

Basic Data Analysis Using ROOT

Getting Started on Windows 2000

Normally you'll run ROOT on a Linux system running X-windows. However, in room 118 we're using Windows machines. In order to access a Linux system, you'll have to do the following:

1. Log on to Windows.

Hit Ctrl-Alt-Del on the Windows 2000 login screen. The account name is "reu", the password is "kubrick". Select "AUTH_SERVER" from the pop-up menu at the bottom of the dialog box.

2. Start Exceed (an X-windows emulator) in passive mode.

Wait until Windows has finished starting up. Double-click on the "Exceed" icon on the Windows desktop. The only thing that should happen is that an "Exceed" box should appear on the Taskbar at the bottom of the screen. If you see anything else happen, call Bill Seligman over to re-configure Exceed to Passive mode for you.

3. Start Secure-CRT (a terminal communications program).

Double-click on the "SecureCRT" icon.

4. Log on to the Linux system assigned to your group.

You'll see a dialog box. Under "Hostname", type in the name of the server assigned to your working group (for ATLAS or eBubble it's "kolya"; for D0 it's "karthur"; for HiRes it's "nevis1"; for Muon it's "hogwarts"; for Neutrino or ZEUS it's "ladym"). Type your account name next to "Username", hit "Connect", type your password, and hit "OK". Do **not** check the box next to "Remember My Password"! If you see a dialog box entitled "New Host Key", just click "Accept & Save".

You should next start Netscape. Please: *Always start Netscape on the system you're on; never run it on a remote system.* Therefore, start Netscape by double-clicking the Netscape icon on the Windows desktop; move the SecureCRT window if necessary. Fill out the profile information if it asks for it. If Netscape doesn't work, use Internet Explorer.

Once Netscape is running, type the URL <http://root.cern.ch/> in the Location field. Bookmark this site. You will be returning to it often during this class.

Click on the SecureCRT window to make it active. You'll also be doing this a lot during this class; many of the graphics operations will "deselect" the main window.

Getting Started on Linux

On a Linux machine, you won't have to go through the above procedure. Just use your account name and password to login. Type "startx" to run X-windows, if it's not running already. Click **once** on the Netscape icon at the bottom of the screen to start Netscape.

A Brief Intro to Linux

If you're already reasonably familiar with Linux or UNIX in general, skip this page.

You can spend a lifetime learning Linux; I've been working with UNIX since 1993 and I'm still learning something new every day. The commands below barely scratch the surface. You may want to look at *UNIXhelp* <<http://www.geek-girl.com/Unixhelp/>>, *UNIX is a Four-Letter Word* <<http://www.msos.edu/~taylor/4ltrwrld/>>, and the usually out-of-date information at <<http://www.nevis.columbia.edu/software/>>.

To get help on any UNIX command: type "man <command-name>".

For example, to learn about the "ls" command (which lists the contents of a directory), type:

```
man ls
```

To copy a file: use the "cp" command.

For example, to copy the file "example.C" from the directory "~/seligman/root-class" to your current working directory, type:

```
cp ~/seligman/root-class/example.C .
```

In UNIX, a period (.) is short-hand for your current working directory.

To look at the contents of a text file: Use the "less" command.

This command is handy if you want to quickly look at a file without editing it. If the name of the command seems puzzling, it may help to know the "more" command also displays the contents of a text file, and the "less" command was created as more powerful version of "more". So to quickly look at the contents of file example.C, type:

```
less example.C
```

While "less" is running, type a space to go forward one screen, type "b" to go backward one screen, type "q" to quit, and type "h" for a complete list of commands you can use.

To edit a file: use the "emacs" command.

You will almost always want to add an ampersand (&) to the end of any "emacs" command; the ampersand means to keep the current window active and run the command as a separate process. So to edit the file example.C, type:

```
emacs example.C &
```

The "emacs" environment is complex, and you can spend a lifetime learning it. (Already I've spent two of your lifetimes, and the class has just started!) You can get around by using the mouse to move the cursor and look at the menus. As soon as you can (probably not during this class), you should take the Emacs tutorial by selecting it under the "Help" menu.

To create another command window: use the "xterm" command.

Again, you'll always want to put an ampersand (&) after this command. For today's class, you'll definitely want to have more than one command window to work with, so right now you should type:

```
xterm &
```

Starting ROOT (5 minutes)

Before you start using ROOT, you have to type the following command:

```
setup root
```

The command "setup root" sets some Unix environment variables and modifies your command and library paths. If you feel a need to remove these changes, use the command "unsetup root".

One of the variables that is set is \$ROOTSYS. This will be helpful to you if you're following one of the examples in the ROOT User's Guide. For example, if you're told to find a file in \$ROOTSYS/tutorials, you'll be able to do this only after you've typed "setup root".

You have to execute the "setup root" command only once, but you must do it each time you login to Linux. If you wish this command to be automatically executed when you login, you can add it to the .mycshrc file in your home directory.

To actually run ROOT, just type:

```
root
```

Make sure you've typed "xterm &" before you do this, because you're going to want to have at least one other command window to work with as you play with ROOT.

First you'll see the orange-and-red ROOT window appear on your screen. It will then disappear, and a brief "Welcome to ROOT" display will be written on your command window.

If you don't see the orange-and-red ROOT window appear briefly on your screen, it means that X-windows is not functioning properly on your system; tell Bill Seligman. However, this is unlikely, since the "xterm" command would have failed if X-windows were not working.

Click on the SecureCRT window to select it.

You can type "?" (or ".h") to see a list of ROOT commands... but you'll probably get more information than you can use right now. Try it and see.

The most important ROOT line command you need to know is how to quit ROOT. To exit ROOT, type ".q". Do this now, then start ROOT again, just to make sure you can do it.

Plotting a function (15 minutes)

This example is based on the first example in Chapter 2 of the ROOT Users Guide (page 15). I emphasize different aspects of ROOT than the Users Guide, and it's a good idea to go over both the example in the Guide and the one below.

Let's plot a simple function. Start ROOT and type the following at the prompt:

```
TF1 f1( "func1" , "sin(x)/x" , 0 , 10 )
f1.Draw( )
```

Note the use of C++ syntax to invoke ROOT methods. (Page 16 of the ROOT Users Guide has a discussion of this.)

If you have a keen memory (or you type ".h" on the ROOT command line), you'll notice that neither TF1 nor any of its methods are listed as commands, nor will you find a detailed description of TF1 in the Users Guide. The only place that the complete ROOT functionality is documented is on the ROOT web site.

Go to the ROOT web site at <http://root.cern.ch/> (did you remember to bookmark this site?), click on "Reference Guide", then on "The ROOT Class Categories", then on "Histogram", and finally on "TF1". Scroll down the page; you'll see the class methods, then a class description.

Get to know your way around this web site. You'll be coming back often.

Also note that when you executed "f1.Draw()", ROOT created a canvas for you named "c1". "Canvas" is ROOT's term for a window that contains ROOT graphics; everything ROOT draws must be inside a canvas.

Bring the window named "c1" to the front by left-clicking on it. As you move the mouse over different parts of the drawing (the function, the axes, the graph label, the plot edges) note how the shape of the mouse changes. Right-click the mouse on different parts of the graph and see how the pop-up menu changes.

Position the mouse over the function itself (it will turn into a pointing finger). Right-click the mouse and select "SetRange". Set the range to xmin=-10, xmax=10, and click "OK". Observe how the graph changes.

Let's start getting into a good habit by labeling our axes. Right-click on the x-axis of the plot, select "SetTitle", enter "x [radians]", and click "OK". Let's center that title: right-click on the x-axis again, select "CenterTitle", and click "OK".

Note that clicking on the title gives you a "TCanvas" pop-up, not a text pop-up; it's as if the title wasn't there. Only if you right-click on the axis can you affect the title. In object-oriented terms, the title and its centering are a property of the axis.

It's a good practice to always label the axes of your plots. Don't forget to include the units.

Do the same thing with the y-axis; call it "sin(x)/x". Select the "RotateTitle" property of the y-axis and see what happens.

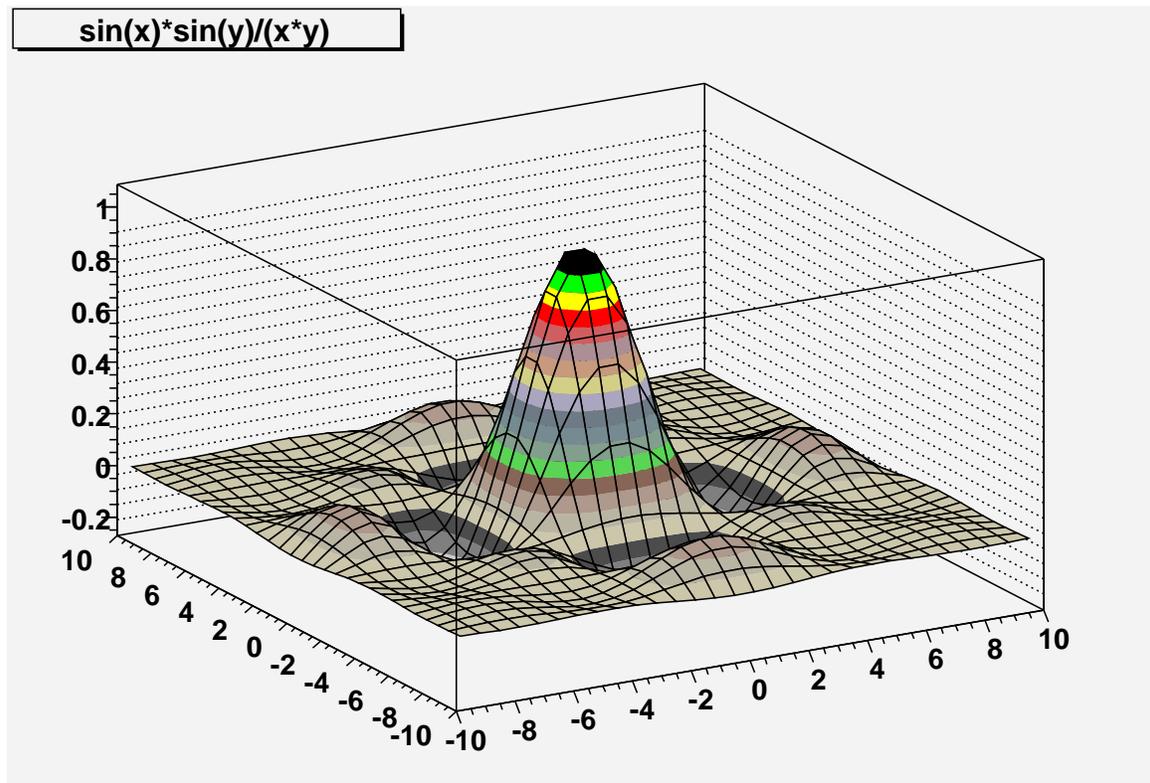
Plotting a function (continued) (15 minutes)

Move the cursor over the function itself, right-click, and select "DrawPanel". Click on "hist", then on "Draw". Now try clicking on "lego1", then on "Draw". Try the other lego-plot options. If you click on "Polar", you may not see much... but then try selecting "surface" as well.

If you "ruin" your plot, you can always quit ROOT and start it again. We're not going to work with this plot in the future anyway.

Exercise

Try to duplicate the following plot:



Hints: Look at the TF1 command on the last page. If class TF1 will generate a one-dimensional function, what class might generate a two-dimensional function? If TF1 takes a function name, formula, and x-limits in its constructor, what arguments might a two-dimensional function class use? What option on the DrawPanel might produce a smooth surface? Don't forget that the ROOT web site is available as a resource.

Working with Histograms (15 minutes)

Histograms are described in Chapter 3 of the ROOT Users Guide. You may want to look over this chapter later to get an idea of what else can be done with histograms other than what I cover in this class.

Let's create a simple histogram:

```
TH1F h1("hist1","Histogram from a gaussian",100,-3,3)
```

Let's look at what these arguments mean for a moment (and you should also look at the description of TH1F on the ROOT web site). The name of the histogram is "hist1". The title displayed when plotting the histogram is "Histogram from a gaussian". There are 100 bins in the histogram. The limits of the histogram are from -3 to 3.

Question: What should be the width of one bin of this histogram? Type the following to see if your answer is the same as ROOT thinks it is:

```
h1.GetBinWidth(0)
```

Note that we have to indicate which bin's width we want (bin 0 in this case), because you can define histograms with varying bin widths.

If you type

```
h1.Draw()
```

right now, you won't see much. That's because the histogram is empty. Let's randomly generate 10,000 values according to a distribution and fill the histogram with them:

```
h1.FillRandom("gaus",10000)
```

```
h1.Draw()
```

The "gaus" function is pre-defined by ROOT (see the TFormula class on the ROOT web site; there's also more on the next page). The default Gaussian distribution has a width of 1 and a mean of zero.

Note the histogram statistics in the top right-hand corner of the plot. Question (for those who've had statistics): Why isn't the mean exactly 0, or the width exactly 1?

Add another 10,000 events to histogram h1 with the FillRandom method (hit the up-arrows until you see "h1.FillRandom("gaus",10000)" again, and hit return). Click on the canvas. Note how the histogram updates immediately, without another "Draw" command.

Move the mouse over the line of the histogram; note that it changes into an arrow instead of a pointing finger. Right-click on the histogram and select "DrawPanel". Click on "E1: error/edges", then on "Draw".

The size of the error bars is equal to the square root of the number of events in that histogram bin. With the up-arrow key in the ROOT command window, execute the FillRandom method a few more times and click on the histogram again. Question: Why do the error bars get smaller?

You will often want to draw histograms with error bars. For future reference, you could have used the following command instead of the DrawPanel:

```
h1.Draw("e1")
```

Working with Histograms (continued) (15 minutes)

Right-click on the histogram and select "SetMarkerAttributes" (at the bottom of the menu). Click on the solid square, click on the blue tile, then click on "Apply". If you click on one of the circles on the bottom third of the panel and then click "Apply", you'll change the size of the markers; try it.

Let's create a function of our own:

```
TF1 myfunc("myfunc", "gaus", 0, 3)
```

The "gaus" (or gaussian) function is actually

$$P_0 e^{-0.5 \left(\frac{x - P_1}{P_2} \right)^2}$$

where P_0 , P_1 , and P_2 are "parameters" of the function. Let's set these three parameters to values that we choose, draw the result, then create a new histogram from our function:

```
myfunc.SetParameters(10., 1.0, 0.5)
myfunc.Draw()
TH1F h2("hist2", "Histogram from my function", 100, -3, 3)
h2.FillRandom("myfunc", 10000)
h2.Draw()
```

Note that we could also set the function's parameters individually:

```
myfunc.SetParameter(1, -1.0)
h2.FillRandom("myfunc", 10000)
```

• Optional note for advanced users:

In ROOT's TFormula notation, this would be "[0]*exp(-0.5*((x-[1])/[2])^2)"; "[n]" corresponds to P_n . I mention this so that when you become more experienced with defining your own parameterized functions, you can use a different formula:

```
TF1 myGaus("user",
  "[0]*exp(-.5*((x-[1])/[2])^2)/([2]*sqrt(2.*pi))")
```

This may seem cryptic to you now. Actually, it's just a gaussian distribution with a different normalization so that P_0 divided by the bin width becomes the number of events in the histogram:

```
myGaus.SetParameters(10., 0., 1.)
hist.Fit("user")
Double_t numberEquivalentEvents =
  myGaus.GetParameter(0) / hist.GetBinWidth(0)
```

Working with multiple plots (optional) (5 minutes)

If you're running short on time, you can skip this page (or any of the other optional pages).

We have a lot of different histograms and functions now, but we're plotting them all on the same canvas, so we can't see more than one at a time. There are two ways to get around this.

First, we can create a new canvas by selecting "New Canvas" from the File menu of our existing canvas; this will create a new canvas with a name like "c1_n2". Try this now.

Second, we can divide a canvas into "pads". On the new canvas, right-click in the middle and select "Divide". Enter $nx=2$, $ny=3$, and click "OK".

Click on the different pads and canvases with the **middle** button. Observe how the yellow highlight moves from box to box. The "target" of the Draw() method will be the highlighted box. Try it: select one pad with the middle button, then enter

```
h2.Draw( )
```

Select another pad or canvas with the middle button, and type:

```
myfunc.Draw( )
```

At this point you may wish that you had a bigger monitor!

Saving and printing your work (15 minutes)

By now you've probably noticed the "Save As" options under the "File" menu on the canvas. What do all these options mean?

- "Save as canvas.ps" will create a Postscript file named <canvas-name>.ps; for example, if you use this option from canvas c1, it will create "c1.ps". The image it creates will be sized to fit an entire page.
- "Save as canvas.eps" will create an Encapsulated Postscript file. This file is suitable for embedding into larger documents such as those created by LaTeX, StarOffice, and MS-Word. Note that ROOT does **not** create an image preview; the Linux command "ps2epsi" can do this for you.
- "Save as canvas.gif" will create a GIF image, suitable for embedding into web pages.
- "Save as canvas.C" will create a file with the ROOT commands necessary to re-create this canvas; more on this below.
- "Save as canvas.root" will create a native ROOT file with all the objects necessary to re-create this canvas; more on this below.

Select "Save as canvas.C" from one of the canvases in your ROOT session (the more complex the better). Let's assume for the moment that the file "c1.C" is created. In your xterm window, type

```
less c1.C
```

As you can see, this can be an interesting way to learn more ROOT commands. However, it doesn't record the procedure you went through to create your plots, only the minimal commands necessary to display them.

Next, select "Save as canvas.ps" from the same canvas; we'll print it later.

Finally, select "Save as canvas.root" from the same canvas. Let's assume the file is named "c1.root". Now quit ROOT with the ".q" command, and start it again.

To re-create your canvas from the ".C" file, use the command

```
.x c1.C
```

This is your first experience with a ROOT "macro", a stored sequence of ROOT commands that you can execute at a later time. One advantage of the ".C" method is that you can edit the macro file, or cut-and-paste useful command sequences into macro files of your own.

Quit ROOT and print out your Postscript file with the command

```
qpr -Pqms1 c1.ps
```

This may be point at which you'll notice that the default background color for ROOT plots is not pure white. You can change the background by right-clicking on a canvas and selecting "SetFillAttributes"; you'll have to do this in the regions both outside and inside the plot.

The ROOT browser (5 minutes)

The ROOT browser is a useful tool, and you may find yourself creating one at every ROOT session. Read pages 25–26 of the ROOT Users Guide to find out how to make ROOT start a new browser automatically each time you start ROOT.

One way to retrieve the contents of file "c1.root" is to use the ROOT browser. Start up ROOT and create a browser with the command:

```
TBrowser tb
```

In the right-hand pane, double-click on the folder with the same name as your home directory. Scroll through the list of files. You'll notice special icons for any files that end in ".C" or ".root". If you double-click on a file that ends in ".C", ROOT will assume the file contains a ROOT macro and interpret the contents. Try this on "c1.C", then close the canvas window.

Now double-click on "c1.root". Nothing will appear to change. Now click on the "ROOT Files" folder in the left-hand pane; this is the list of files currently opened by ROOT. Double-click on "c1.root" in the right-hand pane, then double-click on "c1;1".

What does "c1;1" mean? You're allowed to write more than one object with the same name to a ROOT file (this topic is part of an optional lesson later in this class). The first object has ";1" put after its name, the second ";2", and so on. You can use this facility to keep many versions of a histogram in a file, and be able to refer back to any previous version.

At this point, saving a canvas as a ".C" file or as a ".root" file may look the same to you. But these files can do more than save and re-create canvases. In general, a ".C" file will contain ROOT commands and functions that you'll write yourself; ".root" files will contain complex objects such as n-tuples.

Fitting a histogram (15 minutes)

I created a file with a couple of histograms in it for you to play with. Switch to your xterm window and copy this file into your directory:

```
cp ~seligman/root-class/histogram.root .
```

Go back to your browser window. (If you've quit ROOT, just start it again and start a new browser.) Click on the folder in the left-hand pane with the same name as your home directory. If you don't see "histogram.root", select "Refresh" from the "View" menu.

Double-click on "histogram.root", click on "ROOT Files" in the left-hand pane, then double-click on "histogram.root" in the right-hand pane. You can see that I've created two histograms with the names "hist1" and "hist2". Double-click on "hist1".

You can guess from the x-axis label that I created this histogram from a gaussian distribution, but what were the parameters? In physics, to answer this question we typically perform a "fit" on the histogram: you assume a functional form that depends on one or more parameters, and then try to find the value of those parameters that make the function best fit the histogram.

Right-click on the histogram and select "FitPanel". Click on "gaus", then click on "Fit". You'll see two changes: A function is drawn on top of the histogram, and the fit results are printed on the ROOT command window.

Interpreting fit results takes a bit of practice. Recall that a gaussian has 3 parameters (P_0 , P_1 , and P_2); these are labeled "Constant", "Mean", and "Sigma" on the fit output. ROOT determined that the best value for the "Mean" was 5.96 ± 0.03 , and the best value for the "Sigma" was 2.47 ± 0.02 . Compare this with the Mean and RMS printed in the box on the upper right-hand corner of the histogram. Statistics questions: Why are these values almost the same as the results from the fit? Why aren't they identical?

On the canvas, select "Show Fit Parameters" from the "Options" menu. Click on "Fit" on the FitPanel again. Just as a check, click on "landau" on the FitPanel and click on "Fit" again; then click on "expo" and fit again.

It looks like of the three choices (gaussian, landau, exponential), the gaussian is the best functional form for this histogram. Take a look at the "Chi2 / ndf" value in the statistics box on the histogram ("Chi2 / ndf" is pronounced "kie-squared per degrees of freedom"). Do the fits again, and observe how this number changes. Typically, you know you have a good fit if this ratio is about 1.

Fitting a histogram (continued) (15 minutes)

Go back to the browser window and double-click on "hist2". Uh-oh; this doesn't look like a gaussian. Right-click on the histogram and select "FitPanel" (be careful not to re-use the FitPanel from "hist1"). Click on "gaus" and then on "Fit". Uggh --- that's a terrible fit! You can try the landau and exponential functions, but they won't work much better.

You've probably already guessed by reading the x-axis label that I created this histogram from the sum of two gaussian distributions. But you've probably also noticed a button on the FitPanel labeled "user"; this is for fitting to a user-defined function.

In order to use this button, you have to define a function that has the name "user".

Define a user function with the following command:

```
TF1 func ( "user" , "gaus(0)+gaus(3)" )
```

Note that the internal ROOT name of the function has to be "user", but not the function object itself.

What does "gaus(0)+gaus(3)" mean? You already know that the "gaus" function uses three parameters. "gaus(0)" means to use the gaussian distribution starting with parameter 0; "gaus(3)" means to use the gaussian distribution starting with parameter 3. This means our user function has six parameters: P_0 , P_1 , and P_2 are the "constant", "mean", and "sigma" of the first gaussian, and P_3 , P_4 , and P_5 are the "constant", "mean", and "sigma" of the second gaussian.

Now try to do a fit by going to the FitPanel, clicking on "user", then clicking on "Fit".

If you look at the ROOT command window, you'll see that all six parameters have the value "nan", which means "Not A Number." For all but the simplest fits, ROOT needs to have some starting values for its fit parameters.

Let's set the values of P_0 , P_1 , P_2 , P_3 , P_4 , and P_5 :

```
func.SetParameters(5.,5.,1.,1.,10.,1.)
```

Then click on "Fit" on the FitPanel.

The results are not much better. This is because I deliberately picked a poor set of starting values. Let's try a better set:

```
func.SetParameters(5.,2.,1.,1.,10.,1.)
```

These starting values don't look much different, but try to fit the histogram again.

These simple fit examples may leave you with the impression that all histograms in physics are fit with gaussian distributions. Nothing could be further from the truth. I'm using gaussians in this class because they have properties (mean and width) that you can determine by eye.

Chapter 5 of the ROOT Users Guide has a lot more information on fitting histograms, and a much more realistic example.

If you want to see how I created the file histogram.root, go to the xterm window and type:

```
less ~seligman/root-class/CreateHist.C
```

Saving your work, part 2 (optional) (15 minutes)

So now you've got a histogram fitted to a complicated function. Try "Save as canvas.C" from the "File" menu, and examine the result with

```
less c1.C
```

I know it looks like a bunch of C++ commands. But if you look carefully, you'll see that the user function you just created is **not** re-created in the macro.

If you were to use "Save as canvas.root", quit ROOT, restart it, then load canvas "c1;1" from the file, you'd get your histogram back with the function superimposed... but it's not obvious where the function is or how to access it now.

What if you want to save your work in the same file as the histograms you just read in? You can do it, but not by using the ROOT browser. The browser will only open files in read-only mode. To be able to modify a file, you have to open it with ROOT commands.

Try the following: Quit ROOT (note that you can select "Quit ROOT" from the "File" menu of the canvas or the browser). Start ROOT again, then modify "histogram.root" with the following commands:

```
TFile file1("histogram.root", "UPDATE")
```

It is the "UPDATE" option that will allow you to write new objects to "histogram.root".

```
hist2.Draw()
```

For the following two commands, try hitting the up-arrow key until you see them again. ROOT stores the last 80 or so ROOT commands you've typed in the file ".root-hist" in your home directory, and let's you re-use them with the arrow keys.

```
TFl func("user", "gaus(0)+gaus(3)")
```

```
func.SetParameters(5., 2., 1., 1., 10., 1.)
```

The following command is the same as clicking on "user" and "Fit" on the FitPanel:

```
hist2.Fit("user")
```

Now you can do what you couldn't before: save objects into the ROOT file:

```
hist2.Write()
```

```
func.Write()
```

You should close the file to make sure you save your changes:

```
file1.Close()
```

Quit ROOT, start it again, and use the ROOT browser to open "histogram.root". You'll see a couple of new objects: "hist2;2" and "user;1". Double-click on each of them to see what you've saved.

Chapter 9 of the ROOT Users Guide has more information on using ROOT files.

Dealing with PAW files (optional) (5 minutes)

Suppose someone gives you a file that contains n-tuples or histograms that were created with PAW, HBOOK, or CERNLIB (actually, to first order these are three different names for the same thing). How do you read these files using ROOT?

The answer is that you can't, at least not directly. You must convert these files into ROOT format using the command "h2root".

For example, if someone gives you a file called "testbeam.hbook", you can convert it with the command

```
h2root testbeam.hbook
```

This creates a file "testbeam.root" that you can open in the ROOT browser.

There is no simple way of converting a ROOT file back into PAW/HBOOK/CERNLIB format. You generally have to write a custom program with both FORTRAN and C++ subroutines to accomplish this task.

Note that the "h2root" command is set up (along with ROOT) with the command

```
setup root
```

that you type when you log in. If you accidentally type "h2root" (or "root") before you set up ROOT, you'll get the error message:

```
h2root: Command not found
```

You can get more information about "h2root" by using a special form of the "man" command:

```
man $ROOTSYS/man/h2root.1
```

Accessing variables in ROOT NTuples/Trees (15 minutes)

I've created a sample ROOT n-tuple in `~seligman/root-class/tree.root`. However, I don't want you to copy this file into your home directory; typical n-tuples are stored in large files and it would take too much disk space to make multiple copies.

Start fresh: quit ROOT and start it again. Start a new browser with the command

```
TBrowser b
```

Select "Open..." from the "File" menu. In the pop-up menu at the top of the dialog box, select the "Root" line (the one with an icon that looks like a disk drive).

You may be confused by the different meanings of "Root". Unfortunately, the name of this analysis package is also the name the super-user account in UNIX, and is also the term used for the base directory of the operating system. In this case we want to move to the base directory of the Linux system.

In the "File name" text box, type "`~seligman/root-class/tree.root`" and click "Open".

In the left-hand pane of the browser, click on "ROOT files" and you'll see that "tree.root" is open. Double-click on "tree.root" in the right-hand pane to look at the contents of the file. There's just one object inside: "tree1", a ROOT tree (or n-tuple) with 100,000 simulated physics events.

Actually, there's little or no real physics associated with the contents of this tree. I created it solely to illustrate ROOT concepts, not to demonstrate real physics with a real detector.

Right-click on the "tree1" icon, and select "StartViewer". You're looking at a TreeViewer, an interface to help you analyze ROOT trees. Focus on the second column in the right-hand pane of the TreeViewer; these are the variables stored in the tree.

In this overly-simple example, an imaginary particle is travelling in a positive direction along the z-axis with energy "ebeam". It hits a target at $z=0$, and travels a distance "zv" before it is deflected by the material of the target. The particle's new trajectory is represented by "px", "py", and "pz", the final momenta in the x-, y-, and z-directions respectively. The variable "chi2" represents a confidence level in the measurement of the particle's momentum.

Did you notice what's missing from the above description? One important thing that's missing is the units; for example, I didn't tell you whether "zv" is in millimeters, centimeters, inches, yards, etc. Such information is not usually stored inside an n-tuple; you have to find out what it is and include the units in the labels of the plots you create. For this example, assume that "zv" is in centimeters (cm), and all energies and momenta are in GeV.

Double-click on the names of the variables and see what happens.

There is a "Help" menu on the upper right-hand side of the TreeViewer menu. Unlike the Help menus on the browser and canvas menu, which are only marginally informative, the Help offered on the TreeViewer is genuinely useful.

By the way, the variable "event" is just the event number (it's 0 for the first event, 1 for the second event, 2 for the third event... 99999 for the 100,000th event).

Correlating variables: scatterplots and expressions (15 minutes)

Left-click on a variable and hold the mouse down. Drag the variable next to the blue curly "X" in the first column, over the word "-empty-", and let go of the button. Now select a different variable and drag it over next to the curly "Y". Click on the scatterplot icon in the lower left-hand corner of the TreeViewer (it's next to a button labeled "STOP", you may have to move the TreeViewer window).

This is a scatterplot, a handy way of observing the correlations between two variables. Be careful: it's easy to fall into the trap of thinking that each (x,y) point on a scatterplot represents two values in your n-tuple. In fact, the scatterplot is a grid and each square in the grid is randomly populated with a density of dots that's proportional to the number of values in that grid.

Drag different pairs of variables to the "X" and "Y" boxes and look at the scatterplots. Do you see any correlations between the variables?

If you just see a shapeless blob on the scatterplot, the variables are likely to be uncorrelated; for example, plot "px" versus "py". If you see a pattern, there may be a correlation; for example, plot "pz" versus "zv". It appears that the higher "pz" is, the lower "zv" is, and vice versa. Perhaps the particle loses energy before it is deflected in the target.

You can also create expressions that are functions of the variables in the tree. Double-click on one of the "E()" icons that has the word "-empty-" next to it. In the dialog box, type " $\sqrt{px*px+py*py}$ " in the box under "Expression", and type "pt" in the box under "Alias". Then click on "Done". Now double-click on the word "~pt" in the TreeViewer.

When you're typing in the expression, you don't have to type the name of any variable in the tree. You can just click on the name in the TreeViewer. By the way, "pt" is the transverse momentum of the particle, that is, the component of the particle's momentum that's perpendicular to the z-axis.

Let's do this again to calculate theta, the angle the particle makes with the z-axis. Double-click on a different "E()" icon with "-empty-" next to it. Type " $\text{atan}(\sim\text{pt}/\text{pz})$ " under "Expression", and "theta" under "Alias". Click "Done", then double-click on "~theta".

After an expression is no longer empty, you can't double-click on it to edit it; that will just cause the expression to be plotted. To edit an existing expression, right-click on it and select "EditExpression."

Note that you can have expressions within expressions (such as "~pt" in the definition of "~theta"). All expressions that you create must have names that begin with a tilde (~), and the expression editor will enforce this. A common error is to forget the tilde when you're typing in an expression; that's the reason why it can be a good idea to insert a variable or an alias into an expression by clicking on it in the TreeViewer.

By the way, the units of theta in this example are radians.

Correlating variables: cuts (15 minutes)

Let's create a "cut" (a limit on the range of a variable to be plotted). Edit another empty expression and give it the formula " $z_v < 20$ " and the alias "zcut".

Note how the icon changes in the TreeViewer. ROOT recognizes that you've typed a logical expression instead of a calculation.

Drag "~zcut" to the scissor icon. Double-click on "zv" to plot it. Double-click on some of the other variables and look at the "Nent" in the statistics box of the histograms; the z-cut affects all the plots, not just the plot of "zv".

Double-click on the scissor icon to turn off the cut; note the change in the scissor icon. Double-click on the icon again to turn the cut back on.

Now edit "~zcut" by right-clicking on it and selecting "EditExpression". Edit the expression to read " $z_v < 20 \ \&\& \ z_v > 10$ " and click "Done." Plot "zv". Has the cut changed? Now drag "~zcut" to the scissors and plot "zv" again.

A note for advanced users: A "cut" is actually a weight that ROOT applies when filling a histogram; a logical expression has the value 1 if true and the value 0 if false. If you want to fill a histogram with weighted values, use an expression for the cut that corresponds to the weight. For example, a cut of " $1/e$ " will fill a histogram with each event weighted by $1/e$; a cut of " $(1/e) * (\text{sqrt}(z) > 3.2)$ " will fill a histogram with events weighted by $1/e$, for those events with $\text{sqrt}(z)$ greater than 3.2.

In a complex analysis, you may have large number of expressions and cuts in your TreeViewer. You can save them so you don't have to type them all again. From the "File" menu, select "Save source". ROOT will create a file named "treeviewer.C".

If you want to look at the results, switch to your xterm window and type:

```
less treeviewer.C
```

Quit ROOT and start it again. Type

```
.x treeviewer.C
```

ROOT will open the tree file for you, and set up the TreeViewer with the same expressions and cuts.

If you are analyzing more than one tree, you may want to save multiple TreeViewers. A simple way to do this is to rename the treeviewer.C file after you save it. The Linux command to rename a file is "mv"; for example:

```
mv treeviewer.C dileptonviewer.C
```

A simple analysis example (15 minutes)

Let's walk through a simple example of what can be done with the tools you've learned so far. An actual physics analysis is much more complex, but this may give you a "feel" for what's involved.

Make a histogram of "chi2".

There's a peak around 1, but the x-axis extends far beyond that, up to $\text{chi2} > 18$. Evidently there are some events with a large chi2 , but not enough of them to show up on the plot.

Make a scatterplot of "chi2" versus "ebeam".

On the scatterplot, we can see a dark band that represents the main peak of the chi2 distribution, and a scattering of dots that represents a group of events with anomalously high chi2 .

The chi2 represents a confidence level in reconstructing the particle's trajectory. If the chi2 is high, the trajectory reconstruction was poor. It would be perfectly acceptable to apply a cut of " $\text{chi2} < 1.5$ ", but let's see if we can correlate a large chi2 with anything else.

Make scatterplots of "chi2" versus the other variables, including the expressions. Does anything stand out?

Take a careful look at " chi2 " vs. " $\sim\text{theta}$ ". It looks like all the large- chi2 values are found in the region $\text{theta} > 0.15$ radians. It may be that our trajectory-finding code has a problem with large angles. Let's put in both a theta cut and a chi2 cut to be certain we're looking at a sample of events with good reconstructed trajectories.

Create a new cut " $\sim\text{cutChi2}$ " with an expression of " $\text{chi2} < 1.5$ ". Create another new cut " $\sim\text{cutTheta}$ " with an expression of " $\sim\text{theta} < 0.15$ ". Now create a third cut " $\sim\text{cutAll}$ " with an expression of " $\sim\text{cutChi2} \ \&\& \ \sim\text{cutTheta}$ ". Drag " $\sim\text{cutAll}$ " to the scissor icon and plot " chi2 ".

There's nothing wrong with just creating one cut " $\text{chi2} < 1.5 \ \&\& \ \sim\text{theta} < 0.15$ ", but I wanted to illustrate that you can combine cut expressions in the same way that you can combine any other expression.

I must confess: I cheated when I pointed you directly to theta as the cause of the chi2 problem. I knew this because I wrote the program that created the tree. If you want to look at this program yourself, go to the xterm window and type:

```
less ~seligman/root-class/CreateTree.C
```

By the way, you've made a lot of plots, but you've probably only looked at them one at a time. Don't forget that you can view multiple plots in a couple of ways: You can select "New canvas" from the "File" menus of the browser, any canvas, or the TreeViewer. You can select one canvas, right-click on it, select "Divide", and enter the number of horizontal and vertical divisions; don't forget to use the middle button to select the pad or canvas you wish to draw to.

There is a much more detailed description of trees and how to analyze them in Chapter 10 of the Users Guide.

A technical problem (optional) (10 minutes)

Make a histogram of "zv". That looks like an exponential distribution; let's try to do a fit. Right-click on the histogram, select "FitPanel", click on "expo", then on "Fit".

The ROOT command window displays the result of the fit, but no function was superimposed on the histogram. What happened?

By default, when you make a plot using the TreeViewer, it's written to a histogram named "htemp". You can see this name in the TreeViewer; it's in the "Histogram" field in the status bar just below the menu bar.

"htemp" is a handy storage area for histograms, but it has a couple of problems: it gets overwritten all the time, and it has some simplifying properties. Among those simple properties is that you cannot store or display an associated function with the histogram.

What you can do is create a new histogram with whatever properties you like.

In the "Histogram" field of the TreeViewer, type a new name ("zhist", for example). Double-click on "zv" to make a histogram; this time a new histogram named "zhist" will be created. Right-click on the new histogram, select the FitPanel, and fit to an exponential distribution.

Now, before you do anything else: double-click on "chi2".

Oops -- neither the x-axis nor the plot labels changed. Your new histogram "zhist" is not the same thing as ROOT's "htemp". If you supply a new histogram via the TreeViewer, you must remember to change it back to "htemp" or a different histogram before making new plots.

By the way, you don't have to let the TreeViewer create the histogram for you. If you want to specify the properties of the histogram yourself (for example, the bin limits), you can create the histogram using a ROOT command; for example:

```
TH1F zhist("zhist","z vertex distribution",100,10.,20.)
```

and then put "zhist" in the TreeViewer.

Analysis Macros (optional, for C++ programmers)

The TreeViewer can get you started, but it's not enough for serious data analysis. At some point, you need to leave the graphic user interface behind and start working with ROOT macros.

ROOT makes the procedure relatively painless, but it's still a bit complex for the beginning student. The following section is for both you and your supervisor. Typically, your supervisor will prepare the ROOT/C++ code for you; all you have to do is run it and make occasional changes.

The first step to have ROOT write the skeleton of an analysis class for you. This is done with MakeClass. This example assumes you're still working with my "tree1" example.

If you don't already have the ROOT tree open, open it with the following command:

```
TFile myFile( "~seligman/root-class/tree.root" )
```

Now create an analysis macro for "tree1" with MakeClass:

```
tree1.MakeClass( "Analyze" )
```

Switch to the xterm window and examine the files that were created:

```
less Analyze.h
```

```
less Analyze.C
```

Unless you're familiar with C++, this probably looks like gobbledy-gook to you. (I know C++, and it looks like gobbledy-gook to *me*.) Fortunately, there are only two routines that should be modified: Loop and possibly Cut.

As an example, I've prepared two files for someone familiar with C++ to compare with the code that MakeClass generates. You can see the differences with the commands:

```
diff Analyze.h ~seligman/root-class/AnalyzeExample.h
```

```
diff Analyze.C ~seligman/root-class/AnalyzeExample.C
```

After an analysis class has been prepared, you use it by loading it into ROOT, creating an object of that class, and invoking its Loop() method. The class will open the tree file for you if it's not already open.

Try this: copy my example into your directory with

```
cp ~seligman/root-class/AnalyzeExample.* .
```

and enter the following lines in ROOT:

```
.L AnalyzeExample.C
```

```
Analyze a
```

```
a.Loop( )
```

It may help to remember that while the names of my files begin with "AnalyzeExample", the name of the class defined in those files is just "Analyze".

If you make any changes to an analysis macro, you have to load it again with ".L".

Feynmann diagrams (optional)

This page is meant for physicists who are going over these notes; it's unlikely that summer students will have to worry about making Feynmann diagrams.

Let's start fresh: quit ROOT and start it again. Create a new canvas with the command:

```
TCanvas diagram("diagram", "Feynmann diagram")
```

Now select "Editor" from the "Edit" menu.

Most of the options are self-explanatory. For example, to create a line, click on the "Line" button in the Editor. Click on the canvas, hold the button down, and drag the mouse to draw the line.

Once you've created a line or other graphical element, you can edit it... but it's not always clear where you have to click or drag the mouse. You have to carefully watch how the shape of the cursor changes as you move it over your drawing. In particular, Ellipses and CurlyArcs can be tricky to work with; their "anchors" are located outside the ellipse.

Try right-clicking on the shapes to see what drawing options you have. You can change a "curly" line (gluon-like) to a "wavy" line (photon-like) by right-clicking on it and selecting "SetWavy".

For text, click on the "Text/Latex" button, then click on the canvas where you'd like the text to go. The cursor will change into a question mark. Type your text, then hit the enter key. Try entering the text " q_1 " and see what happens; ROOT contains a subset of Latex. For more information, see the section on "Text and Latex Mathematical Expressions" in Chapter 8 of the Users Guide.

Once you've got your diagram, you should save it with "Save as canvas.root" or "Save as canvas.C" under the "File" menu (so you can edit it later), and with "Save as canvas.eps" (so you can include it in your article).