Presenter Professional Files

Since most 3D formats are designed to represent polygons, CADMOVER writes polygons into the Presenter Professional format when you are saving a file in Presenter Professional format. The Presenter Professional application has facilities to convert polygons into spline surfaces automatically, so the transition from polygons to spline surfaces is an easy one.

Sculpt 3D/4D Files

The SCULPT 3D program, from BYTE BY BYTE Corporation of Austin, Texas, is a 3D modeling and rendering program which provides a full set of 3D editing tools, and is capable of generating excellent photorealistic renderings. SCULPT 4D is essentially the same program, with animation controls added, so that entire movies can be produced.

2D or 3D - CADMOVER Does Both

CADMOVER both reads and writes the SCULPT 3D format. As its name implies, SCULPT is a full 3D program. CADMOVER can move data to SCULPT 3D from any other 3D format which it recognizes, including IGES and DXF. In addition, CADMOVER can translate 2D data into SCULPT 3D format.

SCULPT Extrudes 2D Data

2D data moved into SCULPT format can be easily extruded into the third dimension by the SCULPT program.

Translating SCULPT Data to Other Formats

CADMOVER offers two-way linkage to the SCULPT application. Models built or modified in SCULPT 3D or SCULPT 4D can be exported to the full spectrum of formats supported by CADMOVER. Full RGB color, measurement units, and surface geometry are maintained for all translations.

Tessellation

See the section in this manual about "3D Tessellation" to find out how you can detail CADMOVER's translation of data into the SCULPT format to either include or exclude surfaces with five or more sides.

Stereolithography (STL)

The STL format describes 3D objects that can be built using a stereolithography machine. STL files can be formatted either as Text files or as Binary files. CADMOVER reads and writes both types. The files themselves represent geometry only; they contain no information about surface attributes such as color or texture.

Reading STL Files

CADMOVER makes it very easy to read STL files. In fact, there are no options at all that affect the reading of STL files. CADMOVER will automatically recognize whether any given STL file is in Text or Binary format, and adjust itself accordingly. STL files are suitable for translation to any 3D format. It is generally expected that all of the facets in any STL file face outward from the object of which they are a part; therefore it is not necessary to turn on the "Two Sided Surfaces" option when translating STL to other 3D formats. In fact, you should be sure that the "Two Sided Surfaces" option is OFF before translating STL into other formats.

Of course, you may also translate STL files into any 2D format supported by CADMOVER.

Valid vs. Buildable

An STL file may contain all valid data, yet still not produce a buildable model. Producing buildable models is the ultimate goal when writing STL files. A "valid" file is one that is correctly formatted and is internally consistent. But to be a "buildable" model, more criteria must be met. In order to be buildable, a model must be comprised of one or more nonintersecting closed surfaces. Furthermore, within each surface, each face must be a triangle, facing outward, and each face

must be unique. The model must lie entirely within non-negative coordinate space. Lastly, the model must be composed in such a way that no uninterrupted edge of one face passes through a corner point between two or more other faces. Several of these rules may be summarized by saying that all edges in the model must be shared by exactly two faces, no more, no less.

CADMOVER automatically takes care of many of these conditions. First of all, tessellation is automatic. That means that if you have read in a 3D model that contains surfaces with four or more sides, CADMOVER will automatically replace those surfaces with an equivalent set of triangles. Duplicate surfaces, if there are any, are eliminated automatically. Then, a validity check is performed to be sure that all edges are shared by exactly two triangles. Surfaces which pass this test are deemed to be "closed" surfaces. Finally, once a surface has been determined to be a closed surface, CADMOVER then performs a facet alignment operation on the surface, to be certain that all facets are facing outward.

Writing STL Files

CADMOVER can produce STL Text files and STL Binary files. Except for the distinction between text and binary files, it makes no difference to CADMOVER which variety you choose to generate.

It is strongly recommended that you turn on the "Show Log File" preference in CADMOVER before writing out any STL files. This is because CADMOVER writes a status report into the log file, describing any problems encountered during translation, and ultimately issuing a declaration of whether the file appears to represent a buildable model.

Placement

In order to be buildable in most stereolithography machines, all of the coordinates in the model must be nonnegative (i.e., zero or positive.) You can use CADMOVER's Placement option to ensure that STL files written by CADMOVER appear in the proper coordinate space.

Super 3D Files

SUPER 3D, from Aldus Corporation, can import and export properly formatted text files. CADMOVER both reads and writes this format.

Bringing 2d Into 3d

SUPER 3D is strictly a 3D program. CADMOVER can convert data drawn in a 2D program into a form usable by SUPER 3D. Lines, arcs, and open polygons are converted into 3D lines, and closed figures such as circles, ellipses, rectangles, and

closed polygons are converted into 3D polygon faces. Of course, CADMOVER also imports all 3D constructs from other programs into SUPER 3D.

SUPER 3D considers polygon faces to be one-sided; i.e., each polygon has a distinct "front" and "back" side. The sidedness associated with a closed face is generally a function of the direction in which it was drawn; objects which were drawn in a clockwise fashion will face a direction opposite to those objects which were drawn in a counterclockwise fashion. This becomes important when you want to view an object as a solid within SUPER 3D.

When you are bringing files in from other systems, there's usually no way to tell how the faces are oriented. How can you be sure that all the polygons in your file have compatible directionality? The answer to that is the "Two Sided Surfaces" switch on CADMOVER's Options Menu. Be sure to read the section in this manual which describes that feature in detail. Briefly, it is usually a good idea to have this option turned ON when you are using CADMOVER to translate data into SUPER 3D format. When the switch is ON, CADMOVER will generate two polygon faces (one in each direction) for each closed object in the input file. Then, no matter how you rotate your model once it has arrived in SUPER 3D, you will always be looking at the "front" side of each face.

Wavefront (.OBJ)

About .OBJ Files

CADMOVER can read and write Wavefront Object (.OBJ) files containing polygon data. In order to be recognized as a Wavefront file, the file must be of type Text, and its name must end with ".OBJ".

These files describe 3D geometry only. They do not contain any information about texturing, color, or any other surface attributes such as transparency or luminance. When CADMOVER reads such a file, it assigns a default color (black), which is used for translations to other formats that do contain color information. Conversely, when writing .OBJ files, any color information associated with the file is omitted, since there is no way to represent it.

Different Computers

Wavefront software generally runs on UNIX systems. Therefore, if you will be writing Wavefront files, it is generally a good idea to set the "Line Terminators" preference to UNIX. When reading Wavefront files, CADMOVER automatically adjusts itself to whatever line terminators the file contains, so no special translation is necessary before CADMOVER reads the file.

Zoom Text Files

Zoom is a 3D modeling and rendering package from Abvent, of Paris, France. CADMOVER has a bi-directional exchange capability with Zoom Text files.

Text vs. Binary

Zoom normally stores its files in a binary format, but it will alternately use the text format. The advantage of using the text format is that it is much less subject to change than the binary format; therefore, as time goes on, if you upgrade to a new level of Zoom, and the binary format has undergone changes, you can still continue to use the same version of CADMOVER.

Discrete Objects

Zoom works with 3D objects. When CADMOVER creates a Zoom file, each set of contiguous surfaces defines an object. It doesn't matter whether your input file defines objects — CADMOVER assembles them for you.

Color

The Zoom Text format does not support color. All objects in a Zoom Text file will be treated as though they are black.

Zoom Files

Even though CADMOVER can read and write Zoom's text exchange format, it also has the capability of reading and writing Zoom files directly.

What's the Difference?

Mostly, color. The Zoom Text format does not support color, but the Zoom format does. CADMOVER can read the color information from Zoom files, and when files are translated into Zoom format by CADMOVER, color information will be stored in the file.

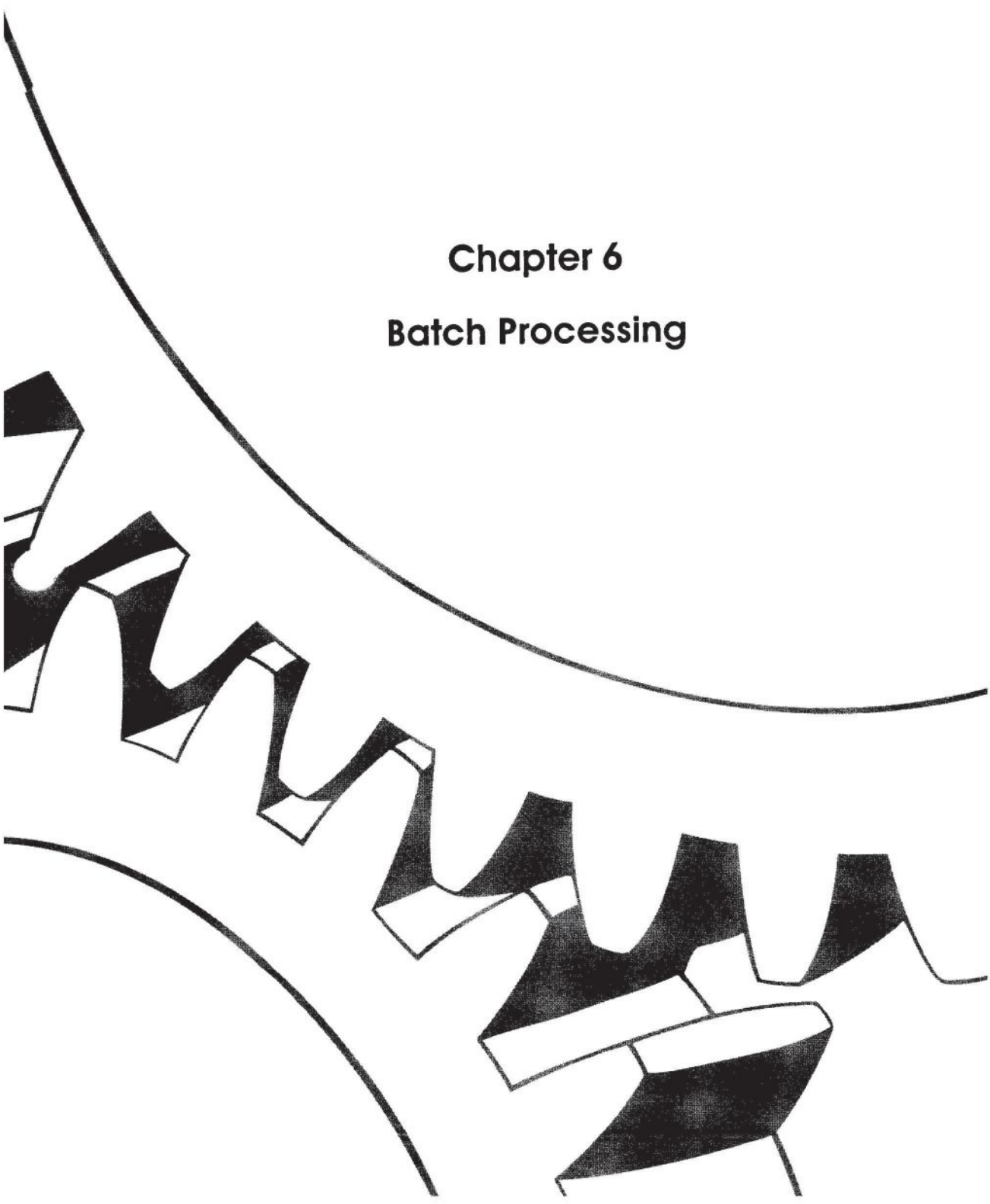
Time is also a factor. Using the direct Zoom format is faster, both reading and writing, for both the CADMOVER application and the Zoom application.

AutoCAD® DWG Files

It is unusual to document things that a program will not do, but here is an exception to that rule. CADMOVER does not process DWG files created by AutoCAD. If you want to translate a drawing made by AutoCAD, you must use either the DXFOUT command or the IGESOUT command in AutoCAD to export a file in either DXF or IGES format. CADMOVER will be able to process either type of exported file.

Chapter 6

Batch Processing



Batch Mode Processing

Batch Mode and Scripting

CADMOVER can process a group of files easily, without having to open and Save each one individually.

How To Do It

First, be sure that your Preferences are set the way you want them to be. Most users will want to run with Display Picture turned OFF (because nobody will be looking at the screen anyway), and Keep Log File turned ON (so you can get a report of how your translations progressed). Once your Preferences have been set, quit the program and return to the Finder.

Be sure that there is plenty of disk space on your target volume before starting multiple translations.

- System 7 Users
  First, select all of the files you want to translate. Then drag this group of selected files right on top of the CADMOVER icon, and release the mouse button. This technique is known as "Drag and Drop".

- System 6 (or earlier) Users
  Select all the data files you wish to translate, and select the CADMOVER application. Double-click on the application to set the translations in motion.

If you are not an experienced Macintosh user, then refer to your Macintosh owner's manual for information on how to select multiple files at one time.

CADMOVER will start up and begin reading a file. When the first file has been read in, the "Scale..." Dialog will appear. ALL THE DRAWINGS IN A BATCH WILL BE SCALED BY THE SAME SCALE FACTOR. Select the scale factor you want, and click OK. Then the "Save As..." Dialog will appear. Select the output format you want, and also select the output volume or folder where the translated files will be put. All of the translated files will be placed into the same folder or volume you select at this time. Since CADMOVER automatically generates a new name for each output file, there will be no need to assign names to each file.

After the last file has been processed, CADMOVER will go in to its normal interactive mode, waiting for instructions from you.

The Easy Way

The easiest way to translate a batch of files is to have two folders (perhaps on different disk drives, if you have more than one hard drive attached to your system). Place all the files you want to translate in one folder, and place the CADMOVER application in the same folder. Then, from the Finder's File Menu, issue the Select All command (or type command-A), followed by the Open command (or type command-O). CADMOVER will start up and begin reading files. When the "Save As..." Dialog appears, select the output file format you want, and direct the output to the second folder.

If You Change Your Mind

If you hit the "Cancel" button in the "Save As..." Dialog, the entire batch will be canceled. No translations will take place.

Multile Formats In Batch Mode

It doesn't make any difference how many different input types there are; you may select a group of files consisting of IGES, DXF, SCULPT 3D, MiniCad+ Text, etc., all in the same batch. CADMOVER will automatically determine the format of each file. Remember, though, that the same scale factor will be applied to all the files you translate.

CADMOVER's batch processing option is easy to use, but does not always provide the detailed kind of control you might like over a series of unattended translations. To run batches of files with more control on a per-file basis, CADMOVER provides a scripting feature.

Using CADMOVER's Scripting Feature

A script is actually a text file containing a series of commands. CADMOVER reads and processes the commands one at a time, in the order in which they appear. The commands, listed alphabetically, are:

```
BEEP
FORMAT
MOVE
QUIT
SCALE
SOURCE
TARGET
```

File Naming Rules

Some of the commands require file names or format names. The following rules apply to all file or format names:

1. If the name contains any blanks or special characters, it should be enclosed in quote marks, which can be single quotes ('), double quotes ("), or "smart" quotes (") and (").

2. The format names should be spelled exactly as they appear on the popup menu in the "Save As..." Dialog.

3. If the name is the name of a folder (see SOURCE and TARGET commands,) then the last character in the name should be a colon (:).

4. If it is a file name, it may be fully qualified. This means that you can specify the entire name of the file, including any folders that the file is in. For example, MYDISK:MYFOLDER:MYFILE is a fully qualified file name, assuming that the disk MYDISK contains a folder named MYFOLDER, and that that folder contains a file named MYFILE.

Optional operands are introduced by a hyphen followed by a letter.

Available Scripting Commands

BEEP	Causes the computer to emit a BEEP sound.
FORMAT	Specifies the target format that files will be translated into.
MOVE	Names the file or files to translate.
QUIT	Quits execution of CADMOVER.
SCALE	Specifies an enlargement factor.
SOURCE	Specifies the folder where input files are to be found.
TARGET	Specifies the folder where output files are to be placed.

Scripting Details

In the following list, optional operands are enclosed in square brackets. Words in italics should be replaced by an appropriate name or value. Any other text should be used verbatim.

- `BEEP`

  This command has no operands. Just place the word BEEP on a line by itself.

- `FORMAT` *`formatname`*

  Specify the format into which files are to be translated. For example,

  ```
  FORMAT IGES
  or
  FORMAT 'SUPER 3D'
  ```

  All MOVE commands which appear in the script following a FORMAT command will translate files into the format specified here (unless the MOVE command contains the "-f" option.

- `MOVE` *`filename`* `[-f` *`formatname`*`][-o` *`outputfilename`*`][-s` *`scale`*`]`

  The MOVE command is the one that really does the work. In its simplest form, it tells CADMOVER to translate the named file into the format specified on the most recent FORMAT command.

  The "-f" option lets you put a format name right on the same line with the MOVE command. For example,

  ```
  MOVE myfile.dxf -f IGES
  ```

  tells CADMOVER to translate the file "myfile.dxf" into IGES format, regardless of the most recent FORMAT command.

  The "-o" option specifies a file name for the output file. For example,

  ```
  MOVE myfile.dxf -o yourfile.abc
  ```

  tells CADMOVER to translate the file "myfile.dxf", and name the output file "yourfile.abc".

The "-s" option specifies an enlargement factor to be used by this command only. For example,

```
MOVE myfile.dxf -s 0.5
```

tells CADMOVER to translate the file "myfile.dxf", using a scale factor of 0.5, which results in a 2:1 reduction.

The optional operands can appear in combination, so that you can specify any or all of the options right on the command line.

- `QUIT`

  This command should appear on a line by itself. It causes CADMOVER to quit.

- `SCALE` *`enlargement factor`*

  This command specifies a scale to use when translating files. The default value is 1 (no scaling.) This scale factor will be in effect until the next Scale command, or the end of the script, whichever comes first.

- `SOURCE` *`foldername`*

  The SOURCE command tells CADMOVER where to look for files. If you don't use the SOURCE command, then CADMOVER will look for files in the same folder where the CADMOVER application is located.

- `TARGET` *`foldername`*

  The TARGET command tells CADMOVER where to put translated files. If you don't use the TARGET command, then CADMOVER will put output files in the same folder where the CADMOVER application is located.

Comment Lines

To enhance readability, you can put all-blank lines in the script. You can also put in comments containing any text you like by placing an octothorpe character (#) at the beginning of the line.

```
# These are sample comment lines.
# CADMOVER will ignore them.
```

Important Note: Keeping a Batch Processing Log

When CADMOVER operates from a script, it will echo the script commands into the Log file. It is strongly suggested that before running a script, you set up CADMOVER's Preferences to include either "Show Log File" or "Keep Log File" or both. Furthermore, if you do not choose to keep the log file, you probably should not use the QUIT command, because if you do, then CADMOVER will quit as soon as the last translation is complete, closing the log file window as it quits.

Running the Script

In order to run a script, first launch CADMOVER, and then select "Run Script..." from the File Menu (or type command-R) The dialog box that appears will list all files of type Text. Choose your script file, and click on the Open button. CADMOVER will immediately begin to execute the script.

ScriptMaker

The ScriptMaker™ application, which comes with CADMOVER, is designed to assist you in creating scripts. Start up ScriptMaker and you'll see a window that looks like this one.

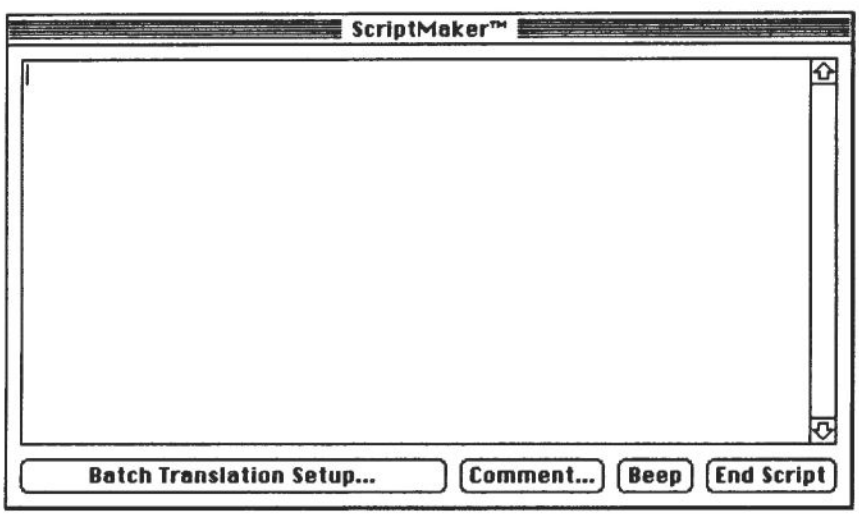


Fig. 6-1: ScriptMaker's Main Window In Normal Mode.

ScriptMaker Expert Mode

Choose "Expert Mode" from the windows menu and the window will change to look like this:

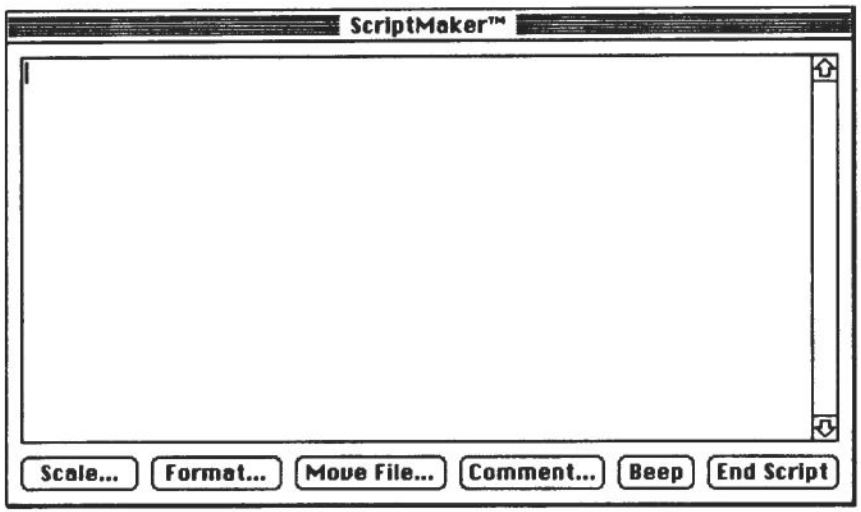


Fig. 6-2: ScriptMaker's Main Window In Expert Mode.

Each one of the buttons at the bottom of the window corresponds to one or more of the CADMOVER script commands. For each of the first buttons in Normal mode (those with three dots,) when you click on the button, you will be prompted for more information appropriate to that command. In particular, clicking on the "Move File..." button will display a standard "Open" dialog box. Choose the file you want to translate. The MOVE command, including the file you have selected, will be placed into the script. In Normal mode, clicking the large "Batch Translation Setup..." button will display a dialog where you can set three of the first CADMOVER script commands ("Move File...", "Format..." and "Scale...", at the same time.

The "End" button places a "Quit" command in the script. (It is named "End" instead of "Quit" to avoid confusion with the command that quits ScriptMaker.)

ScriptMaker does not offer a "Source" or "Target" button, because it always places fully qualified file names into the script, thereby obviating the need for those commands.

You may make your own edits right in this window, just as in any other text editor.

Opening An Existing ScriptMaker File

ScriptMaker has no "Open" command. To do further work on a previously created script, choose "Import Text..." from the File Menu.

Fig. 6-3: The ScriptMaker Document Icon

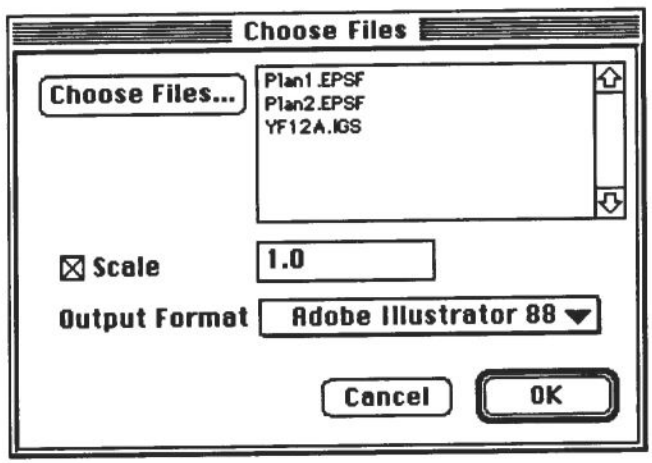


Fig. 6-4: The Choose Files Dialog.

Under System 7, CADMOVER offers special treatment to scripts that were created by ScriptMaker: You can drag the icon of a ScriptMaker document on top of the CADMOVER icon, and drop it. CADMOVER will recognize that the file is a script file instead of a graphic image, and process it accordingly. This *will not work* with scripts created by other programs. To run those, you will have to start CADMOVER, and choose "Run Script..." from CADMOVER's File Menu.

Sample Script

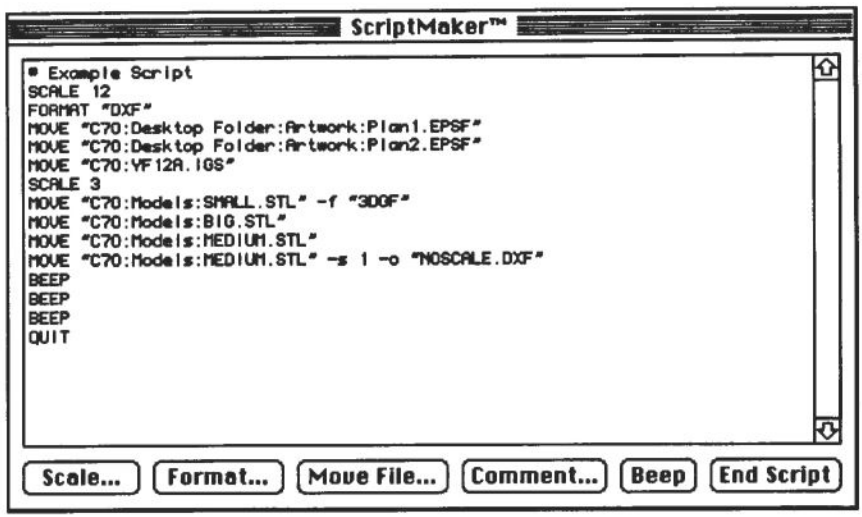


Fig. 6-5: A Sample Script

In this example, seven files are to be translated. The script starts with a comment, which can be any text at all. Then comes the SCALE command, which sets the enlargement factor to 12 for all subsequent translations until the next SCALE command. The FORMAT command specifies that all files will be translated into DXF.

The next three MOVE commands call for translation of two EPSF files from one folder, and an IGES file from a different folder.

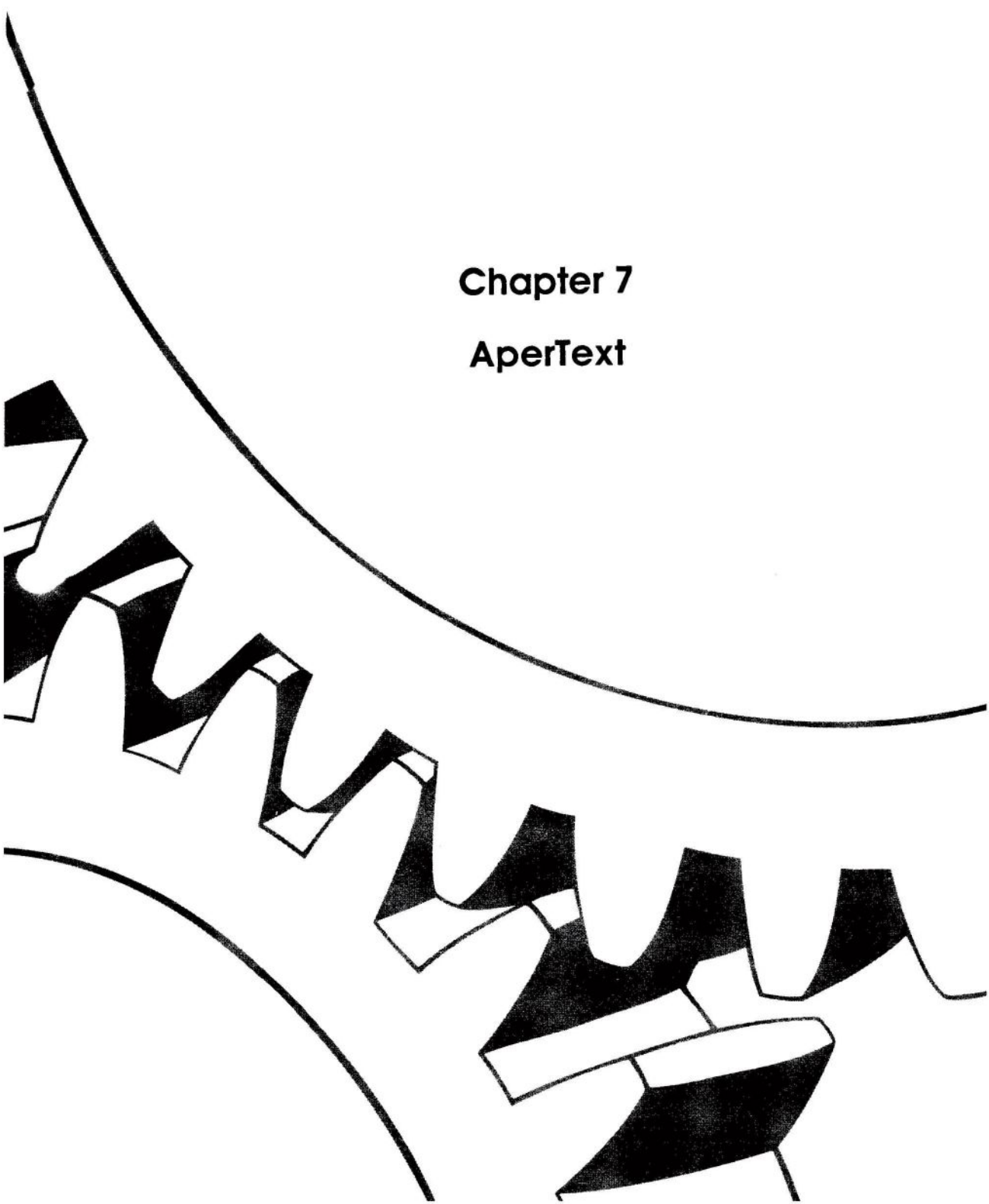
The next command, "SCALE 3," tells CADMOVER that the enlargement factor for the following translations is to be 3.

Then, the file "SMALL.STL" is to be translated into 3DGF. (Note the -f on the command line. This *temporarily* overrides the selection of DXF as the output format.) After that, "BIG.STL" and "MEDIUM.STL" are both translated into DXF.

For the last translation, "MEDIUM.STL" is translated again, but this time with a scale factor of 1 (note the -s in the command,) and the resulting output file will be named "NOSCALE.DXF" (note the -o in the command.) Finally, CADMOVER will BEEP three times and then quit. The translations have been done.

Chapter 7

AperText



AperText

Fig. 8-1: The AperText Application Icon

AperText is a Macintosh application which converts text files into aperture description files compatible with Kandu Software's CADMOVER® application.

Each line in the source file should contain four numbers:

1. D-Code
2. Shape code (1,2,3,4) = (square, rectangle,circle,ellipse) respectively
3. Aperture width
4. Aperture height

The measurement system for the aperture width and height is the same as the measurement system being used for the plotter geometry. See how your Gerber Preferences are set up in CADMOVER before filling in these fields.

The width and height may contain decimal points. The D-Code and Shape code may not.

There should be four numbers on each line. The preferred separator character is a comma, but blanks or tabs are OK as well. Each line should end with a single carriage return character, but the program will accept MS-DOS files (carriage return AND line feed) or UNIX files (line feed only) as well.

The first line in the file describes aperture number 1, the second line describes aperture number 2, and so on.

For example, the following lines describe an aperture list with 5 apertures:

```
10,1,25,25
11,2,25,75
12,2,50,100
13,3,10,10
14,4,25,10
```

The first line represents aperture number 1. Its D-code is 10, the shape is square, and the size is 25 plotter units on each side. Aperture number 2 has D-code 11, it is rectangular, 25 units wide and 75 units high. Aperture number 3 is also a rectangle, 50 units wide and 100 units high. Aperture 4 is a circle 10 units in diameter, and aperture 5 is an ellipse 25 units wide and 10 units high.

You can use any text editor (such as TeachText) to create an aperture list like the one above.

When you launch AperText, it will immediately show you an "Open..." Dialog, listing all files of type Text. Choose the file you want to translate. The translation is automatic, and the "Open..." Dialog will appear again. Then you may choose to translate another aperture list, or quit. (The escape key on the keyboard will activate the Quit button.) The generated output file will be named after the input file, with ".APERTURES" appended to the end.

You can launch CADMOVER using the new aperture list simply by double-clicking on the aperture list icon. CADMOVER will use the apertures in that file until you quit the application or change the aperture Preferences within CADMOVER itself.

Fig. 8-2: The Icon For The Aperture List File Created By AperText

Chapter 8

Troubleshooting

Troubleshooting

Overview

This section lists some possible areas of difficulty that might be encountered during the file translation process. In most cases, problems with the translation can be resolved by changing the settings of one or more controls in CADMOVER.

Part 1 — Things That Can Happen When Files Created By CADMOVER Are Read In To Other Applications

• Adobe Illustrator

Adobe Illustrator - Offending Command 'z'

This is a cryptic message, but it indicates that there is some problem with a text item in the input file. Usually, it is an indication that a text font size has become either too large or too small. Sometimes, it can occur if you have done a font mapping operation, and have specified a target font name that contains blanks.

Response

Check your settings in CADMOVER's "EPSF Output" dialog, page 60. These can help CADMOVER identify sizing problems. If you have used any font names that contain blanks, consider using a different font.

Adobe Illustrator - Erroneous Or Missing "%%Boundingbox"

This is a sizing problem. What this message really means is, "Your file will not print on the largest sheet of paper that this program can handle."

Response

Use CADMOVER's "Scale" dialog, page 27, to reduce the size of the file. Make sure that no dimension within the file is greater that 220 inches.

Adobe Illustrator - No Visible Data On Screen

The most likely cause for this is that your file has not been scaled down enough. If the file took a while to open, but you see nothing on the screen, then there probably is artwork in the file, but the default view is zoomed in on a blank area.

Response

You can retranslate the file with CADMOVER, or, for a possible quick fix, you can Select All and then use the magnifying glass tool to scale your entire file smaller.

• Aldus Freehand

Aldus Freehand - "Too Many Points in Path"

Response

Use CADMOVER's "Maxima" dialog to set the maximum number of vertices for an output polygon to 1000 or less. See page 84 for more information.

• AutoCAD

Curve Into Line Segments - Illustrator > DXF

"It was a curve in Adobe Illustrator, but now it's made of a bunch of short line segments."

Response

If the curves were originally circular segments, turn on "Recognize Arcs" in the "EPSF Input..." dialog. (Be sure to read the section in this manual that describes the "EPSF Input..." dialog, page 64.) For curves that are not intended to be truly circular, the conversion process will produce POLYLINE structures in DXF. The "Number of Spline Segments" field in the Preferences window can be increased to produce a larger number of shorter segments, which can make the curve appear smoother. See page 52 for "Number of Spline Segments" information.

Part 2 - Unexpected Events That Can Happen When Files Of Dubious Origin Are Read In To CADMOVER

Problem — Blank Screen

The "Display Picture" Preference is turned on and no picture appears on the screen.

Response

Turn on the "Gray Background" option. This problem is caused by drawing all-white graphics on a white screen. When the screen background turns gray, the graphic should appear. See page 50 for more informatjion.

Problem — Misplaced Text

(Dreams, MacDraft) The graphics look OK, but all the text is out of place.

Response

The file probably had not been saved in "1X" view. Reopen the file in the application that created it, and change the view magnification to 1X.

Problem — Unknown Dialect Of EPSF

A PostScript or EPSF file has an unrecognized dialect.

Response — A Further Explanation

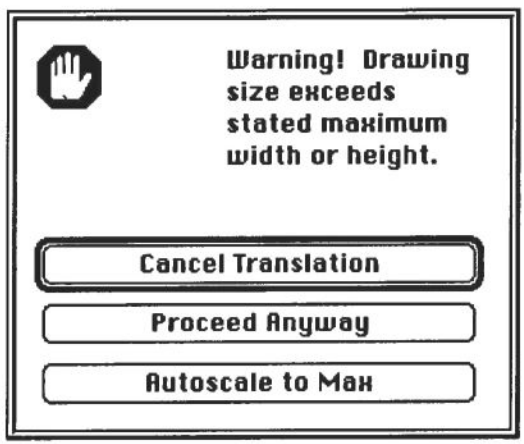
CADMOVER can open PostScript or EPSF files generated in a wide variety of applications, notably Adobe Illustrator, Aldus Freehand 2.0, AutoCAD, and MicroStation. But, PostScript files can take many forms, each of which

constitutes a dialect if you see the dialog above, it indicates that CADMOVER is able to recognize the format of your file, but does not know the dialect. Unfortunately, what this means is that CADMOVER will not be able to translate this file.

Kandu Software Corp. encourages users who have files like this to send copies of the files in for analysis. It is sometimes possible to make minor changes to the files to make them acceptable to CADMOVER. In any case, if you do send a file, please include a note with your name, address, phone number, and the reason you are sending the file.

Problem — Maximum Drawing Size Exceeded

The translated output file is larger than the maximum set in the "Maxima " Preferences Dialog.



The drawing size exceeds the maximum size specified by the "EPSF Output" preference dialog. Depending on what you intend to do with the file, it may or may not be useable. The most common way to get this dialog is to forget to scale when translating DXF into EPSF. It is important to remember that the limit that's been exceeded is only a caution limit; CADMOVER is just warning you that your file's drawing size may be larger than you expected it to be.

Response

The buttons in the "EPSF Output" dialog list the possible actions.

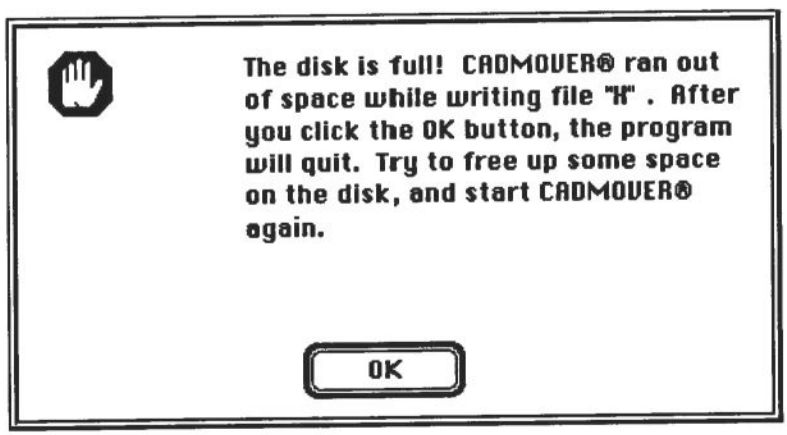
- Cancel Translation: This halts the translation, but leaves the input file open and untouched so that you may use the Scale dialog to scale your drawing.

- Proceed Anyway: This is not necessarily a bad idea. But if you try to open the resulting file in an application such as Adobe Illustrator, and you get a message along the lines of "invalid or missing %%BOUNDING BOX", try translating the file again at a smaller scale.

- Autoscale to Max: This button tells CADMOVER to scale your file such that the larger of its width and hieght does not exceed the value specified by the drawing size limit. You can inquire or change the drawing size limit by using the "EPSF Output..." button in the main Preferences window (command-;).

Problem — Disk Is Full

Either the hard disk or the floppy disk has run out of space during a file translation.



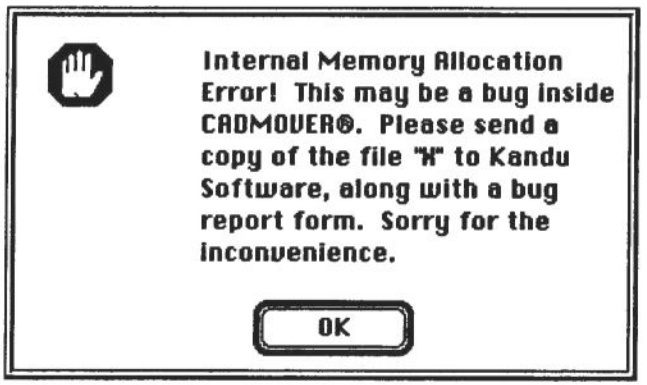
This message appears when the hard disk that contains CADMOVER's output file fills up. In this example, "X" is the name of the output file. CADMOVER is not able to recover from this condition, so it will quit after you click the OK button.

Response

Either free up space on the hard disk you are using, or direct CADMOVER to place the output file on a different hard disk.

Problem —Memory Allocation Error

CADMOVER reports an internal memroy allocation error using the dialog below.



This message indicates fairly serious trouble with your translation. Very often, this message is the result of reading a corrupted file into CADMOVER. But sometimes, it can result from a valid file. In either case, the makers of CADMOVER would be happy to take a look at whatever is causing the problem. See the section on Trouble Reporting (page 152) for information on how to send a file to Kandu.

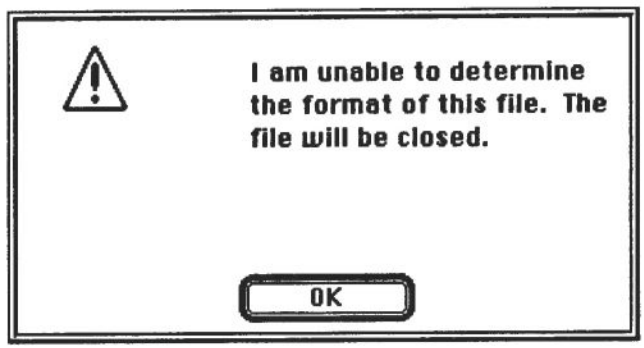
Before you send a file, however, please check to make sure that the file is of a version that CADMOVER can accept. For example, CADMOVER does read Adobe Illustrator 88 files, but not Adobe Illustrator 5.0 files.

Response

If you are sure that your file is an appropriate version, and it has not been corrupted, please send it to Kandu.

Problem — Unknown File Format

CADMOVER does not recognize the file type you would like to translate.



This message appears when you ask CADMOVER to open a file whose format is unknown to it. In general, CADMOVER will only put files of known types into the list of files shown in the "Open..." dialog. But several formats are normally stored in files of type TEXT, including DXF, IGES, HPGL, and others. Therefore, CADMOVER will attempt to open any file of type TEXT.

Response

Check your file to be sure that it has not been corrupted. If you are trying to translate an HPGL file, be sure that the file's name ends with ".HPG", since CADMOVER will use that suffix as a clue in determining the file's format.

Trouble Reporting

If you are having a problem that cannot be rectified using any of the suggestions above, please put your input file on a floppy disk, and send it to the Technical Support Department at Kandu Software Corporation.

Be sure to include the following information:

Your name.
Your address.
Your telephone number.
A general overview of the problem.
The format of the file you are trying to translate.
The format you are trying to translate into.
The scale factor you are using (if you are scaling your file).
The exact model of computer you are using, e.g., Mac IIci, 80MB hard disk, 8MB RAM, etc..
The System level you are using, e.g., System 6, System 7.1, etc.
The settings of any options that may have a bearing on the translation, e.g., if you are translating an Adobe Illustrator file, indicate the settings of the "EPSF INPUT..." dialog.

Also, it is particularly useful if you can provide a printed picture of what the file is supposed to look like.

In general, the more detailed information you can provide, the more quickly your problem is likely to be resolved.

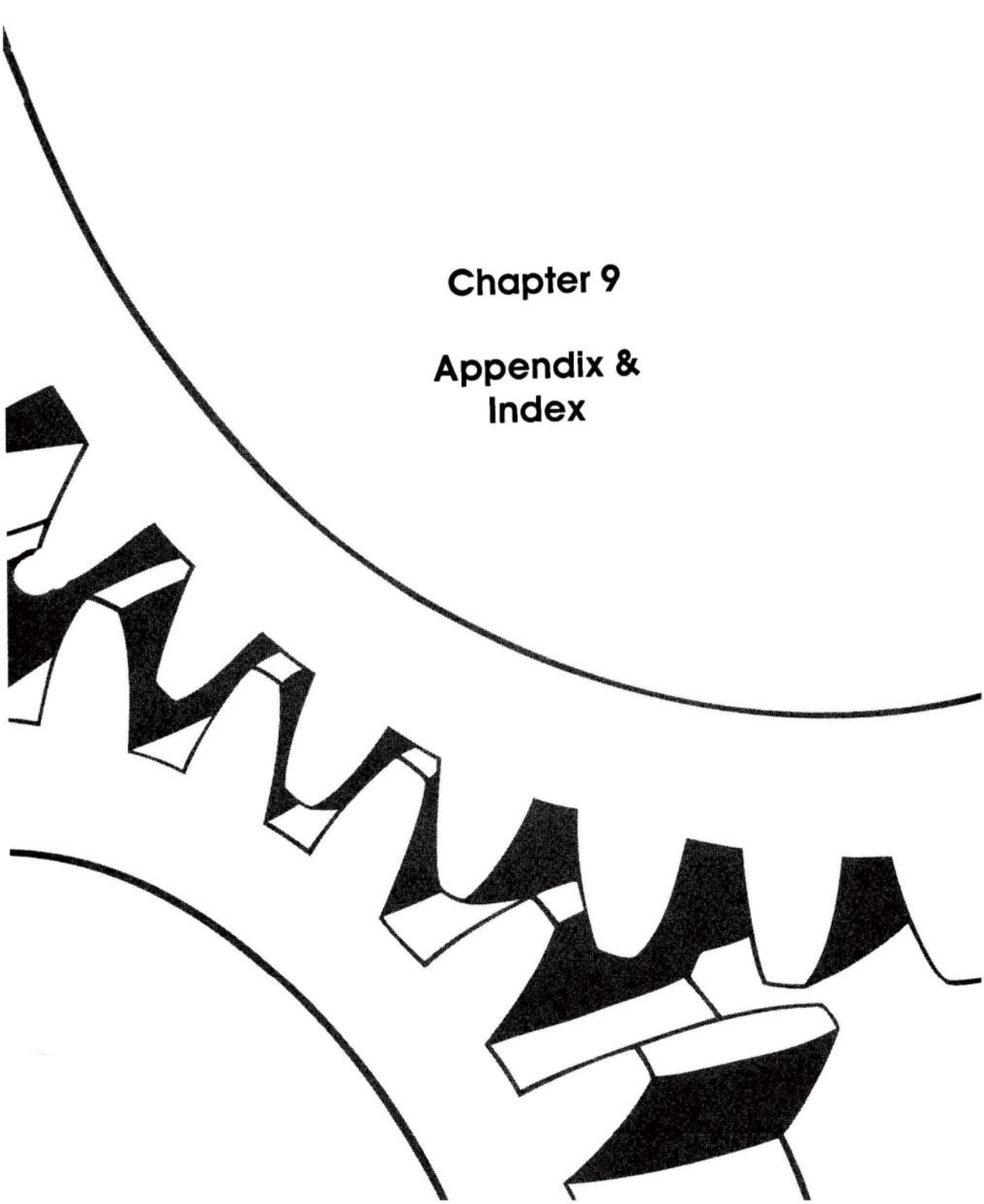
Send the disk and other information to:

Kandu Software Corp.
131 Great Falls Street, Suite 100
Falls Church, VA 22046
Attn: Problem File

The "Attention" line is very important. Without it, your package may not receive the attention it deserves.

Chapter 9

Appendix & Index



Appendix

CADMOVER Reads All of the Following Formats:

3DGF
Adobe Illustrator 1.1
Adobe Illustrator 88
Architrion II
CGM
ClarisCad
DDES2/IT8.6
Dimensions
Dreams
DXF Binary
DXF v. 9 (2D & 3D)
DXF v. 10 (2D & 3D)
DXF v. 11 (2D & 3D)
DXF v. 12 (2D & 3D)
ENVISAGE 3D
EPSF AutoCAD v.11
EPSF AutoCAD v.12
EPSF Freehand v. 2.02
EPSF Gplot
EPSF Harvard Graphics
EPSF ICIS
EPSF Mecca
EPSF MicroStation
EPSF Mitre
EPSF Pro Engineer (vector only)
EPSF Schema
Focus
Focus Text
Gerber Plotter
HPGL
IGES v. 1.0 (2D & 3D)
IGES v. 2.0 (2D & 3D)
IGES v. 3.0 (2D & 3D)
IGES v. 4.0 (2D & 3D)
IGES v. 5.0 (2D & 3D)
IGES v. 5.1 (2D & 3D)
MacArchitrion
MacDraft
MacDraw II
MiniCad+ Text v. 1.0 (2D & 3D)
MiniCad+ Text v. 2.0 (2D & 3D)
MiniCad+ Text v. 3.0 (2D & 3D)
MiniCad+ Text v. 4.0 (2D & 3D)
PICT
Presenter Professional
SCULPT 3D 2.0
SCUPLT 3D 1.0
STL (Stereolithography)
STL Binary
Super 3D Text
Wavefront (.OBJ)
Zoom
Zoom Text

CADMOVER Writes All of The Following Formats:

3DGF
Adobe Illustrator v. 1.1
Adobe Illustrator v. 88
Adobe Illustrator v. 3.0
Adobe Illustrator v. 5.0
CGM
DDES2/IT8.6
DXF Binary
DXF v. 9 (2D & 3D)
DXF v. 10 (2D & 3D)
DXF v. 11 (2D & 3D)
DXF v. 12 (2D & 3D)
Dimensions
Dreams
ENVISAGE 3D
Focus
Focus Text
Gerber Plotter
HPGL
IGES v. 1.0 (2D & 3D)
IGES v. 2.0 (2D & 3D)
IGES v. 3.0 (2D & 3D)
IGES v. 4.0 (2D & 3D)
IGES v. 5.0 (2D & 3D)
IGES v. 5.1 (2D & 3D)
MacDraft
MiniCad+ Text v. 1.0
MiniCad+ Text v. 2.0
MiniCad+ Text v. 3.0
MiniCad+ Text v. 4.0
PICT
Postscript
Presenter Professional
STL
STL Binary
SCULPT 3D v. 1.0
SCULPT 3D v. 2.0
Super 3D Text
Wavefront (.OBJ)
Zoom
Zoom Text

CADMOVER 4.0 Is Compatible* With The Following Programs:

Program	Format	Import/Export
Adobe Illustrator	EPSF	R/W
Anvil 5000	IGES	R/W
Aperture	DXF	R/W
Archicad	DXF	R/W
Architrion II	Internal, Text	R
Ashlar Vellum	IGES, DXF, PICT	R/W
AutoCAD	DXF, IGES, HPGL, EPSF	R/W (EPSF R/Only)
BluePrint	DXF, PICT	R/W
CADAM	IGES	R/W
CadKey	IGES	R/W
Canvas	IGES,CGM,DXF,HPGL, EPSF, PICT	R/W

CATIA	IGES	R/W
ClarisCad	Internal, PICT	R , R/W
Corel Draw	CGM,DXF,HPGL, Ill.88	R/W
Design Cad 2D/3D	DXF	R/W
Design Studio	EPSF	W
Die CAD	DDES2	R/W
Dimensions	Internal	R/W
Douglas CAD	IGES	R/W
Dreams	Internal, PICT	R/W
DynaPerspective	DXF	R/W
Electric Image	DXF, Wavefront, IGES	R/W
ENVISAGE 3D	Internal	R/W
EPS Exchange	EPSF	R
Focus	Internal, Text	R/W
Form-Z	DXF	R/W
FrameMaker	EPSF, PICT	W
Freehand	EPSF, PICT	W, R/W
Generic Cadd	DXF	R/W
GPLOT	EPSF	R
Harvard Graphics	CGM, EPSF, HPGL	R/W,R
HiJaak	DXF	R/W
ICIS	EPSF	R
Infini-D	DXF	R/W
Intergraph	DXF, IGES	R/W
Interleaf	EPSF	W
Laser Point CAD	DXF	R/W
MacArchitrion	Internal, Text	R
MacBravo!	IGES	R/W
MacDraw II	Internal, PICT	R, R/W
MacLink Pro	DXF, PICT	R/W
MacProject	PICT	R/W
MacroMedia Three-D	3DGF	R/W
MacSurf	DXF	R/W
MapGrafix	DXF	R/W
MarcoModel	3DGF	R/W
Mecca	EPSF	R
MicroCadam	DXF, IGES	R/W
Micrografx Designer	DXF	R/W
MicroStation	DXF, IGES	R/W
MicroStation	EPSF	R
MiniCad+	DXF, HPGL, EPSF, Text	R/W
Misomex	DXF	R/W
Mitre	EPSF	R
Modelshop	DXF	R/W
MS Word	EPSF, PICT	W

Pagemaker	EPSF	W
Presenter Professional	Internal, DXF, Dimensions	R/W
Pro Engineer	EPSF	R
Quark Express	EPSF	W
Ragtime	PICT	R/W
Ray Dream Designer	PICT, 3DGF	R/W
Schema	EPSF	R
SCULPT 3D/4D	Internal, DXF, 3DGF	R/W
Sculpt Model	DXF, 3DGF	R/W
Sketch	DXF, PICT, EPSF, IGES	R/W
Stratavision	IGES, DXF, PICT, EPSF	R/W
StreamLine	PICT, EPSF,DXF	R/W
Super 3D	Text	R/W
Touch 3D	DXF	R/W
Unigraphics II	IGES	R/W
Ventura Publisher	EPSF, CGM, HPGL	W
VersaCAD	DXF, IGES	R/W
Virtus Walkthrough	DXF, ClarisCad, MacDraw II	R/W, R, R
Wavefront	.OBJ	R/W
Zoom	Internal, Text	R/W

*Compatibility is based on customer registration card information and interviews with a variety of CADMOVER users during January and February 1994 and all of 1987, 1988, 1989, 1990, 1991, 1992, 1993.

Index

C

D

D

E

F

G

H

I

I

J

L

M

N

O

P

P

Q

R

S

S

T

U

V

W

Z