

Introduction to EViews 2.0

Overview of product

EViews provides regression and forecasting tools on Windows computers. With EViews you can develop a statistical relation from your data and then use the relation to forecast future values of the data. Areas where EViews can be useful include:

- Sales forecasting
- Cost analysis and forecasting
- Financial analysis
- Macroeconomic forecasting
- Simulation
- Scientific data analysis and evaluation

EViews is a new version of a set of tools for manipulating time series data originally developed in the Time Series Processor software for large computers. The immediate predecessor of EViews was MicroTSP, first released in 1981. Though EViews was developed by economists and most of its uses are in economics, there is nothing in its design that limits its usefulness to economic time series. Even quite large cross-section projects can be handled in EViews.

The basic data object within EViews is the time series. Each series has a name, and you can request operations on all the observations just by mentioning the name of the series. EViews provides convenient visual ways to enter time series from the keyboard or from disk files, to create new series from existing ones, to display and print series, and to carry out statistical analysis of the relations among series.

EViews uses the visual features of modern Windows software. You can use your mouse to guide the operation with standard Windows menus and dialogs. Results appear in windows and can be manipulated with standard Windows techniques. Alternatively, you may use EViews' powerful command language. You can enter and edit commands in the command window. You can create and store the commands for a complete research project.

Some of the most important basic capabilities of EViews are:

- Entering, extending, and correcting time series data.
- Computing a new series, based on a formula of any complexity.
- Plots of series on your screen or printer, scatter diagrams, bar graphs, and pie charts.
- Ordinary least squares (multiple regression), least squares with autoregressive correction and two-stage least squares.
- Nonlinear least squares.
- Probit and logit estimation of binary choice models.
- Linear and nonlinear estimation of systems of equations.

- Pooled cross section-time series estimation and forecasting
- ARCH-GARCH estimation and forecasting
- Estimation and analysis of vector autoregressive systems.
- Descriptive statistics: correlations, covariances, autocorrelations, cross-correlations, and histograms.
- Seasonal autoregressive and moving average error processes.
- Polynomial distributed lags.
- Forecasts based on regression.
- Solution of simultaneous models.
- Time-series database management.
- Reading and writing of data files in standard spreadsheet formats.

The EViews Screen

Upon starting EViews, you will be presented with the main screen which consists of the top *main menu*, the *command window*, the middle *work area* and the lower status bar.

The Main Menu

The main menu consists of the following menu items

File	Basic file operations
Edit	Basic editing operations
Objects	Commands for creating and manipulating EViews objects
View	Viewing options
Procs	Importing and exporting data; sample options
Quick	Shortcuts to many EViews operations
Options	Set program options
Window	Switch between windows etc.
Help	Online help system

Note: Specific details about each of these items can be found in **Help/Menu Items**.

The Command window

The command window lies just below the main menu. This is where you enter EViews commands. Many of the operations in EViews can be executed by either typing commands in the command window or by using menu commands. The blinking vertical insertion bar at the left end of the command window indicates EViews' receptivity to a command.

The command you type in the window will be executed as soon as you hit ENTER.

As you enter commands, EViews will remember them. You can scroll up to an earlier command, edit it, and hit ENTER. The modified command will be executed again.

You can resize the command window so that additional previous commands are visible. Use the mouse to move the cursor to the bottom of the window, hold down the mouse button, and drag the bottom of the window downwards.

The Work area

The area in the middle of the window is the work area where EViews will display the various object windows that it creates. Think of these windows as similar to the sheets of paper you might place on your desk as you work. The windows will overlap each other with the foremost window being in focus or active. Only the active window has a darkened titlebar.

When a window is partly covered, you can bring it to the top by clicking on its titlebar or on a visible portion of the window. You can also cycle through the displayed windows by pressing the F6 key.

You can move a window by clicking on its titlebar and dragging the window to a new location. You can change the size of a window by clicking at the lower right corner and dragging the corner to a new location.

The Status bar

The status bar initially shows the default path and the name of the current workfile. During certain EViews operations, messages are also displayed in this area. For example, if EViews is using an iterative algorithm the progress of the algorithm will be displayed on the status bar.

Help System

Most of the information in the EViews manuals is available in online help. Selecting **Help/Contents** from the main menu displays the following topics:

Basics	Overview of EViews basics
How Do I?	Help on performing certain tasks
What is?	Definitions of econometric and forecasting terms
Menu Items	Explanation of menu items
Commands	Manual entries on EViews commands
For MicroTSP Users	Help converting from MicroTSP

In what follows, I will often give specific references to certain help topics.

The Workfile

Before you can do any work in EViews you need to create or load a workfile. Workfiles organize and contain the objects that you will use in your analysis. To create a new workfile use the **File/New/Workfile** command. This brings up a dialogue box where you describe the type of data you will be working with. EViews allows you to work with time series data or (undated) cross

section data. For time series data you must specify the frequency of the data (annual, quarterly, monthly etc) and the starting and ending dates of the desired sample. For cross section data you simply need to specify the maximum number of observations.

Date formats (see **Help/Search/Dates**)

EViews uses dates to identify time periods. Rules for composing dates are:

Annual: the full year, for example, 1981, 1895, 2001, or, for the twentieth century, the last two digits, for example, 95.

Quarterly: the full year or the last two digits of the year, colon, and the quarter number.
Examples: 1992:1, 65:4, 2002:3.

Monthly: the full year or the last two digits of the year, colon, and the month number. Examples: 1956:1, 1990:11.

Weekly and daily: Month number, colon, day number, colon, and year. For weekly data, the week is identified by the first day of the week. Example: 3:10:87 is March 10, 1987 or the week starting that day. You may also use **Options/Dates-Frequency** in the main menu to switch to European notation where 3:10:87 means 3 October 1987.

You may use a period in place of the colon in a date: 59.1, 59:1, 1959.1, and 1959:1 are all equivalent.

Here are some more examples:

89:2	means the second quarter of 1989 or February 1989
89:02	means February 1989
92:4	means the fourth quarter of 1992 or April 1992
82	means the year 1982
1979:2	means the second quarter of 1979 or February 1979
1882	means the year 1882
68:5	means May 1968
76:56	is a mistake, because there is no 56th month
9:30:96	is September 30, 1986 or the week starting that day
2:30:93	is a mistake because there is no February 30

Example:

To create a workfile for the GA caseload data do the following:

File/New/Workfile

Name of Workfile: caseload

Workfile frequency: monthly

Start date: 1973.07

End date: 2002:12

You will see the workfile window titled with the name of the workfile. The workfile will be empty except for two default objects labeled **c** and **resid**. I will comment on these objects later. The **Range** of the workfile gives the starting and ending dates of the data - the entire sample of the data. The **Sample** range of the workfile gives the sample that is relevant for analysis. By default, the **Sample** is set to the workfile **Range**. You can change the effective sample using the **Sample** command. More on this later.

Saving the workfile

You can save the workfile to disk by either pressing the [**Save**] button on the workfile toolbar or by choosing **File/Save** from the main menu. When you save a workfile, all of the objects currently named in the workfile will be saved.

Workfile buttons

The following workfile buttons provide shortcuts to certain EViews commands:

[View]	View objects in workfile
[Procs]	Import/Export objects and misc. Procedures
[Save]	Save workfile
[Show]	Show objects in window
[Fetch]	Fetch .db files into workfile
[Store]	Store objects in workfile to disk
[Delete]	Delete objects from workfile
[Genr]	Generate new objects from objects in workfile
[Sample]	Set sample range

The operation of these buttons will be illustrated in the example to follow.

Workfile objects

Workfiles can contain the following types of objects:

Series	Data series
Group	Groups of data series
Matrix-Vector-Coeff equation	Matrices, vectors or parameter coefficients
VAR	Equations for estimation
System	Vector autoregression
Pool	System of equations
Model	Pooled sample
Sample	

Table	Spreadsheet table
Graph	Graphics

For more information on objects see **Help/What is?/Object**.

Note: Every object that has a name is saved whenever a workfile is saved. Some objects are created by EViews in the process of running certain commands and are not given names. These are temporary objects and will not be saved when the workfile is closed. More later.

Importing Data

Importing ASCII text data

EViews can easily import ASCII text data. The data can be organized in rows or columns and the names of series can be in the file as long as they precede the data series. Before importing the data you need to know how missing values are coded in the text file.

Example:

The data is in UXCASE.TXT. Single data series is in a column with the name of the series in the first row. Missing values are coded as NA. The EViews commands to import the data are:

Proc/Import Data

File name: uxcase.txt

Order of data: by observation - series in columns

Text for NA: NA (missing value code)

Names for series or number of series if series in columns: 1

The series uxcase has been added to the workfile.

Deleting series from the workfile

To delete a series from the workfile simply highlight the series to be deleted and click [**Delete**] on the workfile toolbar. This operation does not affect data that are stored on disk.

Importing Excel workfiles

Data in Excel worksheets can easily be imported into EViews. Data can be organized in rows or columns and the names of series can be in the file as long as they precede the data series. Before importing the data you need to determine the upper left data cell in the worksheet.

Example:

The data is in UXCASE.XLS. There are two data series in columns with the name of the series in

the first row. The data series are in column B and the dates of the data are in column A. The first data point is in cell B2. Missing values are coded as blanks. The EViews commands to import the data are:

Proc/Import Data

File name: uxcase.xls

Order of data: by observation - series in columns

Upper left data cell: B2

Names for series or number of series if series in columns: 1

Note: missing values have been coded by EViews to NA.

Fetching databank series

EViews Databank files are text files that contain data and basic information about the data such as labels and the sample range. These files can be imported into a workfile with a compatible range using the [**Fetch**] command.

Example:

The data is in the file UXCASE.DB. To import this file into EViews simply click [**Fetch**] on the workfile toolbar, change to the directory to where the file uxcase.db is located, double click on uxcase.db and press [**OK**]. The series uxcase will be added to the workfile.

Alternatively, you can use the fetch command in the command window as follows. Click in the command window and type: **fetch uxcase**. (Note: you may need to first specify the correct path using the **chdir** command).

Entering data from the keyboard

Data can be entered directly into a workfile. To do so, you first need to create a series object and then enter the data for that object using the EViews editor.

Example:

Objects/New Object

Type of Object: Series

Name for Object: uxcase2

The empty series uxcase2 will be added to the workfile. To enter data into this series do the following:

- ! Double click on uxcase to open the Spreadsheet view (you will see a bunch of NAs)
- ! Click [**Edit+/-**] to activate edit mode
- ! Click [**Sample+/-**] to shift the data to one column

- ! Click on the first data cell
- ! Enter the data for that cell and press <Enter>

When you are finished entering the data, click [**Edit+/-**] to deactivate edit mode and then click the close box.

Note: Data can also be cut from other programs and pasted into the EViews editor.

Storing individual series

You can save individual series in a workfile using the [**Store**] command. Simply highlight the series you wish to store and click the [**Store**] button. The individual series will be saved in EViews databank (.db) format on disk according to the current path (as indicated in the lower right hand corner of the status bar).

Data Series object

Data series are the main objects in EViews. EViews assigns several views as well as procedures to data series. These views and procedures can be seen by double clicking on a series which opens a Series Window. By default, the spreadsheet view is initially displayed in the series window. This shows the raw data in a spreadsheet.

Series window buttons

At the top of the series window are a collection of command buttons:

[View]	Opens view menu
[Proc]	Opens procedures menu
[Objects]	Opens objects menu
[Print]	<i>Immediately</i> prints what is in the series window
[Name]	Opens the Name Object dialogue box
[Freeze]	<i>Immediately</i> Freezes the view of the object
[Edit+/-]	Toggles Edit mode
[Insert/Del]	Opens insert/delete dialogue box
[Smpl+/-]	Displays single series in one column
[Label+/-]	Toggle display of label
[Title]	Opens Make title dialogue box

Note: Clicking on some buttons creates a menu whereas clicking on other buttons immediately executes a command.

Naming and Labeling Series

You can change the name of a data series using by clicking [**Name**] in the series window. You can add labels - user supplied comments on a series - in two ways. From the spreadsheet view,

click [**Edit+/-**] to toggle edit mode of the spreadsheet. Then click on a blank row above the data series and enter the information you want. Alternatively, click [**View**] and select **Label**. Enter the information you want and then close the series window.

An overview of Series Views

Clicking on the [**View**] button in the series windows opens the view menu providing the following views of a series:

Spreadsheet	Display series in spreadsheet
Line graphs	Display series as a line graph
Bar graph	Display series as a bar graph
Histogram and stats	Display histogram and simple descriptive statistics
Tabulate	Compute cumulative distribution of series
Correlogram	Compute and plot autocorrelations and partial autocorrelations
Unit Root Test	Compute unit root tests
Label	Show series labels and comments

Example:

Double click on *uxcase* to open the default spreadsheet view. Click [**Smpl+/-**] to display the data in a single column. Notice that missing values are coded as NA. Next click [**View**] and select **Line graph**. You will see the view change to a time plot of *uxcase* over the entire sample period. Notice that the missing values are not plotted. Clicking [**View**] and then **Histogram and Stats** shows a histogram and some simple descriptive statistics including the mean, variance, skewness and kurtosis. Since the data in *uxcase* is trending upward these statistics do not have too much meaning. The **Tabulate** view shows the cumulative distribution of the data - again this is not too interesting for trending data. Clicking [**View**] and then **Correlogram** produces a plot of the *autocorrelation* and *partial autocorrelation* functions of the data (for a user specified lag length) as well as the numerical values of these functions. These functions provide information about the degree of serial dependence or predictability in the data. Notice that the autocorrelation function for *uxcase* declines very slowly. This is typical for trending data because the trend is highly predictable. Next, clicking [**View**] and then **Unit Root Test** brings up the unit root test dialogue box. *Unit root tests* are useful to determine if the trend in the data is smooth and highly predictable or erratic and not very predictable.

Freezing a view

Views displayed in a series window can be “frozen” - i.e. like a snapshot of the view - using the **Freeze** command or [**Freeze**] button. The advantage of **Freeze** is that the tables and graphs created by Freeze may be edited extensively for presentations or reports. Also frozen views do not automatically change when the workfile sample is changed or when the data change.

Freezing spreadsheet produces table object and freezing graph produces graphic object

Example:

Highlight `uxcase` and double click. Select [**View**], **Spreadsheet** and then click [**Freeze**]. A new window titled “Table: Untitled” will appear with the data in a spreadsheet. This is like a snapshot of the data and it is disassociated with the data in the series `uxcase`. This means that you can change the data in the table and it will not change the data in the series `uxcase`. Initially the table is untitled and as such is only a temporary object. If you click the close box the table will be removed from memory and the workfile. To make the table permanent click [**Name**] and give the table a name. Now close the table window. You will see that a new (table) object has been added to the workfile.

Now let's create a frozen graph. Highlight `uxcase` and double click. Select [**View**], **Line Graph** and then click [**Freeze**]. A new window titled “Graph: Untitled” will appear with the data presented in a line graph. Initially the graph is untitled and as such is only a temporary object. If you click the close box the graph will be removed from memory and the workfile. To make the graph permanent click [**Name**] and give the graph a name. Now close the graph window. You will see that a new (graph) object has been added to the workfile.

Note: You can save table and graphic objects separately from the workfile using the **store** command.

Setting the Sample of Observations

One of the important concepts of EViews is the sample of observations. The sample is the set of observations to be included in displays or in statistical procedures. It is frequently a subset of the range of observations included in the workfile.

The [**Sample**] button in the workfile toolbar sets the sample of observations for operations within the workfile. When you push the button, you will see a dialog asking for information about the observations to use.

In the upper window you should enter pairs of dates. Each pair identifies a starting and ending observation for a sequence of observations to be included in the sample. Most of the time, you will just use a single pair. For example, the pair 1980:01 1995:12 informs EViews that only observations for 1980:01 through 1995:12 are to be used in future operations. You need to provide the sample information only when you want to change the observations you are using. When you create a workfile, the sample is set initially to be the entire range of the workfile.

The sample can also be set using the EViews **smpl** command (not the **sample** command - this is different) in the command window. For example, to set the sample to the period 1980:01 1995:12 you would type: **smpl 1980:01 1995:12**.

You may specify sample ranges that include breaks by providing multiple pairs of dates. For example 1980:01 1985:12 1990:01 1995:12 indicates that observations for 1980 through 1985 and observations for 1990 through 1995 are to be used. Observations for 1981 through 1984 are

excluded.

The lower part of the sample window allows you to use IF conditions to allow the data to determine the sample. The sample is the intersection of the set of observations defined by the pairs in the upper window and the set defined by the conditions in the lower window, as expressed in a formula. Here are some examples:

- ! Click [**Sample**]
- ! Upper window: **80:01 93:12**
- ! Lower window: **uxcase>12000**

means the sample is those months from 1980:01 through 1993:12 when the series uxcase is greater than 12000.

Data transformations

Very often you will need to create new series that are some function of the series in your workfile. This can easily be done using the **Genr** command in the command window or by clicking on [**Genr**]. For a list of common transformations see **Help/Search/Functions**.

Note: Series generated by the Genr command are computed only for the observations specified in the current sample specification.

Example:

Three useful transformation are differencing, logging and lagging. The built-in EViews functions for the first two transformations are **D()** and **log**. EViews has a special notation for lagging. To create the natural log of uxcase do the following:

- ! Click [**Genr**] to open the Generate Series by Equation dialogue box
- ! Under **Enter Equation** type: **luxcase = log(uxcase)**
- ! Leave the sample information alone
- ! Click [**OK**]

Notice that the new series luxcase has been added to the workfile.

Next, create the first difference of luxcase by using the **Genr** command in the command window:

- ! Click in the command window
- ! At the insertion point type: **genr dluxcase = d(luxcase)**
- ! Press <**Enter**>

The series dluxcase has been added to the workfile. Recall, the difference in the log of a variable is approximately the percent change or growth rate in the variable.

Finally, let's create a series containing the value of `uxcase` but lagged one month. In EViews, a lagged series is indicated by putting the lag in parentheses with a minus sign. Therefore, to create lagged `uxcase` do the following:

- ! Click in the command window
- ! At the insertion point type: **genr uxcase_1 = uxcase(-1)**
- ! Press <Enter>

To illustrate how formulas work in generating variables, let's compute the percentage change in `uxcase` directly:

- ! Click in the command window
- ! At the insertion point type: **genr pchuxcase = (uxcase - uxcase(-1))/uxcase(-1)**
- ! Press <Enter>

Notice the use of parenthesis in the formula expression.

Generating trend and seasonal variables

For forecasting purposes, it is often necessary to generate deterministic time trend and seasonal variables. EViews has special built-in functions for creating such elements. As in Excel, these special functions are preceded by an "@" character. For a list of special functions see **Help/Search/@functions**.

Example:

To generate a deterministic time trend variable starting in 1973:01 use the **@trend** function as follows:

- ! Click in the command window
- ! At the insertion point type: **genr trend = @trend(1973:07)**
- ! Press <Enter>

The variable `trend` has been added to the workfile. Note: the number in parenthesis of the **@trend** function specifies the starting date of the trend.

To create seasonal dummy variables for each month use the **@seas** command. The format of the command is as follows: **@seas(d)** produces a dummy variable that is equal to 1 in period `d` (month `d` since the workfile frequency is monthly) and zero in all other months. Note that in order to create a full set of monthly seasonal dummy variables you have to use the **@seas** function 11 times. To create 11 monthly seasonal variables do the following:

- ! Click in the command window
- ! At the insertion point type: **genr feb = @seas(2)**

- ! Press <Enter>
- ! Repeat 10 more times for the remaining months increasing the argument in @seas() by one each time.

Transformation of series and the sample

As mentioned earlier, any series computed using the **Genr** command is only computed according to the current sample specification. Failure to keep this in mind can lead to strange and confusing results.

Example:

- ! [Sample]
- ! upper window: 1980:01 1990:12
- ! lower window: leave blank
- ! [OK]
- ! [Genr]
- ! upper window: temp = uxcase^2
- ! lower window: 1980:01 1990:12

Notice that EViews automatically puts the sample period in the lower window which corresponds with the current sample setting for the workfile. If you were to use the Genr command in the command window the same result would occur but you would not be reminded about the restricted sample setting.

The variable temp will only be computed for the subsample period 1980:01 1990:12 and will have missing values (NAs) for the remaining observations.

Generating dummy variables

Dummy variables - variables that take 0-1 values - are often used in forecasting to pick up the effects of structural change, seasonality, one-time events like strikes or legislation changes etc. The most common type of dummy variables are

- impulse dummies** used mostly to deal with outliers
- step dummies** used to deal with structural change

Example:

To illustrate the use of a impulse dummy to deal with an outlier let's add an outlier to uxcase data in 1990:01:

- [Genr]
- upper window: uxcase2 = uxcase
- lower window: leave blank

[Genr]

upper window: **uxcase2 = 5000**

lower window: **1990:01 1990:01**

The preceding creates a new variable called `uxcase2` that is the same as `uxcase` except in 1990:01 where the value is 5000.

Now let's create a dummy variable equal to 1 in 1990:01 and 0 everywhere else:

[Genr]

upper window: **i1990 = 0**

lower window: leave blank

[Genr]

upper window: **i1990 = 1**

lower window: **1990:01 1990:01**

Groups

You can create an object called a group to make it easier to deal with a number of series at the same time. A group is basically a list of names of series. It is not a copy of the data; if you change the data for one of the series in a group, you will see the new data when you work with the group. Also, if you delete a series from the workfile, it will disappear from any group that used to include the series. And renaming a series changes the name in every group containing the series.

Like other data objects, the group has a window. The standard view in the group window is a spreadsheet, with the series in columns. Other views are multiple graphs and descriptive statistics. The easiest way to create a group of existing series is to select the series names in the workfile directory. Then double click on any of the selected names, select **Open Group**, and the spreadsheet view of your group will appear.

Example:

It is useful to create a group variable containing all of the seasonal dummy variables. To do this highlight all of the monthly seasonal variables and then double click. This will open a group variable window showing all of the highlighted series in a spreadsheet. To permanently create the group variable click [**Name**], call the group **Seasons** and then close the group window. A new group object called **Seasons** will be added to the workfile. The usefulness of this action is that in estimation you can refer to the single group variable `seasons` instead of the 11 individual variables.

Analyzing multivariate relationships

Groups are also useful for quickly analyzing multivariate relationships. EViews has special functions assigned to group variables for the analysis of multivariate relationships.

Example:

Suppose you are interested in the relationship between `uxcase` and `uxcase_1`. Highlight both series and double click to create a group object. A group window will appear with both series displayed in a spreadsheet view. For a group variable, the [**View**] menu is adjusted to incorporate different views of multiple time series. The menu items are now:

Spreadsheet	Show raw data for all variables
Graph	Display all series on one graph
Multiple Graphs	Display series in individual graphs
Descriptive Stats	Compute descriptive stats for each series
Crosstab	Compute cross tabulations
Correlations	Compute sample correlation matrix
Covariances	Compute sample covariance matrix
Correlogram	Compute correlogram for 1 st series in group
Cross Correlogram	Compute cross correlogram between 1 st and 2 nd series only
Cointegration Test	Test for cointegration (common trend)
Granger Causality	Compute tests for pairwise Granger Causality

Note: Some of the above functions are only operative for pairs of data.

Graphics

To graph a single series, double click on its name in the workfile directory. This will open a series window showing the spreadsheet view. Use **View/Line Graph** to show the series as a line with dates on the horizontal axis. Use **View/Bar Graph** to show each observation of the series as a vertical bar.

To graph two or more series, select the series from the workfile directory and open a group window. Then use **View/Graph** to change the view to a graph. You will see your series plotted as lines, each a different color. Alternatively, you can use **View/Multiple Graphs** to show a separate line graph for each series.

There are many ways you can change the appearance of a graph. Most of them are available in a big dialog box you can open by **double clicking anywhere in the central area of the graph**. Your most basic choice is the **Graph Type**. The various types available are:

Line Graph	where each series shows with a vertical distance proportional to its value
Stacked Lines	where each line is the sum of all the preceding series, or, to put it another way, the distance between lines is the value of that series,
Bar Graph	where the value of each series is shown as the height of a bar,
Stacked Bars	where the overall height of each bar depicts the sum of the series and the value of each series is shown by the height of a colored segment of the bar,
Mixed Bar and Line	where the first series is shown as a bar graph and the remaining series are shown as lines,

Scatter Diagram where the first series provides the horizontal coordinate of a point (plotted with a +) and the second series provides the vertical coordinate, and

Pie Chart where each observation is graphed as a pie, with each series shown as a wedge in a different color, and with the width of the wedge equal to the percentage contribution of the series to the total for all series for that observation.

You may also change the **Graph Attributes** listed in the left column of the dialog box. Each attribute can be turned on or off by clicking to the left of the attribute. You may put a box around the graph, indent the graph within the box, have a legend identifying the series, draw vertical or horizontal grid lines, draw a horizontal line corresponding to the zero point on the vertical axis, and use color.

Bars and Lines deals with the appearance of the lines in a line graph, the bars in a bar graph, or the wedges in a pie chart. You may let EViews make the choice automatically, make the lines dashed, make the areas solid color or use patterns or hatching. Normally, you should let EViews make the choice. It will use color to distinguish lines on your color monitor and use patterns for the same graph when you print it on a monochrome printer.

Graph Scales lets you specify a single scale on the left, applying to all series, or a left scale for the first series and a right scale for the other series, set up so the series do not cross, or dual scales set up to show the full variation in the series, so that they will probably cross.

Graph Scaling gives you choices about the scales on the left and possibly the right. You can choose to let EViews choose the scaling to show the full variation of the series, but with round numbers on the scale. You can force the scale on the vertical axis to include zero. In addition to these two linear choices, you can choose a logarithmic scale. You can choose to have the data normalized, so that each series has a mean of zero and a standard deviation of one. Finally, you can choose manual scaling in place of automatic scaling by supplying the maximum and minimum values for the left and right scales.

Line Graphs lets you choose whether to show just lines connecting the data points for each series, or little symbols marking the data points, or both.

Line Patterns lets you choose the type of line used to plot the series. Click in the box and a menu of options will drop down.

Bar Graphs lets you choose where to put the date label or not to have a label.

Space Bars tells EViews to put a space between the bars in a bar graph between one observation and the next.

Scatter Diagram options are to connect the consecutive points with lines and to plot the line corresponding to the one-variable regression of the vertical axis series on the horizontal axis series.

Pie Graphs gives you the choice of putting dates under each of the pies.

Default Options

If you click **Options/Graphics Defaults** in the main EViews menu, you will see a big dialog box similar to the one just discussed. The choices you make in this box will determine the appearance of future graphs when you first create them.

Other ways to modify graphs (from **Help/Other ways to modify your graphs**)

You can double click on various other parts of graphs to change those parts.

If you double click on the vertical or horizontal axis, outside the graph area, you will get a dialog box that allows you to choose whether the axis is marked with ticks, numbers, or both. You can choose whether the ticks are outside the graph area, inside the area, or both. And you can press the font button to choose a font for the axis label.

If you double click on the legend, you will open a dialog box that allows you to modify the labels of the series (so that they are different from the names of the series), to specify the location of the legend in the graph, to specify whether to draw a box around the legend, and to specify the number of columns in the legend. You can press the Font button to change the legend font.

If you double click on the axis labels in a scatter plot, you can change their appearance. You can change the justification, specify a box, specify the position, edit the text of the labels, and change the font.

You can move the legend box in any graph and either axis title in a scatter plot by dragging it with your mouse.

With multiple graphs, you can drag each graph to a new position.

Freezing graphs (from **Help/Frozen Graphs**)

Often you will wish to preserve a graph so that it does not change as your work progresses. The [**Freeze**] button on the group window toolbar will make a frozen copy of your graph. The frozen version of the graph is an object that can be stored in your workfile. A frozen graph does not change when the data in the workfile change; it contains a copy of the data as they existed when you froze the graph.

You can change a frozen graph by all of the methods for graph views.

In addition, the toolbar on a frozen graph has two buttons that let you make changes that are not allowed in graph views, because they would be inconsistent with the changes that might occur if the data changed.

The [**AddText**] button opens a dialog box that allows you to put one or more lines of text anywhere on the graph: Enter the text you want displayed in the large edit field. Press ENTER after each line. The Justification buttons determine how multiple line labels will be aligned relative to each other. Check Text in Box if you want the label to be surrounded by a rectangle. Finally, Position indicates the initial placement of the text within your graph. At any time, you can change the position of the text by dragging it to the position you choose.

On a frozen graph's toolbar, the [**AddShade**] button shades intervals of observations. A common use is to mark recessions. The dialog box asks you to specify the starting and ending observations of the shaded area. You can use [**AddShade**] multiple times to shade a number of intervals. If your starting and ending observations are the same, [**AddShade**] will draw a sharp vertical line at that observation rather than shading an interval. With multiple graphs, first highlight the graph to which you wish to add shading before clicking [**AddShade**].

The [**Name**] button on the frozen graph's toolbar allows you to give a graph a name and thus save it in the workfile.

The [**Remove**] button removes an element of a frozen graph that you have previously highlighted by clicking with your mouse. For example, to remove text that you have placed on the graph, click on the text. A border will appear around the text. Press the [**Remove**] button and it will disappear. The same method can be applied to legends, scales, and shading.

*Creating multiple graphs (from **Help/Combining Graphs**)*

EViews will combine multiple graphs in a single window and print them on a single page. You can combine your own graphs, including all of modifications that you have made, into a single multiple graph.

All of the graphs you wish to combine must be named graph objects in the same workfile. From the workfile window select the graphs you wish to combine and then double click on one of them or choose **View/Open Selected** from the toolbar. Or you can choose **Quick/Show** and type the names of the graphs in the Show dialog. You will see all of the graphs displayed together as a single multiple graph. Your original graphs will be copied unchanged into the new multiple graph.

You can position the individual graphs by dragging them and you can add additional text or other modifications to the combined graph. You can apply a template to give all of the graphs the same appearance options. You can name the combined graph by pressing the [**Name**] button in its toolbar and it will be added to the workfile.

*Copying graphs to the clipboard (from **Help/Moving Graphs into Documents**)*

A useful way to print your graph is to incorporate it into a document in your Windows word processor and then use the word processor to print the entire document, including the graph. To do this, make sure the graph you want to copy is in the active window and click **Edit/Copy** on the EViews main menu. You will see a dialog box entitled **Copy Graph as Metafile**. You can

copy the graph to the Windows clipboard or to a disk file. You can request that the graph be in color and that its lines should be in bold. We recommend that you paste graphs in black-and-white unless you will be printing on a color printer.

If you copy a graph to the clipboard, you can switch to your word processor and paste the graph into your document. In WordPro, Word, or WordPerfect, the standard paste operation will give you a completely usable graph without any further adjustment. All of these programs have options for sizing, positioning, and adding further modifications to the graph. You can also paste graphs into drawing programs such as Paintbrush, which comes with Windows, and make further modifications before pasting into your word processor or other software.

A word-processing document will become bulky if it contains a large number of pasted graphs. A better procedure is to have EViews copy your graphs to disk files and then insert references to the files in your word processor. For example, in Microsoft Word, you can use **Insert/Picture** to do this. Check the Link to File box and uncheck the Save with File box.

Printing graphs

The [**Print**] button on the series or group window or frozen graph toolbar will print your graph on your printer. You can control the printing by using **File/Print Setup** on the main EViews menu. You will get a dialog box. The choices are mostly self-explanatory. Scale to page means scale the graph so that it takes up the specified percent of the space on the page. The Set Up button allows you to change settings specific for your printer.

Printing

Most views of objects can be printed by simply pressing the [**Print**] button of the appropriate toolbar. You can also choose **File/Print** or **Objects/Print** on the main EViews menu bar.

Redirecting printer output to a file

You may specify that the Print commands, that would normally send a view or procedure output to the printer, instead freeze the view as table or graph, or put it into an ASCII text file. Selecting **File/Print Setup** brings up a dialog. At the top you are given three choices for the Destination of printed output:

- ! **Printer**
- ! **Frozen Output**
- ! **Text File**

By default printing sends output to the printer.

If you choose **Frozen Output** you must also supply a base name. Each print command will then create a frozen object within the active workfile and name it with the base name followed by a number. For example, if you supply the base name of OUT, the first print command will generate

a table or graph named OUT1, the second print command will generate OUT2, etc. Whenever you wish you can select some or all of the frozen objects in the workfile window, open them for copy-and-paste to your word processor, print them (after tuning off output redirection), store them, and delete them.

If you choose **Text File**, you must supply a file name. Any instruction to print tables or text will then redirect the output to the text file. **Graphic views will continue to be sent to the printer.**

Batch Programs

A program is a file of EViews commands. A program is not an object within a workfile, but is instead an entity that is capable of controlling EViews objects and manipulating workfiles themselves.

The simplest program is just a list of commands. Execution of the program is equivalent to typing the commands one by one into the command window. But the real power of programs comes from their ability to execute commands more than once. For example, you can write a program with commands to deal with the data from one industry, and then have the program do the same operations for each of many industries.

One of the important benefits of programs is that they automatically document your project. If you want to know exactly how you got a set of results, it helps enormously if you have created a program that begins with creating a workfile and reading the raw data and ends with your final results.

If you have experience with any lower-level computer programming language, such as BASIC, you will find most of the tools for controlling programs quite familiar. The main novel feature of the EViews programming language is the ability to create object names by combining variables containing pieces of names.

Creating a Program

To create a new program, click **File/New/Program**. You will see a standard text editing window where you can type the lines of the program. You may also open the program window by typing **PROGRAM** in the command window, followed by a program name if you wish. Example:
PROGRAM FIRSTPRG

Example:

Let's create a program to create a workfile, load an Excel spreadsheet and transform some data. You will use two new EViews commands:

Create	Creates EViews workfile
Read	Reads text or spreadsheet files into EViews

Also, comments in EViews programs are preceded by the apostrophe character ‘.

For more info on these commands see **Help/Search/Create** and **Help/Search/Read**.

From the main menu, select **File/New/Program** to open the program window. In the program window type the following commands:

```
‘ program to create GA caseload data
‘ use chdir to change to the appropriate directory
create m 1973:07 2002:12
read(E,B2) 1
genr luxcase = log(uxcase)
genr dluxcase = d(luxcase)
genr trend = @trend(1973:07)
```

Click [**SaveAs**] and name the program caseload. That’s all there is to it!

Loading a Program

To load a program previously saved on disk, click on **File/Open**, change the **List File of Type** scrollbar to look for PROGRAM files, and click on the desired name. Alternatively, from the command line, you can just type **OPEN** followed by the program name. EViews will look for the program in the EViews working directory. You can include the full path name for your program file if it is in another directory.

Running or Executing a Program

To execute the program, either push the [**Run**] button or enter the **RUN** command in the command window. If you use the button, you will see a dialog box where you can supply arguments (described below), set the number of error messages after which you will halt execution, and choose between quiet and verbose modes. In verbose mode many messages are sent to the status line and the workfile window is continuously updated as objects are created and deleted. This information is suppressed in quiet mode. **Quiet mode** is the appropriate way to run a debugged program efficiently--quiet mode saves quite a bit of time the computer would otherwise spend writing on the screen.

In **interactive mode**, an error causes a dialog box or status line message. By default, a running program behaves the same way: errors will immediately terminate the program and display a message. If you type a number into the "Maximum number of errors before halting" field in the **RUN** dialog box, EViews will instead send error messages to a log window and continue executing until either the program finishes, or too many errors occur. There are some errors which are serious enough to halt a program regardless of how many have occurred; these errors will stop the program immediately.

Because **RUN** is a command, one program can execute another. Use **RUN(V)** to run a program

in verbose mode and **RUN(Q)** to run a program in quiet mode.

Stopping a Program

The F1 key halts execution of a program. It may take a few seconds for EViews to respond to the halt command. Programs will also stop when they encounter a **STOP** command.